A Compendium of Problems in Transport Phenomena

David T. Leighton, Jr.

Department of Chemical and Biomolecular Engineering
University of Notre Dame

Diffusive Press
https://www.diffusivepress.org

Published by Diffusive Press
https://www.diffusivepress.org

© 2025 David T. Leighton.

This Compendium may be freely distributed for educational, non-commercial purposes. A free PDF version is available from ***https://www.diffusivepress.org****. The LaTeX source files used to generate this book—including templates for exams, homework, and generative AI prompts—are available at no charge to faculty by request from Diffusive Press.*

About the Cover: The front cover image shows the second cascade of the breakup of a toroidal miscible drop at low Reynolds numbers, the dimensional analysis of which forms one of the problems of this Compendium. This image was captured during a demonstration from the companion volume, *A Teacher's Guide to Classroom Demonstrations in Transport Phenomena*, also available from Diffusive Press.

Introduction

Faculty often find the creation of homework or exam questions to be a difficult and time consuming task - or at least I always found it to be so. After teaching at Notre Dame for some four decades, however, I have accumulated a large quantity of such questions. Rather than let this material "go to waste" I decided to extract these questions from old exams and compile them in a way in which they could be more easily accessed by me and others, thus this compendium was created. Originally all of the files were written using early versions of Microsoft Word (and trying to open a file written in Word 5.1 on a modern computer is a real challenge!), however (fortunately) I had the PDF files - although these are scarcely more useful in creating a new exam. With the advent of AI, it is now possible to recover old problem sets and convert PDFs into editable LaTeX files. LaTeX was chosen for its ability to seamlessly integrate both technical line drawings (via the `TikZ` package) and the many equations required for teaching Transport Phenomena.

For compiling these LaTeX files back into PDFs, I use the `TeXShop` application on macOS. TeXShop is free and easy to use, and even for a LaTeX novice like me, the combination of TeXShop for compiling and ChatGPT for debugging has proven remarkably simple and effective. I downloaded the PDF files, fed the text into Chat, and (with appropriate prompts) got the LaTeX scripts back which could be compiled in TeXShop. The prompt is important for consistency, but even that was produced by Chat after reducing several problem files and identifying issues. The prompts that were most effective (and which required subsequent editing) are also included in this Compendium.

This compendium is designed to work seamlessly with TeXShop, but the LaTeX source files are portable and can be compiled on *any* platform using other LaTeX editors, such as `TeXworks`, `TeXstudio`, or `Overleaf`. All of these are free to download and use. While you can browse the problems directly in the PDF version of the compendium for inspiration, practical use requires obtainiing the file `transportproblems.zip` available to faculty free of charge by request from `https://diffusivepress.org/`. This archive contains all the `.tex` files used to generate this document, including the master script `acompendium.tex`, which can be compiled with any standard LaTeX toolchain. Note that because of how LaTeX handles table of contents generation, you must typeset the document twice to produce a complete table of contents and a third time to get the pagination correct.

In theory it is possible to use these LaTeX files without knowing or using LaTeX iitself, as AI agents can convert the files from LaTeX back into an editable word document, even preserving the equations in an editable MS Word format and LaTeX TikZ figures as images. This is in theory anyway, and what Chat will tell you it can do: it certainly does -not- work on the free version of ChatGPT-4o. AI tools are rapidly improving, and as this is a recurring issue it may well be fixed in the future, however for now I recommend learning the rudiments of LaTeX. With AI assistance this is actually a fairly painless process.

While there are a very large number of problems here, a substantial number have only minor variations. There are only so many ways that you can construct a question regarding laminar flow down an incline, for example, and I think that I have used all of them over the course of four decades. I have removed all true duplicates (like many faculty I do reuse problems for closed book exams) however I have left those with minor variants included in this Compendium. I often change things "a little bit" so that students learn the method rather than just memorizing an answer.

Many of the problems presented here have solutions available on the web via `https://diffusivepress.org/`. While efforts have been made to keep this stable, that is always a problem with web based resources. After some time, it would be expected that, even if the site disappears, it should still be visible on the "wayback machine" (`https://web.archive.org`). The video narration courseware and solutions associated with this compendium are more problematic as those are currently served via Panopto through the University of Notre Dame, and can vanish at any time (and are certain to vanish eventually).

Use in an AI Environment

The rapid evolution of the AI Chatbots such as ChatGPT is greatly changing the way in which we teach. The problems presented here are highly amenable for use with an AI, as the LaTeX scripts can simply be pasted in and absorbed by the AI, and a LaTeX script of the AI's solution can be cut and pasted into TeXShop for viewing. This can all be done using the free version (at the time of this writing), although the number of problems it can solve without plan upgrade is very limited. In general, ChatGPT-4o does fairly well - some problems it gets completely right (short answer (usually, as it can access the web), dimensional analysis, conservation of momentum). Others it struggles more with (scaling analysis, lubrication, boundary layers) but does about as well as top students do on a timed closed-book exam. Even there, with some help it can (sometimes) converge to the correct answer. It has the most trouble with problems where there is less guidance, although since it has access to sources on the web it may get it right. As an example, if you simply ask it to calculate the dispersion of a slug traveling through a capillary it gets the right answer the first time (correct Taylor dispersion result, etc.). If you then ask it to solve the equivalent problem for a large aspect ratio microchannel it gives the parallel-plate result (which is incorrect, as dispersion is actually dominated by the side walls in this geometry). Pointing out that the dispersion is much higher than Chat's answer yields additional equations - and even a few correct literature citations - but it was unable to come up with the correct result. After a few tries the current (free) version of ChatGPT-4o times out.

So what are the implications of this in teaching? Clearly we need to educate students on how to effectively employ Chat or other AI tools in the same way engineers were once taught how to use a slide rule (I still have mine). We also need to educate our students sufficiently in the fundamentals so that they can recognize when Chat is giving incorrect answers - because they are often "reasonable" and "convincing", but also wrong - the sorts of answers which

would yield significant partial credit on a closed book exam, but be very bad if used in engineering practice. One possible strategy is to give students one of these LaTeX questions (that Chat gets wrong) and then as their task figure out where Chat is making the error and how it can be persuaded to get the correct answer (if it can). The interaction with Chat can be recorded and turned in for evaluation. Another approach is to assign the LaTeX files of different problems to small groups of students, have them use Chat to generate LaTeX solutions (which requires them to know when Chat is getting it wrong!) and then exchange with other groups to have their LaTeX output graded for accuracy, readability, and educational value. The best of these could be assembled into a companion solutions manual at the end of the semester. At the other extreme is to make it more difficult for students to even use Chat to solve a homework assignment, forcing them to work through it themselves. Any PDF can be converted into an image which makes rendering by an AI very difficult (although it does not prevent students from typing in the question themselves, of course). The process of rasterizing a PDF depends on the platform used (e.g., Mac or PC), but Chat itself will provide the instructions on methods of doing this. As AI rapidly improves it will be even more important to keep our teaching methods up to date so that our students are prepared for this new and challenging environment.

Subsequent to the distribution of the first version of this Compendium, OpenAI released ChatGPT-5. In their press conference CEO Altman commented that ChatGPT-4o was like having a good undergraduate on your desktop, while having ChatGPT-5 (with its improved reasoning skills) was like having a PhD expert on your desktop. Testing out the more complicated of the problems presented here with the new version of Chat shows that this is not far from the truth. With ChatGPT-4o solutions to these problems can be obtained via the "prompt chain" approach, where intermediate steps are output as LaTeX scripts, checked for accuracy, and then assembled. This process is tedious, but effective (provided you can identify the errors!). The same problems input into ChatGPT-5 with minimal prompt instructions (simply asking it to act as an agent, solve it step-by-step, be sure that you have the correct answer, and provide output as a LaTeX script, followed by the LaTeX script of the problem) yielded the solutions which escaped ChatGPT-4o. When queried about aspects of the provided solution, ChatGPT-5 would provide detailed (and correct) explanations and derivations of intermediate steps. What the implications of this improvement are to our teaching methods are as yet unclear, and of course we can expect these LLM AI models to undergo further rapid improvement.

Directory Structure

The problems in this Compendium are taken from the courses CBE 30355 Transport Phenomena I (focusing on fluid mechanics) and CBE 30356 Transport Phenomena II (focusing on heat and mass transport). These are Junior level departmental courses. The problems are taken from several aspects of the courses. The bulk of the problems are drawn from the in-class closed book exams. These are given the file names problem0xxx.tex for 355 and problem 1xxx.tex for 356. Prior to the pandemic many of the exams in 355 had "short answer" questions. These are compiled in eight files saxxx.tex, broken down into rough categories. Unsorted collections of these short answer questions (as they appeared as exam problems) as well as additional pump curve analysis questions are collected in a separate PDF which is generated from the LaTeX file `ashortanswercollection.tex` included in the `transportproblems.zip` file.

The homework problems for each class have file names hw0aax.tex (where aa is the problem set number) and hw1aax.tex. As a result of changes forced by the pandemic the short answer questions on exams were replaced by four question quizzes associated with each lecture. These are contained in files quiz0aa.tex where aa is the lecture number (and quiz1aa.tex for 356). These were extracted from Canvas and translated into LaTeX by ChatGPT-4o. Chat can translate the LaTeX files back into whatever platform you are using if you wish to create such quizzes for your class (or at least it thinks it can: some editing will surely be required). The .xml zip file they are extracted from (which also contains the quiz answers) is available on request to faculty only. Finally, for CBE 30356 I instituted an in-class "Problem of the Day" which was solved in class. These problems are located in the files `pod1aa.tex` where aa is the lecture number. The solutions for these are found on the individual lecture pages at `https://diffusivepress.org/courseware/` in the course directory.

For exam and homework questions, each problem has both a problem page and an AI generated summary. The summary is given the same number as the problem, but with `s` appended (e.g., `problem0013.tex` and `problem0013s.tex`). These files are all in LaTeX, but they do not include the document header and package information. They are compiled by inserting the two file names into the file `aproblemtest.tex`, which can be run using TeXShop. In this case, LaTeX2e is used. TeXShop is freely available and works on MacOS, however other similar programs are available for other platforms. These files were prepared and tested using TeXShop on a M4 MacBook Pro running OS 15.5 Sequoia.

Most of the problems are extracted from my exams, and about half of these have solutions available on the web. Problems which are solved are so designated in the summary. To find the solution, you go to the `Directory` hyperlink at the bottom of the summary page and then, on that page, navigate to the past exam page to find the narrated video file for the particular exam in question. You may need to click on the exam pdf file itself to see which problem is which on a particular exam. Note that the problems may differ slightly from those on the old exams due to editing and conversion to LaTeX, and thus the solutions may

differ as well (but will match the problems on the exam pdfs).

The problems are loosely organized by subject material, and the summary has a rough estimate of time of completion and difficulty. I used Chat to generate these summaries, and in some cases went back and modified them to make it more consistent and correct - but time and difficulty estimates may be off. I have left Chat's estimate of these unchanged, as in general this depends on how closely aligned the exam question is to the class homework assignments. For example, a problem of recasting vector notation into index notation is simple - but only if index notation is covered in the class! Other problems such as using Archimedes Law to determine density and metal content is rated as "moderate" by Chat, but in my case it is nearly identical to previously assigned homework. I find that the Chat time estimates are (sometimes) inflated by a factor of two, although I'm afraid my students would disagree with this assessment. Thus, these are only rough guides in selecting questions for an exam. You may be able to get better estimates from Chat by pasting in the problem (or exam) and then asking it to solve it and give a time estimate - occasionally the solutions produced by Chat are even correct!

Finding a Problem

There are very many problems included in this compendium, so identifying a particular problem can at first appear intimidating. Each summary and problem file (e.g., `problem0013s.tex` and `problem0013.tex`) includes metadata at the top. These can be used in Python scripts to filter problems by topic or difficulty. However, flipping through the PDF of the Compendium is often more reliable for finding appropriate problems. The easiest method is simply opening this PDF in Preview or other PDF reader (not TeXShop as it has only a limited PDF search utility) which allows for searching for particular key words, highlighting problems where the words appear (e.g., "lubrication" appears on about 50 pages). Flipping through these pages allows for quick identification of problems of interest to be included in an exam or homework set.

Implementation

To use these problems without modifications:

1. Browse the compendium and note the numbers of any problems you wish to use.
2. Download and unzip the archive `transportproblems.zip`. This contains all of the files, including the LaTeX script `acompendium.tex` which produced this document.
3. In TeXShop, open the file `aexamtemplate.tex` (or duplicate it first). Replace the header with your exam name, date, and any exam conditions (e.g., "closed book" or provide the exam date).
4. Insert the selected problem filenames in order, assigning point values as desired.

5. Click `Typeset` to generate the PDF.

To generate a homework set, use the `ahomeworktemplate.tex` file similarly.

The default margin is 1 inch but can be adjusted in the `\geometry` command in the template file preamble (the material before the `\begin{document}` command). In general, LaTeX documents are less compact than word documents, and thus they take up more space - but are also more readable. Lines can be spaced closer together by removing or changing the `vspace` commands in the individual problem tex files.

Maximum compression can be achieved using Chat. To fully compress an exam, for example, open the individual problem scripts (e.g., `problem0013.tex`) and paste the LaTeX into your exam tex template, replacing the `input` commands so that the entire exam is in one file. Once this has been done, locate the Chat prompt file in the unzipped directory:

`ai_latex_compression_prompt.txt`

Then, simply paste it and your exam script in the appropriate spot. Chat will then produce a LaTeX script which will yield a much more compressed typeset output in TeXShop. Do check to make sure that Chat hasn't changed any parts of the problems (despite instructions), however.

Editing and Creating Problems

I have tried to make the directory structure of this Compendium as flexible and customizable as possible. Ultimately, I expect that it will be customized to your specific course needs. Because every class is different, you will certainly want to either modify the enclosed problems (e.g., you may -not- have demonstrated Hero's fountain in class, or previously assigned the sluice gate problem for homework) or add your own problems, and this is quite easy to do.

For minor edits to a problem:

- Open the `.tex` file in TeXShop or any text editor.
- Modify the content as needed. Vertical spacing can be adjusted using `\vspace{1em}` (positive or negative values allowed).
- To preview changes, use `aproblemtest.tex` or a custom exam file that includes the preamble information and the end of document command.

Do not try to compile individual problem files directly—they lack the necessary preamble. Note that if you kill a file by accident, you can always recover the original by unzipping the original zip file.

For major edits to a problem:

- Open the `.tex` file in TeXShop (you will want to use the LaTeX editor for this)
- Copy the contents into a file containing the necessary preamble (I use the appropriately named `junk.tex` file which is included in this directory).
- Modify the problem as desired, checking to see that you are getting what you expect.
- Create a new file and paste in the contents below the `\begin{document}` line and above the `\end{document}` line.
- Save the file under a name such as `myproblem001.tex`
- Add the appropriate `\input{myproblem001}` to the `myproblems.tex` file if you would like for it to be complied in the Compendium. A toc subsubsection line also needs to be added (or included in a separate `myproblem001s.tex` summary page) if you would like it to appear in the table of contents.
- Use it as you would any of the problems in this compendium.

Short Answer Questions

If you wish to include short answer questions on your exams, this is most easily done by copying the desired "item" from one of the short answer compilations files (e.g., `sageneral.tex`) and then inserting it into the file `ashortanswertemplate.tex`, editing as appropriate, and saving this under a new file name. You would then input this file into your `aexamtemplate.tex` file in the same manner as the other problems. Note that the short answer template does not have the preamble, and thus cannot be compiled directly. Thus, if you need to quickly see the version of the question (e.g., you are doing lots of edits or adding your own), you can also paste the script into `junk.tex` and compile. The multiple choice questions from the quiz files (e.g., `quiz001.tex`) can be used similarly.

New Problems From Old

You may wish to ask a slightly different problem from one that has been included here. Chat provides a simple way to do this. If you would like a variant on an existing problem, paste in a prompt such as:

```
I would like to generate a variant on an existing transport problem.
Please use the same formatting and packages, and write a latex script
which tests similar concepts to the following problem:
```

and then paste in the text from the LaTeX file and see what it gives you. The output would be copied and pasted into TeXShop for viewing and editing. In general, the figures (if present) will need a lot of work as the current versions of Chat are very bad at this. The problems suggested by Chat can be both interesting and (sometimes) rather strange, but are usually rather banal modifications. To incorporate your finished problem file into the exam template, remove the preamble (packages and begin document) and end of document commands and

save it under an appropriate file name. You may wish to generate a "summary page" for inclusion in your version of the Compendium as well (e.g., `myproblem001s.tex`).

Adding Your Own Problems

Odds are you will have many PDFs of old exams or problem sets that you would like to include in -your- version of this Compendium. You can generate new problems from your files using ChatGPT in the same way the files in this Compendium were generated. To ensure formatting consistency, use the prompt stored in the file:

`ai_latex_problem_prompt.txt`

Open the text file in a text editor and paste the text from the prompt into ChatGPT, followed by the problem statement (copied in "select text" mode from a PDF or other source). Chat will return a LaTeX script which can be copied and pasted into a file such as `junk.tex` in TeXShop. Note that you want Chat to generate a script rather than a file if you are using the "free" version of ChatGPT-4o as the free version severely limits file creation and file uploads, but is quite generous in absorbing material that is pasted in and returning scripts for copying. This entire Compendium was prepared in this manner with the free version of ChatGPT-4o (and "-mini" when -4o timed out). You then compile the .tex file using the Typeset command. I have also included the AI prompts I used for the homework problems, homework summary pages and quizzes, although these may or may not be appropriate for your specific needs.

Important: Carefully review the output. Chat sometimes modifies the problem or introduces errors in equations. Equations in particular are interesting: when copying an equation from a PDF you are essentially pasting in a string of symbols, and Chat then tries to figure out what the equation really was as a puzzle (and may draw the wrong equation off the web as a consequence of trying to find a match) and then gives you the LaTeX script for that equation. If you spot a mistake, re-pasting the equation into Chat often yields a corrected LaTeX script after one or two tries. You can also tell Chat how to correct its errors. Rather quickly, however, you will get the hang of editing the LaTeX of the equations yourself, particularly when you can paste your tries back into Chat and ask it to correct things or make suggestions.

Once satisfied:

- Adjust metadata fields and other problem information as desired.
- Copy the portion from the metadata up to (but not including) `\newpage` into a new file and save it as `myproblem001s.tex` or equivalent.
- Copy the portion from below `\newpage` up to (but not including) `\end{document}` into a new file and save it as `myproblem001.tex` or equivalent. Note that the `\end{document}` command MUST be removed, or the TeXShop compiler will stop compiling when it hits that command!

- Remove the document class, packages, and \begin{document} lines from both files if necessary.
- Insert these filenames into aproblemtest.tex and compile for a final check.
- Insert the problem file name myproblem001.tex into aexamtemplate.tex or your exam file and compile.
- If desired, your problem tex files can be added to this Compendium. Simply add "input" commands with these file names into the LaTeX script myproblems.tex (currently with just a placeholder) and recompile this book. It is necessary to recompile (Typeset) the book twice in TeXShop to update the table of contents and a third time to get the pagination correct, however Chat should have entered the correct toc command line into the summary file.

Figures

Figures are much more challenging than equations. Occasionally Chat will generate a decent figure right off, however if this fails (as it usually does) several alternative methods exist:

1. TikZ Prompt

Try the prompt:

```
please generate a latex script using tikz from the following figure:
```

Then paste the figure (in Preview: use rectangle select). Edit the TikZ output as needed in TeXShop (e.g., save as figuretest.tex). You copy working TikZ figures into your myproblem001.tex file (or junk.tex file) in the appropriate spot, but do not include the header or end of document commands. You generally only get a couple of these conversions if you are using the free version of Chat. This method occasionally works, and is convenient when it does.

2. Descriptive Prompt

Alternatively, describe the desired figure in words. For example:

```
create a latex figure using TikZ that has the following elements:  a
flat plate, a simple shear flow (with a bunch of arrows) in the x-direction
(linearly increasing in y) and a curved arc beginning close to the left
edge of the flat plate showing the boundary layer.
```

This often gets you much closer to the desired output, however it still usually requires LaTeX editing in TeXShop.

3. Further Descriptive Prompts

Sometimes Chat gets close, but needs a little help. You can sometimes get a better result (or one which is more easily editable manually) with commands such as:

```
please replace the rectangle representing the arbitrary control volume
with an angled ellipse.  Be sure that the labels do not overlap, and
have the velocity vector pass entirely through the ellipse.
```

Variants on this would be to ask Chat for TikZ script suggestions for a circle, arc , Bezier curve, etc. that you can include in your figure. There are an amazing number of possibilities, and Chat is pretty good at making useful suggestions and explaining how to modify your diagram. It is also invaluable for diagnosing LaTeX errors in figure drawings. Just paste in the offending line (identified in the TeXShop console) and it will give you the corrected version, or at least one which will compile.

4. Adapting Similar Figures

Often a diagram for your problem will be somewhat similar to a diagram in one of the included problems. While this inevitably requires editing, getting "new from old" is effective. Flip through the problems in this Compendium and identify one which has a somewhat similar diagram - or at least one with similar elements. Then open the relevant `problem0001.tex` file in TeXShop and copy the TikZ script. You then paste it into `junk.tex` or other file containing the necessary preamble, compile, and edit as appropriate. You can also paste the TikZ script into Chat and ask it for suggestions, or to move things around for you - a very interactive process. Ultimately, however, you are likely to need to adjust the coordinates of the various elements of the figure yourself: tedious but effective.

5. Embedded Graphics

If all else fails, take a screenshot of your diagram, save as PNG or JPG, and include the figure with:

```
\begin{figure}[h!]
    \centering
    \includegraphics[width=0.85\textwidth]{cp220i.jpg}
    \caption{Pump Curve for Sykes Pump CP-220i}
    \label{fig:cp220i pump curve}
\end{figure}
```

The size of the image can be changed using the "width" command as above. If you forget these prompts, or other LaTeX commands, having a browser open to Chat and asking: "latex figure insertion" or "tikz figure insertion" will generate the appropriate commands that you can copy and paste into your file.

Using TikZ is preferable for file size (and the figure is imbedded in the problem file), but direct image capture is often much easier and faster to implement. The picture file must be

in the same directory as your problem file so that TeXShop can find it when generating the exam PDF.

Final Notes on Editing

A common error is omitting required packages in the preamble. The template files (`junk.tex`, `aproblemtest.tex`, `aexamtemplate.tex`, `ahomeworktemplate.tex`) include all necessary packages for the problems given here. If Chat adds more in its script (and it may when generating figures in particular), ensure they are also present in the preambles of the template files or ask Chat not to use those packages.

It is also necessary to request Chat not to include unicode characters, as those cause errors in LaTeX compiling (and should be replaced by Chat with the appropriate LaTeX commands). Unicode characters are commonly produced in Word documents, and even with instructions to the contrary Chat will sometimes let them slip through in the conversion (as happened in the creation of this compendium on many occasions). This is particularly aggravating if it contaminates the table of contents file. That is because of the way TeXShop uses the toc file: you typeset a LaTeX script containing a toc at least twice, and preferably three times, the first time using a previous toc (or auto-generated one) and that modifies the toc file. The second time through everything in the new toc (should) appear. If the toc file is corrupted, however, the process no longer works, even if the error has been fixed in the script: very frustrating. The fix is easy, however, as after correcting the error in the script, you delete the auxiliary files TeXShop produces (everything but the .tex file) and typeset again (twice - and be sure to wait for it to finish before clicking Typeset the second time!). Note that this sort of compile error happens fairly often for other script errors as well, or even simply trying to compile the second time before the first one has quite finished. The TeXShop console has a "Trash Aux Files" button for that reason. Interestingly, for this document you actually need to typeset a third time as the third time adjusts the pagination to account for the length of the toc if that has changed with your edits.

Note that if you have any questions about what LaTeX commands do, or what sort of commands you would like to use to customize your document, Chat is great at giving you suggestions for testing things out. While LaTeX is quite intimidating at first (or at least I found it so), using Chat to both generate scripts and explain what is going on yields a very fast ascent of the learning curve. Recognize, however, that Chat may change your text in ways that you do not desire: check any output carefully!

Table of Problems

CBE 30355 Exam Questions **22**
Derivation of Transport Equations 23
Species Mass Conservation with Diffusion and Reaction 24
Derivation of the Continuity Equation 26
Derivation of Transport Equations 28
Cauchy Stress Equations from Momentum Conservation 30
Energy Conservation in a Fluid Element 32
Hydrostatics 34
Hydrostatics and Archimedes Law 35
Hydrostatics and Buoyancy Evaluation of a Gold Chain 37
Hydrostatics: Piston Height Difference in Communicating Tubes 39
Hydrometer Design 41
Archimedes and Immersed Cylinder 43
Hydrostatics and Balloon Ascent 45
Hydrostatics and Archimedes Law for a Straw-Based Hydrometer 47
Capillary Rise and Entry: Hydrostatics 49
Conservation of Momentum 51
Conservation of Momentum – Leaky Tank 52
Force on a Hose Nozzle 54
Flow in a Nozzle from a Pressurized Tank 56
Momentum Balance in Wake Traverse Experiment 58
Jet Deflection Supporting a Mass 60
Momentum Balances and Bernoulli's Equation 62
Torque on a Stationary Sprinkler 64
Jetlev Flyer Momentum Balance 66
Jetlev Flyer Nozzle Optimization 69
Integral Momentum Balance on a Fountain System 71
Integral Momentum Balances – Vertical Water Jet Supporting a Mass 73
Hydroelectric Power Generation on the St. Joseph River 75
Conservation of Momentum and Bernoulli Analysis for Firefighting Nozzle . 77
Conservation of Momentum in the DART Asteroid Test 79
Conservation of Momentum for Asteroid Deflection 81
Thinning Water Stream from Faucet 83
Persian Water Clock and Sinking Time Analysis 85
Pelton Wheel Energy Extraction 87
Vacuum Cannon Projectile Velocity 89
Undershot Waterwheel Pumping from the Edisto River 91
Hydroplaning Threshold Velocity via Dimensional Analysis 93
Unidirectional Flows 95
Unidirectional Flows – Tube Filling Under Viscous Flow 96
Poiseuille Flow in a Syringe and Needle System 98
Flow Resistance in Membranes: Unidirectional Flows 100
Unidirectional Flows in an Inclined Semi-Circular Trough 102

Coil Pump Performance and Flow Analysis . . . 104
Unidirectional Flows . . . 107
Flow Rate in a Channel from Pressure Gradient . . . 109
Capillary Rise in a Narrow Channel . . . 111
Drainage of a Viscous Fluid Through a Narrow Slit . . . 113
Capillary Filling of a Microchannel . . . 115
Lid-Driven Cavity Flow and Pressure Gradient . . . 117
Lubricated Thrust Bearing Analysis . . . 119
Unidirectional Flow in a Viscous Damper . . . 121
Unidirectional Flows . . . 123
Creeping Flow in a Lid-Driven Cavity . . . 125
Viscous Dampers and Cavitation . . . 127
Plane Poiseuille Flow in a Microfluidic Channel . . . 129
Plane-Poiseuille Flow in a Microchannel . . . 131
Pressure-Driven Flow in a Microchannel . . . 133
Plane Poiseuille Flow: Cell Adhesion and Wall Shear Stress . . . 135
Required Flow Rate in a Microchannel to Generate Shear Stress . . . 137
Oscillatory Unidirectional Flow in a Channel . . . 139
Unidirectional Flow Between Cylinder and Rod . . . 141
Flow Down an Incline . . . 143
Viscosity Measurement by Flow Down an Inclined Tray . . . 145
Inclined Film with Counteracting Shear . . . 147
Breeze Opposes Gravity on Inclined Film . . . 149
Continuous Coating of a Sheet with a Viscous Film . . . 151
Unidirectional Flows (or the heat transfer analog!) . . . 153
Electroosmotic Mobility in Microchannels . . . 155
Parallel-Plate Flow . . . 157
Lubrication Flow Under a Plate . . . 159
Falling Slug Rheometry in Cylindrical Geometries . . . 161
Stratified Shear Flow Between Plates . . . 163
Yield Stress Measurement in Ranch Dressing . . . 165
Quasi-Parallel Flows and Lubrication . . . 167
Lubrication Flow in a Collapsing Wedge . . . 168
Lubrication and Cavitation in Separating Plates . . . 170
Lubrication: Closure of a Hinged Plate . . . 172
Lubrication Theory: Detachment of a Strip from a Plane . . . 174
Lubrication Force on Cone in Contact with Plane . . . 176
Lubrication: Detachment of a Sphere from a Plane . . . 178
Spin Coating and Thin Film Dynamics . . . 180
Scaling and Quasi-Parallel Flow in a Porous Fiber . . . 182
Quasi-parallel flows . . . 184
Drainage Flows . . . 186
Two-Dimensional Flows . . . 188
Streamfunction in a Triangular Trough . . . 189
Two Dimensional Drainage from a Slit . . . 191

Boundary Layer Flows . . . 193
Thermal Boundary Layers in Shear Flow . . . 194
Uni-directional Startup Flows . . . 196
Unsteady Unidirectional Flow . . . 198
Scaling analysis of boundary layer flows . . . 201
Scaling Analysis of Stagnation Flow . . . 203
Thermal Boundary Layers with Linearly Increasing Wall Flux . . . 205
Heat Transfer from a Rotating Disk with Boundary Layer Flow . . . 207
Heat Transfer from Plate in Impinging Jet Flow . . . 209
Unsteady Heat Transfer in Solids . . . 211
Scaling of the Boundary Layer Equations for Wedge Flow . . . 213
Thermal and Momentum Boundary Layers . . . 215
Dimensional Analysis and Scaling . . . 218
Dimensional Analysis of Drop Breakup . . . 219
Inspectional Analysis of Energy Transport in Flow Past a Heated Spher . . . 221
Inspectional Analysis of Oscillatory Channel Flow . . . 223
Inspectional Analysis of Ship Drag Using a Scale Model . . . 225
Dimensional Analysis in Ship Design . . . 227
Dimensional Analysis in Ship Design . . . 229
Dimensional Analysis of Drag Scaling for a Ship Model . . . 231
Scaling of Freighter Design via Dimensional Analysis . . . 233
Scaling and Blowout Time for a Draining Straw . . . 235
Dimensional Analysis of Dripping Drop Radius . . . 237
Scaling/Unidirectional Flows . . . 239
Dimensional Analysis . . . 241
Dimensional Analysis/Inertial Flows: Can Crushing by Vapor Condensation 243
Dimensional Analysis: Dip Coating with a Viscous Fluid . . . 245
Scaling Analysis of Radial Flow Between Disk and Plane . . . 247
Scaling / Unidirectional Flows . . . 249
Dimensional Analysis: Emulsion Drop Size . . . 251
Dimensional Analysis and Scaling in Mixing Tanks . . . 253
Scaling Analysis for a Viscous Mixing Tank . . . 255
Hydrostatics and Submarine Implosion Scaling . . . 257
Kolmogorov Scale and Mixing Simulation Requirements . . . 259
High Reynolds Number Flow Down an Incline . . . 261
Axial Force from Rotating Disk in Lubrication Limit . . . 263
Dimensional Analysis of Shockwave Expansion . . . 266
Surface Tension and Drainage of a Sealed Can . . . 268
Dimensional Analysis: Hanford Pulse Jet Mixer Modeling . . . 270
Inertial Impaction and Face Shields . . . 272
Pipe Flows . . . 274
Hero's Fountain . . . 275
A Variant on Hero's Fountain . . . 278
A Simple Fountain . . . 281
Siphon Flow and Spray Height . . . 283

Siphon Drainage Design and Time Estimation . . . 285
Siphon Valve Design for Prescribed Flow . . . 288
Flow Measurement Using Orifice Meters . . . 290
Coil Pump Design for Irrigation . . . 293
Overflow Drain Design for Tank . . . 296
High Re Flow Through Syringe . . . 298
Pumps and Pipes . . . 300
Pumping System and Head Loss . . . 301
Pumps & Pipes! . . . 304
High Re Flow/Pump Curves . . . 307
Pumped Hydroelectric Energy Storage in a Residential Setting . . . 310
Irrigation Pumping with CP-80i: Flow Rate and Efficiency Estimation . . . 314
Pump Curve Analysis and Application . . . 317
Pump Curves . . . 319
Pump Curve Performance and Sizing . . . 321
Collated Short Answer Questions . . . 323
High Reynolds Number Short Answer Questions . . . 324
Turbulence Short Answer Questions . . . 326
Low Reynolds Number Short Answer Questions . . . 327
Term Identification Short Answer Questions . . . 329
Equation Identification Short Answer Questions . . . 331
Short Answer Calculations . . . 333
General Short Answer Questions . . . 336
Index Notation Short Answer Questions . . . 340

CBE 30355 Quizzes . . . 342
Lecture 1: What are Fluid Mechanics? . . . 343
Lecture 2: The Continuum Hypothesis . . . 344
Lecture 3: Hydrostatics . . . 345
Lecture 4: Lagrangian and Eulerian Descriptions . . . 346
Lecture 5: Fluids in Motion: The Continuity Equation . . . 348
Lecture 6: Conservation of Mass in Multicomponent Systems . . . 349
Lecture 7: Conservation of Momentum . . . 350
Lecture 8: Cauchy Stress Equation . . . 351
Lecture 9: The Navier-Stokes Equations . . . 352
Lecture 10: Plane Couette Flow, Flow Down an Inclined Plane . . . 354
Lecture 11: Poiseuille Flow, Couette Flow . . . 355
Lecture 12: Couette Flow, Dynamic Similarity . . . 356
Lecture 13: Dynamic Similarity . . . 357
Lecture 14: Dimensional Analysis . . . 358
Lecture 15: Lubrication Flows . . . 359
Lecture 16: Reynolds Lubrication Equation, Streamfunction . . . 360
Lecture 17: Flow Near a Wiper . . . 361
Lecture 18: Flow Past a Sphere . . . 362
Lecture 20: Bernoulli's Equation . . . 363

Lecture 21: Potential Flow Past a Cylinder . . . 364
Lecture 21: Prandtl Boundary Layer Equations . . . 365
Lecture 22: Flow Past a Flat Plate: The Blasius Equation . . . 367
Lecture 23: Complex Boundary Layer Flows . . . 369
Lecture 24: Integral Boundary Layer Equations . . . 371
Lecture 25: Friction Factors . . . 372
Lecture 26: Piping Networks & Review . . . 373
Lecture 27: Turbulence . . . 374

CBE 30355 Problem Sets . . . 375
CBE 30355 Problem Set 1 . . . 376
Velocity Distribution in a Tube under Pressure Gradient . . . 377
Vector Dot and Cross Products . . . 379
Vector Operators on Scalar and Vector Fields . . . 381
Divergence of the Curl of a Vector Field . . . 383
Transient Simple Shear Flow . . . 385
Vector Identity Involving Double Curl . . . 387
CBE 30355 Problem Set 2 . . . 389
Estimating Viscosity of Air Using Chapman-Enskog Theory . . . 390
Modeling Liquid Viscosity Using Inverse Temperature Fits . . . 392
Hydrostatics of a Self-Gravitating Gas Sphere . . . 394
Hydrostatic Force on a Ball Plugging a Pool Drain . . . 396
CBE 30355 Problem Set 3 . . . 398
Lawn Chair Larry and Hydrostatic Lift . . . 399
Salt Accumulation in an Evaporating Fish Tank . . . 401
Lubrication Flow Between Squeezing Plates . . . 403
Surface Integrals Over Spheres in Index Notation . . . 405
CBE 30355 Problem Set 4 . . . 407
Force on a Firehose Nozzle . . . 408
Force Exerted by a Jet on a Plate . . . 410
Force on a Sluice Gate . . . 412
Determining λ_1 and λ_2 from Settling Cylinder Mobilities . . . 414
CBE 30355 Problem Set 5 . . . 416
Flow Between Parallel Plates (Plane-Poiseuille Flow) . . . 417
Shear Stress Needed to Arrest Flow Down an Incline . . . 419
Curvature Effects in a Couette Viscometer . . . 421
Mobility Tensor and Trajectory of a Falling Washer . . . 423
CBE 30355 Problem Set 6 . . . 425
Parallel Plate Viscometer Analysis . . . 426
Flow in a Tube with an Axial Wire . . . 428
Core-Annular Flow of a Viscous Fluid . . . 430
Viscosity Pump Performance and Efficiency . . . 432
CBE 30355 Problem Set 7 . . . 434
Dimensional Analysis in Cooking a Turkey . . . 435
Damped Pendulum in a Viscous Fluid . . . 437

Scaling of Mixer for Density-Stratified Fluids . . . 439
Stokes Flow and Index Notation . . . 441
CBE 30355 Problem Set 8 . . . 443
Lubrication Layer Between Rollers . . . 444
Zero Error in Parallel-Plate Viscometers . . . 446
Thermodynamics and Scaling Analysis . . . 448
Dimensional Analysis: Modeling Air Entrainment in a Draining Tank . . . 450
CBE 30355 Problem Set 9 . . . 452
Flow inside a 90º trough . . . 453
Film Drainage Flows . . . 455
Dimensional Analysis of Flow Down an Inclined Plane . . . 457
The Sliding Block . . . 459
CBE 30355 Problem Set 10 . . . 461
Scaling Of The Transport Equations . . . 462
Scaling Of The Transport Equations . . . 464
Potential Flow Past a Cylinder . . . 466
Conical Cork in Flow . . . 468
CBE 30355 Problem Set 11 . . . 470
Heated Plate in Shear Flow: Similarity Solution and Heat Loss . . . 471
Boundary Layer Growth with Suction . . . 474
Spinning Disk Electrode and von Kármán Flow . . . 476
CBE 30355 Problem Set 12 . . . 479
Jetlev Flyer Pump Requirements . . . 480
Overflow Drain Design . . . 483
Pump Curve Analysis . . . 485

CBE 30356 Exam Questions . . . 487
CBE 30356 Mid-Term Examinations . . . 488
Dimensionless Groups . . . 489
Heat Conduction in Solids . . . 492
Convective Heat Transfer in Couette Flow . . . 494
Boundary Layer Solution . . . 496
Dimensionless Groups . . . 498
Heat Conduction in Solids . . . 500
Convective Heat Transfer/Scaling . . . 502
Transient Boundary Layers . . . 504
CBE 30356Final Examinations . . . 506
CBE 30356 Final Exam Equation Sheet . . . 507
Heat and Mass Transfer in a Stefan Tube . . . 511
Local Nu in Shear-Heated Boundary Layer . . . 514
Radiative Cooling of a Fin in Space . . . 516
Mass Transfer From a Falling Drop . . . 518
Convective Heat & Mass Transfer . . . 520
The Thiele Problem . . . 522
A Cylindrical Cooling Fin . . . 524

Dimensionless Groups 526

CBE 30356 Problem Sets 528
CBE 30356 Problem Set 1 529
Transport Properties of Methane from 200K to 1000K 530
Heat Loss Through Attic Ceiling and Effect of Insulation 532
Testing the Wiedemann-Franz Law for Metals 534
Anisotropic Thermal Conductivity in Laminated Materials 536
Tensorial Conductivity in Anisotropic Laminates 538
CBE 30356 Problem Set 2 540
Hot Wire Anemometry and Convective Cooling 541
Heat Generation in an Annular Source 543
Long Cooling Fin with Symmetric Convection 545
Thermal Runaway in a Reactive Slab 547
CBE 30356 Problem Set 3 549
Startup of a Heated Slab 550
Transient Solution for Heated Slab with Time-Dependent Boundary 552
Heating of a Sphere with Constant Surface Flux 554
Steady 2D Heat Transfer Using Sturm-Liouville 556
CBE 30356 Problem Set 4 558
Heated Rod with Surface Heat Loss 559
Transient Heating of a Cylindrical Rod 561
Finite Difference Solution to Transient Heating 563
Rectangular Slab with Volumetric Heating 565
CBE 30356 Problem Set 5 567
Transient Start-Up of Flow in a Tube 568
Early-Time Boundary Layer with Internal Heating 570
Numerical Solution of Self-Similar Heating Problem 572
Periodic Heat Flux at the Wall of a Semi-Infinite Medium 574
CBE 30356 Problem Set 6 576
Thermal Entrance Region with Constant Wall Temperature 577
Asymptotic Nusselt Number in a Channel with Constant Heat Flux 579
Asymptotic Nusselt Number for Uniform Flow with Constant Heat Flux . . 581
Asymptotic Nu for Uniform Flow with Constant Wall Temperature 583
CBE 30356 Problem Set 7 585
Transpirational Cooling in a Spherical Tank 586
Heat Exchanger Design and Sizing 588
Natural Convection Scaling from a Heated Plane 590
Heat Loss through Residential Window Glazing 592
CBE 30356 Problem Set 8 594
HB Fuller Heat Transfer Analysis 595
Evaporation of a Floating Drop in Microgravity 597
Evaporation with Thermal Feedback in Microgravity 599
Evaporation Time of a Microgravity Water Drop 601
CBE 30356 Problem Set 9 603

CBE 30356 Problem Set 10 . . . 604
Evaporation and Sedimentation of Water Drops . . . 605
CBE 30356 Problem Set 11 . . . 607
Air-Water Stripping of Trichloroethylene from Groundwater . . . 608
Equimolar Counter-Diffusion with First Order Reaction . . . 610
Taylor Dispersivity Transition via Monte Carlo Simulation . . . 612
CBE 30356 Problem Set 12 . . . 614
Concentration Polarization in Simple Shear Flow . . . 615
Electrostatics and Electroosmosis at a Charged Glass Surface . . . 617

CBE 30356 Problems of the Day . . . 619
Lecture 1 POD: Calculation of Thermal Conductivity of Gases . . . 620
Lecture 2 POD: Design of a "Heat Transfer Stethoscope" . . . 621
Lecture 3 POD: Analysis of the SNAP-27 RTG . . . 622
Lecture 4 POD: Radiation Fins in Space . . . 623
Lecture 5 POD: Large Biot Heat Transfer from a Sphere . . . 624
Lecture 6 POD: Transient Heating of a Slab . . . 625
Lecture 7 POD: Quenching of a Sphere at Finite Bi . . . 626
Lecture 8 POD: Neumann Instability in Finite Difference Solutions . . . 627
Lecture 9 POD: Transient Heating of a Semi-Infinite Slab . . . 628
Lecture 11 POD: Scaling: Squishing a Fluid Between Two Plates . . . 629
Lecture 12 POD: Step Change in Wall Temperature for Tube Flow . . . 630
Lecture 14 POD: Buoyancy Driven Flow for a Vertical Wall . . . 631
Lecture 15 POD: Analysis of the Webb Telescope Sun Shield . . . 632
Lecture 17 POD: Diffusion through a Stagnant Spherical Layer . . . 633
Lecture 18 POD: Thiele Effectiveness Factor for a Flat Particle . . . 634
Lecture 19 POD: Dissolution with First Order Reaction . . . 635
Lecture 20 POD: Mass Transfer Limitations in a Biodetector . . . 636
Lecture 21 POD: Calculation of H_{OX} for a Stripping Tower . . . 637
Lecture 22 POD: Taylor Dispersion in Rectangular Channels . . . 638
Lecture 23 POD: MC Simulation of AF4 Separations . . . 639

CBE 30356 Quizzes . . . 641
Lecture 1: Fourier's Law & the Origin of Thermal Conductivity . . . 642
Lecture 2: One Dimensional Heat Conduction in Solids . . . 643
Lecture 3: Heat Transfer with Generation . . . 644
Lecture 4: Cooling Fins and Heat Transfer Coefficients . . . 645
Lecture 5: The Nusselt & Biot Numbers and the Quenching of a Sphere . . . 646
Lecture 6: The Sturm-Liouville Theorem and the Startup of a Heated Slab . . . 647
Lecture 7: Matrix Methods for Numerical Solutions to SL Problems . . . 649
Lecture 8: Numerical Solutions to Non-Linear Problems . . . 650
Lecture 9: Boundary Layer Solutions to Transient Problems . . . 652
Lecture 10: Periodic Heating / Penetration Depth . . . 653
Lecture 11: Convective Heat Transfer: Derivation and Scaling of Equations . . . 654
Lecture 12: The Graetz-Nusselt Problem . . . 655

Lecture 13: Nusselt Number Correlations and Heat Exchangers 656
Lecture 14: Nusselt Number Correlations for External Flows 657
Lecture 15: Spectral Effects in Radiative Heat Transport 658
Lecture 16: Fick's Law and the Conservation Equations 659
Lecture 17: The Stefan Tube 660
Lecture 18: Gas Phase Mass Transfer with Reaction: The Thiele Modulus 661
Lecture 19: Diffusion with Homogeneous Reaction in Liquids 662
Lecture 20: Convective Diffusion into a Falling Film 663
Lecture 21: Gas-Liquid Stripping: Nox, Hox, and Overall Mass Transfer Coefficients664
Lecture 22: Taylor-Aris Dispersion 665
Lecture 23: Concentration Polarization and Field-Flow Fractionation 666
Lecture 24: Concentration Polarization in Rotating Membrane Systems 667
Lecture 25: Cross-Flow Filtration: From Desalination to Hemodialysis 668
Lecture 26: Elementary Electrostatics in Aqueous Media 669
Lecture 27: Electrokinetics: Electroosmosis and Electrophoresis 670

My Problems **671**
My Problem Placeholder 672

CBE 30355 Exam Questions

This section presents the problems from the CBE30355 Transport I examinations. The problems are broken into rough categories, although many of the problems draw from more than one area: for example, unidirectional startup flow in a channel requires solving the steady-state Poiseuille flow profile (simple), scaling the equations for the transient case to determine characteristic times and stresses, and then solving the boundary layer flow associated with the initial transient. Thus, the division is somewhat arbitrary. The problems are also somewhat repetitive: flow down an incline is looked at for several different possible boundary conditions, for example.

Derivation of the Transport Equations

In this section are gathered problems which use integral conservation laws and apply the divergence theorem to yield the differential form of transport equations. As the technique is essentially the same in all cases, the differences are really about the different applications: the continuity equation, conservation of mass in a multicomponent system, the energy equation, and even a relationship for mass conservation in suspensions.

Species Mass Conservation with Diffusion and Reaction

Subject Tags: species conservation, diffusion, reaction, control volume, divergence theorem

Difficulty Level: Moderate

Estimated Time: 30 minutes

This problem concerns the derivation of the species conservation equation in a reacting, multicomponent fluid. The system involves convective transport, diffusive fluxes, and source terms due to chemical reaction. Starting from a verbal control volume description, you will formulate an integral expression and apply the divergence theorem to obtain the microscopic differential form. No constitutive law is assumed for diffusion.

Skills Tested:

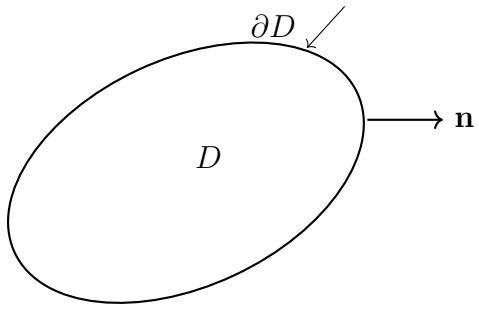
- Conservation of species in control volume form
- Conversion from integral to differential forms using the divergence theorem
- Interpretation and handling of vector fluxes and reaction sources

Source: CBE 30355 1st Hour Exam, 2001 **File:** problem0006.tex

Species Conservation in a Reacting Multicomponent Flow

Consider the arbitrary fluid element depicted below. If we have a flow containing several species which are undergoing reaction (a source/sink per unit volume) and diffusion (a flux of each species in addition to convection), derive the equation which governs the conservation of each species.

The source of species i due to reaction will be denoted R_i (units of mass of i per unit time per unit volume), and the total mass flux of species i (diffusion and convection) is given by $\rho_i \vec{u} + \vec{j}_i$, in which ρ_i is the mass of species i per unit volume, $\vec{u}$ is the total mass-average velocity of the fluid, and $\vec{j}_i$ is the diffusive flux of species i. Note that both $\vec{u}$ and $\vec{j}_i$ are vectors. We are not using index notation in this problem.



a. Write down in words the law governing the conservation of the mass of species i for the control volume D. Don't forget diffusion and reaction.

b. Write down the integral relationship equivalent to that in part a.

c. Through application of the divergence theorem, obtain a microscopic equation valid at every point in the flow. This will still be in terms of $\vec{j}_i$ (we have not specified a constitutive relation here for diffusion, analogous to Newton's law of viscosity or Fourier's law of heat conduction — don't worry about it).

Derivation of the Continuity Equation

Subject Tags: Continuity Equation; Compressible Flow; Mass Conservation; Derivation

Difficulty: Moderate

Estimated Time: 20 minutes

Summary: This problem guides the student through the derivation of the continuity equation for time-dependent, compressible flows by starting from the fundamental principle of mass conservation. The resulting equation is a foundational expression in fluid dynamics and serves as a basis for more advanced flow analysis.

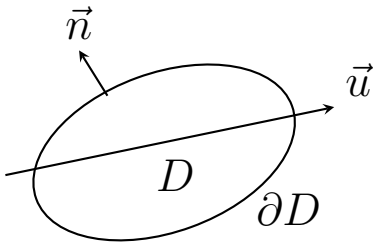
Skills Tested:

- Understanding of mass conservation
- Derivation of partial differential equations
- Application to compressible, time-dependent flows
- Interpretation of divergence and flow field behavior

Source: CBE 30355 Final Exam, 2001 **File:** problem0012.tex

Derivation of the Continuity Equation

Beginning with the statement that mass is a conserved quantity, derive the continuity equation for a time-dependent, compressible flow.



Derivation of Transport Equations

Subject Tags: Transport Phenomena; Particle Suspensions; Conservation Laws; Diffusion; Convection

Difficulty Level: Moderate

Estimated Time: 40 minutes (10 min derivation, 10 min substitution, 20 min nondimensionalization)

Summary: This problem leads students through the derivation of a microscopic transport equation for a dilute particle suspension in a fluid, emphasizing the analogies between momentum, mass, and energy transport. It incorporates convective, settling, and diffusive fluxes and culminates in a nondimensional form revealing important physical parameters.

Skills Tested:

- Applying the integral form of conservation laws
- Using the divergence theorem
- Expressing physical processes through flux terms
- Nondimensionalizing transport equations
- Identifying and interpreting dimensionless parameters

Source: CBE 30355 Second Hour Exam, 2002 **File:** problem0022.tex

Dilute Suspension Transport Equation

In this problem we derive a microscopic balance describing the concentration distribution $c(x,t)$ of a very dilute suspension of small particles in an incompressible, unsteady flow. Note: $c(x,t)$ is the local volume fraction of particles in the fluid (volume of particles per volume of fluid), and is therefore dimensionless.

a. The flux of particles is the sum of contributions from convection, settling $\vec{q}_s$, and diffusion $\vec{q}_D$, all of which have units of volume per area per time. Write an integral balance for the conservation of particles over an arbitrary control volume D. Then, using the divergence theorem, convert all surface integrals into volume integrals to obtain a differential equation valid at any point in the fluid.

b. For small particles of radius a, the settling and diffusive fluxes are given by:

$$\vec{q}_s = \frac{2}{9}\frac{\Delta\rho\, a^2}{\mu}\vec{g}\, c, \qquad \vec{q}_D = -\frac{kT}{6\pi\mu a}\nabla c$$

Using these expressions, substitute into your equation from part (a) to derive an explicit partial differential equation for $c(x,t)$.

c. Using reference scales for velocity U, length L, and time L/U, render the equation from part (b) dimensionless. Identify any resulting dimensionless groups and describe their physical significance. This equation provides a foundation for modeling suspensions where Brownian motion significantly affects the particle distribution.

Cauchy Stress Equations from Momentum Conservation

Subject Tags: Stress Tensor; Momentum Conservation; Cauchy Equations; Continuum Mechanics

Difficulty: Moderate

Estimated Time: 20 minutes

Summary: This problem asks the student to derive the local form of the momentum balance using the stress tensor and external body forces. The goal is to arrive at the Cauchy momentum equation, a foundational result in continuum mechanics.

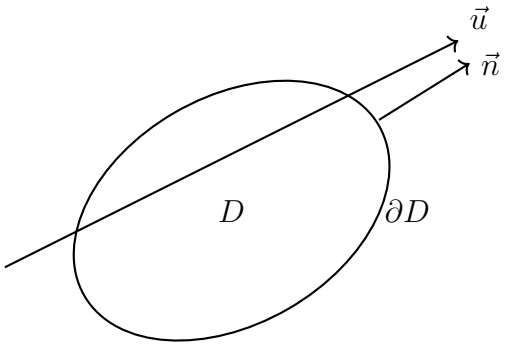
Skills Tested:

- Derivation from first principles
- Tensor notation and interpretation
- Application of conservation laws to control volumes

Source: CBE 30355 Final Exam, 2002 **File:** problem0025.tex

Cauchy Stress Equations from Momentum Conservation

As you know from Newton, momentum is conserved! Given that the momentum flux is described by the stress tensor $\boldsymbol{\sigma}$, and that the body force per unit volume due to gravity is $\rho\vec{g}$, derive the local form of the momentum balance equations, known as the Cauchy Stress Equations. Begin with a statement of conservation of momentum in words for an arbitrary control volume, write down the corresponding integral relationship, and then apply the divergence theorem. Leave your final equation in terms of the stress tensor $\boldsymbol{\sigma}$ - you would need to use Newton's law of viscosity to obtain the Navier-Stokes equations, which limits the result to Newtonian fluids.



Energy Conservation in a Fluid Element

Subject Tags: Energy transport; Integral and differential analysis; Fourier's law; Heat conduction; Source terms

Difficulty Level: Moderate

Estimated Time: 25 minutes (10 minutes conceptual setup, 15 minutes derivation)

Summary: This problem involves writing both the integral and microscopic energy balances for a flowing liquid where the internal energy per unit volume is given by $\rho C_p T$. The student must combine convection, conduction (using Fourier's Law), and a source term to formulate both a general integral form and then the corresponding differential form of the energy conservation equation, analogous to the derivation of the Navier-Stokes equations for momentum.

Skills Tested:

- Integral balance formulation
- Microscopic transport derivation
- Use of Fourier's law
- Conceptual understanding of source terms

Source: CBE 30355 First Hour Exam, 2016, solved **File:** problem0055.tex

Energy Conservation in a Fluid Element

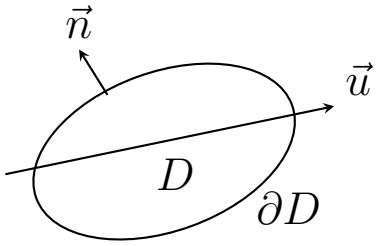
Consider the arbitrary fluid element depicted below. A fluid with thermal energy per unit volume $\rho C_p T$ (where T is the temperature and C_p is the heat capacity per unit mass) is flowing through the element with velocity $\vec{u}$. In addition to convective heat transfer, we also have the conductive (diffusive) heat transfer flux, given by Fourier's Law of heat conduction:

$$\vec{q} = -k\nabla T$$

which is analogous to the shear stress from Newton's law of viscosity. Here k is the thermal conductivity. There is also a source of thermal energy (heat generation) per unit volume S due to chemical reactions or other sources. Both the velocity $\vec{u}$ and the temperature T may be functions of position and time, but physical properties such as k, ρ, and C_p are taken to be constant.

a. Given the statement that thermal energy is conserved, derive an integral equation governing the thermal energy in the arbitrary control volume depicted below.

b. Using the results of part a, derive the microscopic equation governing energy transport in a flowing liquid analogous to the Navier-Stokes equations. You will use this equation a lot next term!



Hydrostatics

In this section are gathered problems that relate to hydrostatics: everything from high altitude balloons to the design of hydrometers and the application of Archimedes law to determining gold content.

Hydrostatics and Archimedes Law

Subject Tags: hydrostatics, buoyancy, density, Archimedes principle, material valuation

Difficulty level: Moderate

Estimated time: 20–30 minutes

Summary: This problem uses Archimedes' principle to determine the gold content of a submerged alloy chain based on measured weight differences in water and air. The solution involves calculating the alloy density, inferring the mass fractions of gold and copper, and then determining the karat value and monetary worth of the chain.

Skills Tested:

- Hydrostatics and buoyancy principles
- Use of Archimedes' law to compute volume and density
- Weighted average calculations of density and composition
- Material valuation based on composition and market value

Hydrostatics / Archimedes Law

You are given the task of determining the gold content of a "red gold" chain (actually a gold/copper alloy) that a friend has brought back from a trip to the Middle East, and thus how much it should be insured for.

Using a scale, you tare out a beaker of water (e.g., get it so that it reads zero). You dangle the chain via a fine thread in the water so that it is completely immersed (but suspended above the bottom), and find that the scale reads 13.5 g. You then let it settle to the bottom and find that the scale reads 171.4 g.

The density of gold is 19.3 g/cm^3 and costs \$57.90/gram. The density of copper is 8.6 g/cm^3 and costs \$0.0094/gram.

Given this:

a. What karat is the chain? (Recall that 24 karat = pure gold.)

b. How much is the metal in the chain worth?

Hydrostatics and Buoyancy Evaluation of a Gold Chain

Subject Tags: hydrostatics; buoyancy; Archimedes principle; divergence theorem; density calculation; applied problem

Difficulty Level: Moderate

Estimated Time: 35 minutes

Summary: This multipart problem builds from a derivation of Archimedes Law using integral force balance and the divergence theorem, and concludes with an applied question involving a submerged gold chain. The student must apply hydrostatic concepts and density data to estimate the true value of a jewelry item.

Skills Tested:

- Applying integral and differential force balances
- Using divergence theorem in fluid statics
- Relating weight change in submerged measurements to density
- Performing applied calculations with physical properties

Source: CBE 30355 First Hour Exam, 2019, solved **File:** problem0083.tex

Hydrostatics and Buoyancy Evaluation of a Gold Chain

Assume that the fluid is at rest and gravity is constant.

a. Using an integral force balance (such as was done in class!) derive Archimedes Law.

b. Using the divergence theorem, use this to obtain the equation governing the pressure distribution in a fluid at rest.

c. A friend picked up a gold chain at an estate sale "on spec," which the vendor said was "pure gold." Now jewelry is almost never pure gold (e.g., 24 karat = 24/24 parts by weight gold) as it is really too soft to use. The value of the chain depends on just how much gold is in it, and your friend turned to you as an expert in hydrostatics to see how much it is actually worth.

You fill a beaker with water and measure a mass of 350.0 g. You ball the chain up, tie a thread around it, and then suspend it in the water just above the bottom: the scale now reads 360.0 g. You then let the thread go and allow the chain to rest on the bottom and get a third reading of 470.0 g.

If the density of gold is 19.3 g/cm^3, the density of the metal alloy is 9.6 g/cm^3 (this depends on the composition, but is a typical value for the mixture of silver, copper, and zinc used in yellow gold), and the cost of gold is \$48 per gram, how much is the chain worth?

Hydrostatics: Piston Height Difference in Communicating Tubes

Subject Tags: hydrostatics; pressure balance; fluid statics; piston dynamics

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: Two vertical tubes connected at the bottom are filled with fluid and sealed with frictionless pistons of differing masses. The problem is to determine the new equilibrium heights of the pistons and the height difference between them, using hydrostatic balance and known initial conditions.

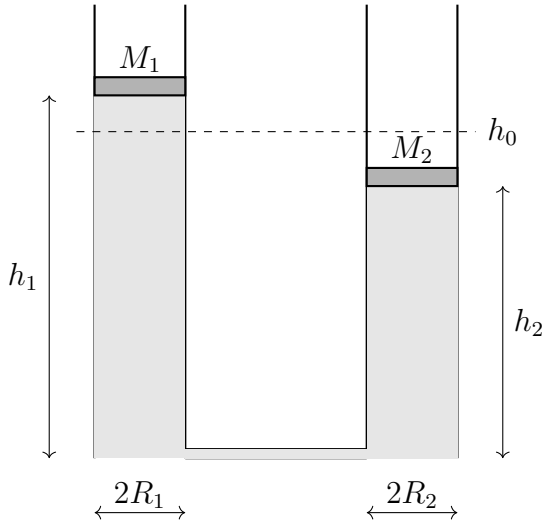
Skills Tested:

- Application of hydrostatic pressure equations
- Use of boundary conditions in fluid systems
- Force balance involving pistons and pressure
- Algebraic rearrangement and interpretation

Source: CBE 30355 First Hour Exam, 2011, solved **File:** problem0028.tex

Hydrostatics: Piston Height Difference in Communicating Tubes

Consider the system depicted below. Two tubes, filled with a fluid of density ρ, are connected at the bottom. The tube radii are R_1 and R_2, respectively. A piston (ignore friction!) sits on the top of the fluid in each tube. The mass of the piston in tube 1 is M_1, and the mass of the piston in tube 2 is M_2. Initially both pistons are at a height h_0, and then they equilibrate to different heights h_1 and h_2.



a. Calculate the equilibrium height difference $h_2 - h_1$.

b. Determine both h_2 and h_1 in terms of the initial height h_0.

Hydrometer Design

Subject Tags: buoyancy, fluid statics, density measurement, hydrostatics

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: This problem requires deriving a relation for the height a hydrometer column protrudes above a salt water solution. Using basic force balance and density relationships, the height is related to known material and geometric parameters.

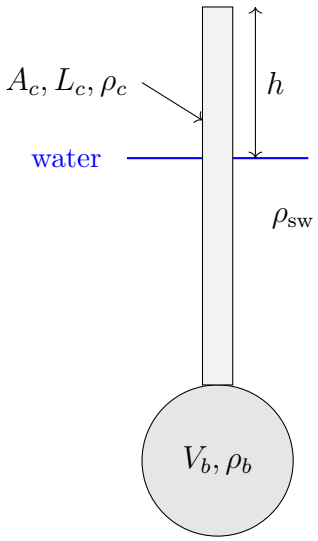
Skills Tested:

- Setting up and applying force balances
- Understanding buoyancy in static fluids
- Applying Archimedes' principle and density relations

Source: CBE 30355 First Hour Exam, 2012, solved **File:** problem0040.tex

Hydrometer Design

In this problem we design a hydrometer, a simple device for measuring the density of a liquid, usually used in determining the salt concentration. A simplified version is depicted below, and consists of a bulb of volume V_b and density ρ_b, and a column of length L_c, cross-sectional area A_c, and density ρ_c. Given this, determine the relation between the height h the column sticks up out of the water as a function of ρ_{sw} (the density of the salt water solution we are interested in) and the other parameters in the problem.



Archimedes and Immersed Cylinder

Subject Tags: Hydrostatics; Buoyancy; Archimedes Law; Fluid Statics

Difficulty Level: Moderate

Estimated Time: 20 minutes (10 min setup, 10 min derivation/application)

Summary: This problem explores the origin and application of Archimedes' Law. Part (a) requires deriving the buoyant force by applying an integral force balance over a submerged body. Part (b) involves applying Archimedes' Law to a cylinder floating at the interface of two fluids of different densities.

Skills Tested:

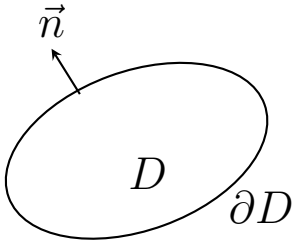
- Applying an integral force balance to a static fluid
- Deriving Archimedes' Law from first principles
- Solving for equilibrium conditions at a fluid interface
- Understanding buoyancy in multiphase systems

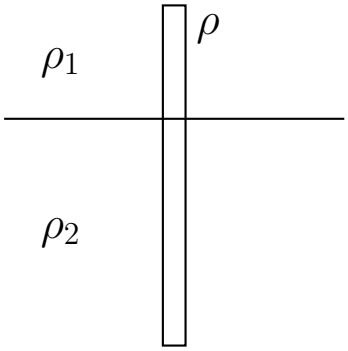
Source: CBE 30355 First Hour Exam, 2017, solved **File:** problem0066.tex

Archimedes and Immersed Cylinder

a. From an integral force balance over the arbitrary object depicted below, derive Archimedes' Law for a fluid at rest.

b. A cylinder of radius R and length L is immersed at the interface of two fluids as depicted below. If the density of the object is ρ, and the densities of the two fluids are ρ_1 and ρ_2 such that $\rho_1 < \rho < \rho_2$, use Archimedes' Law to determine the fraction of the object immersed in the second fluid.





Hydrostatics and Balloon Ascent

Subject Tags: Hydrostatics, Adiabatic Expansion, Atmospheric Physics

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: This problem asks students to compute the final altitude of a balloon that expands adiabatically as it rises through a well-mixed atmosphere. The total mass is constant, and students must apply hydrostatics and the adiabatic gas law to model the equilibrium condition at maximum volume.

Skills Tested:

- Application of hydrostatic pressure balance
- Use of the adiabatic gas law in the atmosphere
- Relating buoyancy to altitude and density
- Interpreting physical constraints in a rising gas balloon system

Source: CBE 30355 First Hour Exam, 2018, solved **File:** problem0074.tex

Hydrostatics and Balloon Ascent

Consider the balloon depicted below. At takeoff the balloon volume is actually fairly small, but as it rises the gas in the balloon expands to fill the envelope, eventually reaching its maximum volume V. The weight of the balloon, gas, and payload is fixed at a value M (assuming no leaks!).

If we make the well-mixed atmosphere approximation made for Lawnchair Larry (good to about 20km), use the equation governing hydrostatics and the adiabatic gas law to calculate the final altitude of the balloon.

Figure 1: Source: https://www.csbf.nasa.gov/balloons.html

Hydrostatics and Archimedes Law for a Straw-Based Hydrometer

Subject Tags: hydrostatics; Archimedes principle; buoyancy; density measurement; salt-water

Difficulty Level: Moderate

Estimated Time: 25–35 minutes total (a: 20 min, b: 10–15 min)

Summary: This problem considers a homemade hydrometer made from a plastic sphere and a hollow straw. Using Archimedes' law and density relations, the task is to determine the straw length required to float the sphere at a known fluid density and then predict the exposed straw height when the fluid density changes.

Skills Tested:

- Application of Archimedes' principle
- Calculation of fluid displacement
- Relating geometry to buoyancy forces

Source: CBE 30355 Midterm Exam, sp2021, solved **File:** problem0093.tex

https://diffusivepress.org/courseware/cbe30355/

Hydrostatics and Archimedes Law

A hydrometer is a classic tool for measuring the salt concentration in a fluid by determining its density (used in salt water fish tanks, for example). You can make one easily by taking a plastic sphere with a density a bit greater than your target density and gluing an empty straw (air filled, so negligible density) to it. The density of the fluid is determined by dropping the hydrometer into the fluid and seeing how much of the straw sticks out of the water. Here we examine such a system.

a. For an acrylic sphere (density of $1.18\,\mathrm{g/cm^3}$) and radius $1.5\,\mathrm{cm}$, what length of straw of radius $0.25\,\mathrm{cm}$ is necessary to measure the density of water ($1.0\,\mathrm{g/cm^3}$)?

b. If our saltwater tank is at the correct salinity, it should have a density of $1.024\,\mathrm{g/cm^3}$. What length of the straw would stick out of the water for this density fluid?

Capillary Rise and Entry: Hydrostatics

Subject Tags: Hydrostatics, Capillarity, Surface Tension, Fluid Statics

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: A narrow tube inserted into a liquid can exhibit capillary rise or resistance to entry depending on the wetting properties of the material. This problem asks the student to derive the capillary rise and entry depth based on a simple force balance involving hydrostatics and capillary pressure. Numerical evaluation is required in part (d) for water and a specified tube size.

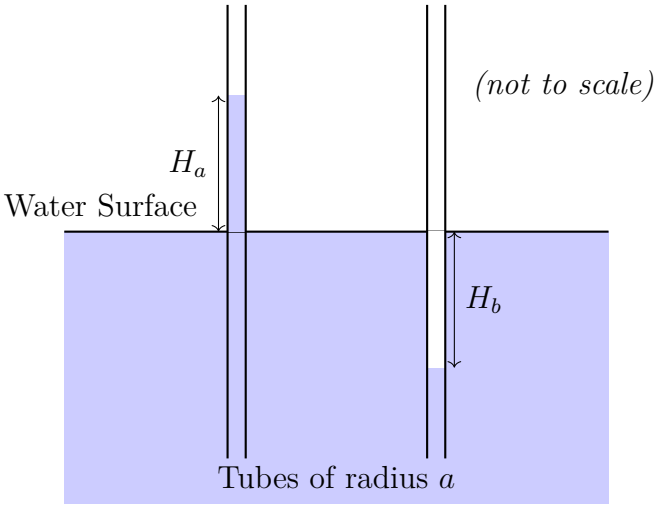
Skills Tested:

- Force balance and pressure head in capillary systems
- Surface tension and capillary pressure concepts
- Estimation using physical constants
- Understanding wetting and non-wetting behavior

Source: CBE 30355 First Hour Exam, 2013 **File:** problem0144.tex

Capillary Behavior in Tubes

Hydrostatics: Consider the system depicted below. A tube of radius a is inserted into a liquid (say, water) of density ρ and viscosity μ as shown below. The surface tension in both cases is given by Γ, and the capillary pressure is given by $\Delta P_{cap} = 2\Gamma/a$.



a. If the tube material is wetted by the water (hydrophilic), the liquid will rise into the tube. What is the equilibrium height of this capillary rise H_a given the parameters of the problem?

b. If the tube interior is coated with a hydrophobic material (say, a water repellent coating), water will resist entry into the capillary. How far below the surface of the water H_b can we stick the tube before water begins to enter?

c. If we insert the tube to twice the depth determined in part (b), what happens?

d. If the fluid is water (the surface tension for clean water is 70 dynes/cm), and the tube radius is 100 μm, what is the magnitude of H_a in cm?

Conservation of Momentum

In this section are gathered one of the larger groups of questions involving conservation of momentum. Essentially, it is looking at all the applications of $F = ma$ for flowing systems, but where we are ignoring all losses (e.g., Bernoulli's equation applies). This includes classic problems such as the force on a nozzle, but also problems like the extraction of energy from a Pelton wheel or the torque on a sprinkler arm.

Conservation of Momentum – Leaky Tank

Subject Tags: fluid mechanics, Bernoulli, conservation laws, momentum balance

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: A tank filled with water is punctured halfway up its height. This problem uses conservation of energy and momentum to determine the fluid exit velocity, the initial draining rate, and the horizontal range of the water jet. Simplifying assumptions include negligible viscosity and energy losses.

Skills Tested:

- Application of Bernoulli's equation
- Transient tank drainage analysis
- Projectile motion from a fluid jet

Source: CBE 30355 Midterm Exam, 2021, solved **File:** problem0004.tex

`https://diffusivepress.org/courseware/cbe30355/`

Conservation of Momentum: Leaky Tank

Consider the tank of radius R filled with water to a height H. The top is open to the atmosphere. A small hole of radius $a \ll R$ is poked in the side at height $z = H/2$, as shown in the figure. Ignoring all losses, determine the following:

a. What is the velocity of the fluid out the hole?

b. What is the initial rate of change of the fill height of the tank?

c. How far to the side does the stream project before it hits the ground?

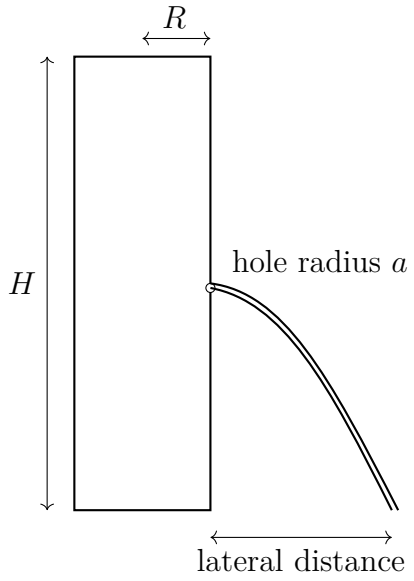


Figure 2: Schematic of tank with hole and fluid stream

Force on a Hose Nozzle

Subject Tags: control volume, momentum balance, Bernoulli equation, fluid force

Difficulty Level: Moderate

Estimated Time: 30 minutes

This problem involves the use of control volume analysis to determine the force exerted by water flowing through a hose nozzle. The setup involves differing inlet and outlet diameters and the use of pressure and velocity data. You will derive a general force expression, simplify it using assumptions, and apply Bernoulli's equation for a final quantitative result.

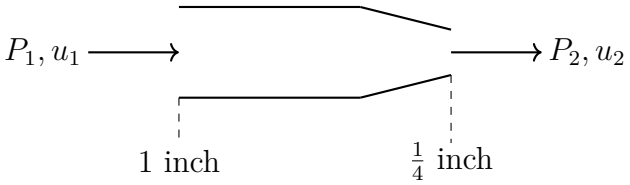
Skills Tested:

- Control volume momentum analysis
- Simplification using physical assumptions
- Application of Bernoulli's equation

Source: CBE 30355 1st Hour Exam, 2001 **File:** problem0007.tex

Control Volume Analysis of a Hose Nozzle

Consider the hose nozzle depicted below. You are asked to determine the force exerted by the fluid on the nozzle in order that the hose-nozzle coupling may be properly designed. The inlet to the nozzle is 1 inch ID, the outlet (and emerging jet) is $\frac{1}{4}$ inch ID, and the water pressure in the hose before the nozzle is 40 psig ($2.75 \times 10^5\,\mathrm{N/m^2}$ above atmospheric pressure). Neglect any forces due to gravity.



a. Develop an equation for the force on the nozzle in terms of u_1, u_2, P_1, P_2, and the two areas. What key assumption are you making here?

b. Taking $P_2 = 0$ psig and employing conservation of mass, reduce the above result to one involving only P_1, u_1, and the areas.

c. If we neglect frictional losses of mechanical energy, the velocities and pressures are governed by Bernoulli's equation:

$$\frac{1}{2}\rho u_1^2 + P_1 = \frac{1}{2}\rho u_2^2 + P_2$$

Use this relation to get a quantitative answer for the force on the nozzle.

Flow in a Nozzle from a Pressurized Tank

Subject Tags: Inviscid Flow; Continuity; Pressure Distribution; Bernoulli Equation

Difficulty: Moderate

Estimated Time: 20 minutes

Summary: This problem involves water flowing from a pressurized tank through a hose and out a nozzle. The flow is assumed to be inviscid and irrotational, which allows for the use of the continuity and Bernoulli equations to relate the velocity and pressure distributions to the volumetric flow rate. The goal is to determine the average velocity distribution in the nozzle and evaluate the pressure at the tank outlet.

Skills Tested:

- Application of Continuity Equation
- Use of Bernoulli's Equation under simplifying assumptions
- Pressure-velocity relationship in inviscid flows
- Conceptual understanding of flow uniformity and irrotational conditions

Source: CBE 30355 2nd Hour Exam, 2001 **File:** problem0009.tex

Flow in a Nozzle from a Pressurized Tank

Water from a pressurized tank passes through a hose and out a nozzle as is depicted below.

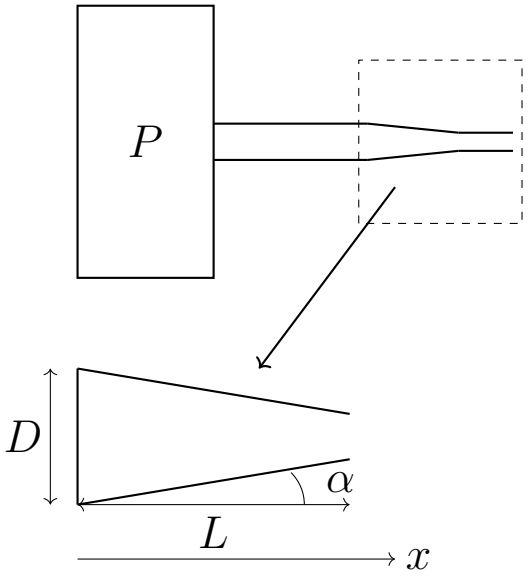


Figure 3: Schematic of pressurized tank and nozzle flow

a. If we assume that the flow is inviscid and irrotational (no frictional losses), use the continuity equation to determine the average velocity as a function of position in the nozzle and estimate the pressure distribution as a function of the flow rate Q. You may assume that the flow in the nozzle is uniform, i.e., that it does not vary radially (perfect slip at the walls), but rather only with the distance down the nozzle.

b. What is the pressure in the tank at the level of the hose outlet?

Momentum Balance in Wake Traverse Experiment

Subject Tags: momentum balance; control volume; experimental fluid mechanics; drag force

Difficulty: Moderate

Estimated Time: 35 minutes

Summary: This problem analyzes an experiment from the Senior Laboratory involving flow past a cylinder at high Reynolds number. A wake traverse technique is used to measure upstream and downstream flow conditions. Students apply the integral momentum balance over a control volume surrounding the cylinder to determine the drag force per unit length. The velocity profile downstream is non-uniform and must be included as an integral in the analysis.

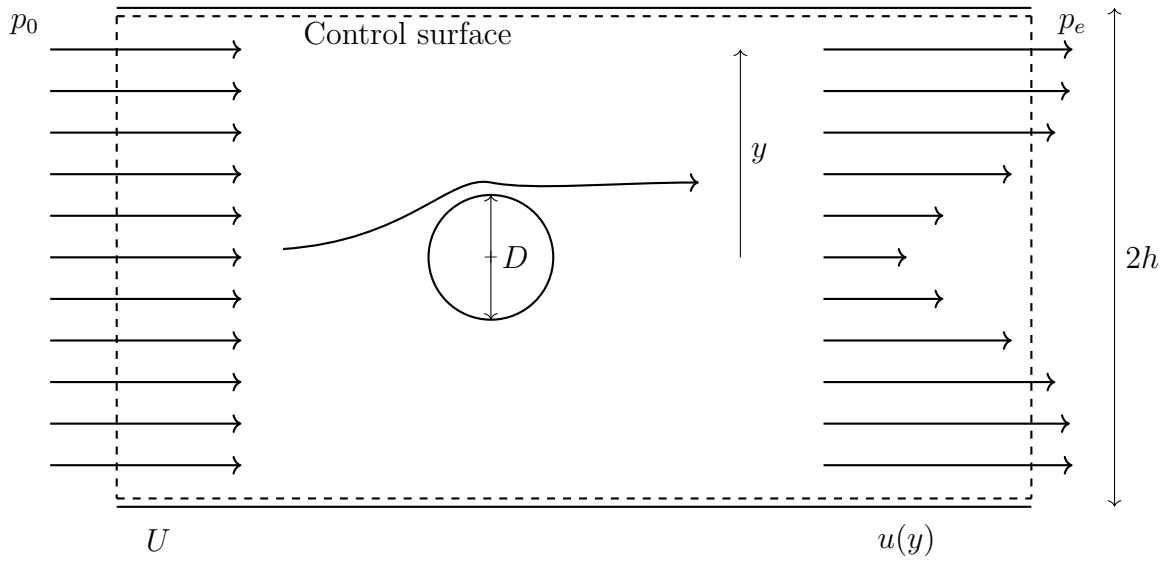
Skills Tested:

- Formulating control volume momentum balances
- Working with pressure and velocity boundary conditions
- Interpreting non-uniform outlet velocity profiles
- Connecting experimental measurements to theoretical models

Source: CBE 30355 First Hour Exam, 2002 **File:** problem0019.tex

Momentum Balance in Wake Traverse Experiment

In Senior Laboratory one of the experiments is measuring the drag on a cylinder at high Reynolds number. One of the techniques used is called a *wake traverse*, in which we measure both the uniform velocity U upstream of the cylinder, and the non-uniform velocity $u(y)$ downstream of the cylinder. We also measure the upstream pressure p_0 and the downstream pressure p_e. The diagram from the lab manual is given below.



a. Write down in words the integral momentum balance around the control surface depicted above.

b. Given that we can neglect any drag on the side walls of the control surface, develop an explicit integral expression for the drag per unit length (into the paper) D exerted by the fluid on the cylinder in terms of p_0, p_e, U, and $u(y)$. Both p_0 and p_e may be taken to be constant over the inlet and outlet, respectively.

Hint: This problem is pretty much identical to the sluice gate problem you solved for homework, except here the upstream and downstream pressures are measured (no gravity), and the downstream velocity profile is non-uniform, and thus must be left in integral form in the momentum balance. In lab it's this downstream velocity measurement that's the tricky part of the experiment...

Jet Deflection Supporting a Mass

Subject Tags: fluid mechanics, momentum conservation, jet deflection, control volume

Difficulty: Moderate

Estimated Time: 25 minutes

Summary: This problem involves using an integral momentum balance to determine the equilibrium height of a plate deflected by a curved water jet. The analysis incorporates the volumetric flow rate, fluid density, change in momentum direction, and Bernoulli's equation to relate flow variables and gravitational force on the mass.

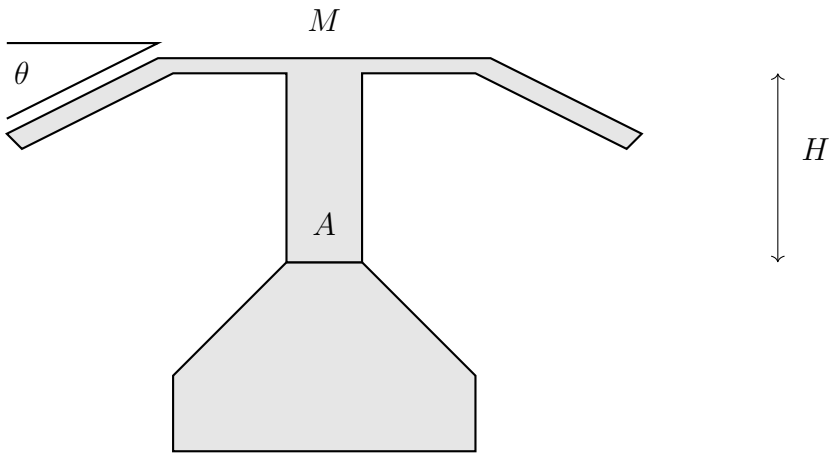
Skills Tested:

- Control volume momentum balances
- Application of Bernoulli's equation
- Free body diagram interpretation
- Geometric and trigonometric resolution of forces

Source: CBE 30355 First Hour Exam, 2011, solved **File:** problem0030.tex

Jet Deflection Supporting a Mass

Consider the plate and mass depicted below. The mass M is supported by the deflection of a jet of water of volumetric flow rate Q and density ρ. The jet emerges from a nozzle of cross-sectional area A, yielding some initial velocity U_0. The curvature of the plate deflects the stream backwards by some angle θ. Using all this (and an integral momentum balance!) calculate the equilibrium height of the plate. Note that the jet slows down as it moves upwards: think of Bernoulli's Equation!



Momentum Balances and Bernoulli's Equation

Subject Tags: Momentum Balance, Bernoulli Equation, Jet Deflection, Control Volume Analysis

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: This problem involves applying Bernoulli's equation and an integral momentum balance to a deflected fluid jet that supports a mass. Given a curved plate and a water jet emerging from a nozzle, you are to determine the velocity of the jet, the pressure in the supply pipe, and the equilibrium height at which the plate is held aloft by the jet.

Skills Tested:

- Application of Bernoulli's equation in a control volume
- Use of momentum balances in deflected jets
- Analysis of jet-plate interactions

Source: CBE 30355 First Hour Exam, 2018, solved **File:** problem0075.tex

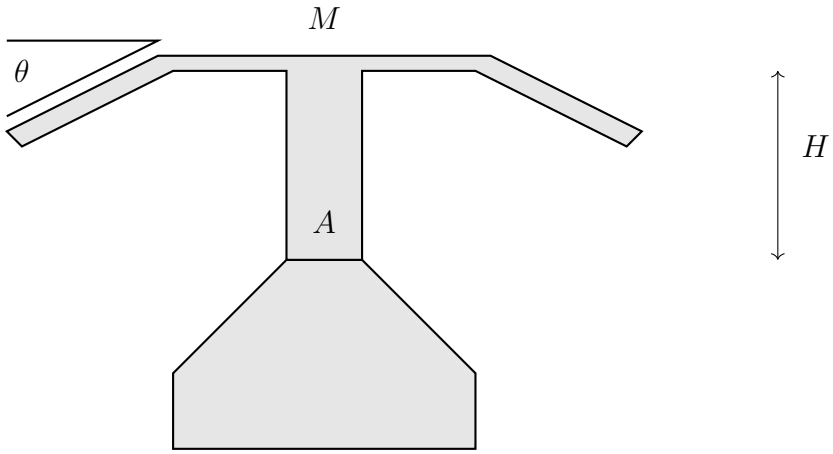
Momentum Balances and Bernoulli's Equation

Consider the plate and mass depicted below. The mass M is supported by the deflection of a jet of water of volumetric flow rate Q and density ρ. The jet emerges from a nozzle of cross-sectional area A. The curvature of the plate deflects the stream backwards by some angle θ.

a. Determine the velocity of the jet at the nozzle U_0.

b. If the pipe supplying the nozzle is really fat (cross-sectional area much greater than A), what is the pressure in the pipe?

c. Using all the provided information (and an integral momentum balance!) calculate the equilibrium height of the plate. Note that the jet slows down as it moves upwards!



Torque on a Stationary Sprinkler

Subject Tags: Integral Momentum Balance; Angular Momentum; Fluid Jets

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: This problem applies the integral momentum balance to a steady-state, incompressible flow through a stationary rotating sprinkler. The goal is to compute the torque exerted on the sprinkler hub due to the reaction forces from the jet outlets.

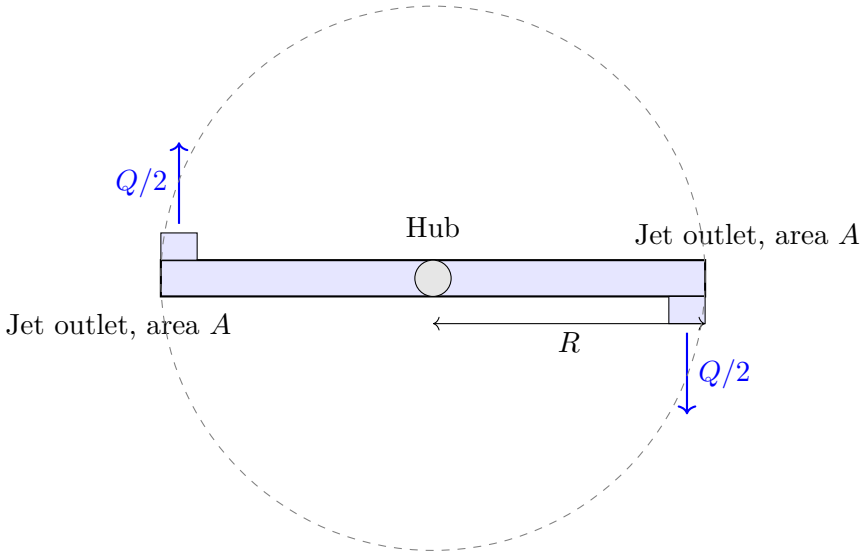
Skills Tested:

- Application of integral angular momentum balance
- Control volume selection and analysis
- Torque and force calculations from fluid flow

Source: CBE 30355 First Hour Exam, 2012, solved **File:** problem0042.tex

Torque from Jet Deflection in a Sprinkler

Consider the sprinkler depicted below. The radius of the arms is R, the area of the jet outlet is A (for each arm), and the total flow rate of water is Q (i.e., $Q/2$ through each arm). Using an integral momentum balance, determine the torque exerted on the center of the sprinkler if the arms are stationary.



Jetlev Flyer Momentum Balance

Subject Tags: Bernoulli, Fluid Mechanics, Control Volume, Pressure Drop, Conservation of Momentum

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: In this problem, students analyze the flow rate and required pump pressure to operate a water-propelled jetpack known as the Jetlev Flyer. The analysis is based on the conservation of momentum and Bernoulli's equation, and includes the weight of the flyer, equipment, and water in the feed tube. Students neglect frictional losses to simplify calculations and isolate the primary fluid dynamic forces.

Skills Tested:

- Application of Bernoulli's equation
- Conservation of momentum for control volumes
- Estimating weight of fluid in conduits
- Force balance on systems involving fluid jets

Source: CBE 30355 First Hour Exam, 2015, solved **File:** problem0051.tex

https://diffusivepress.org/courseware/cbe30355/

Jetlev Flyer Momentum Balance

In a fun *WSJ* article on water toys for the super-rich, I came across the Jetlev Flyer. It consists of a small boat/pumping unit, a 10 m long 4" ID feed tube, and a jet backpack which redirects the water downwards through two 1.5" ID nozzles. According to their advertisements, you can fly for hours (before the modified jetski boat pump runs out of gas), and travel up to 25 mph towing the pump boat behind you. Although more than a bit pricey (it costs around the same as a new car!) you can actually get a 15-minute ride on one for less than $200 at a bunch of places. Here we analyze the requirements for the pumping system.

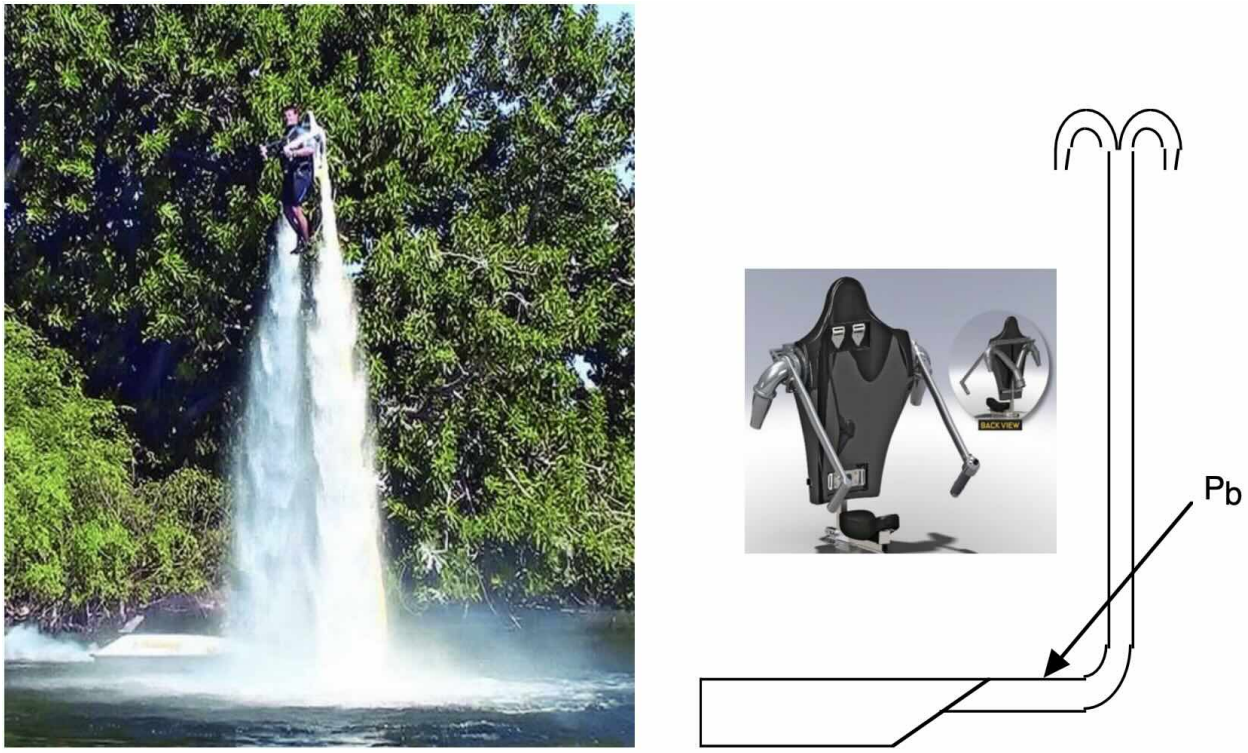


Figure 4: Source: https://jetlev-flyer.com

Given data:

- Flyer weight: 100 kg
- Jetpack weight (water filled): 10 kg
- Feed pipe dry weight: 0.25 kg/m
- Feed pipe length: 10 m
- Feed pipe ID: 4 inches
- Nozzle diameter: 1.5 inches each (two nozzles)
- Maximum altitude: 10 m

Using this information, and neglecting all frictional losses (e.g., assume that Bernoulli's equation applies), determine:

a. The required flow rate

b. The pressure at the base of the feed pipe (e.g., the pressure the pump would have to supply) to support our flyer at the maximum altitude of 10 m.

Don't forget to include the weight of the water in the feed pipe! While the flow rate you calculate is about right, the required pressure (and pump power rating) would be significantly higher due to losses, as we shall study later this term.

Jetlev Flyer Nozzle Optimization

Subject Tags: Jet Propulsion, Bernoulli Equation, Dimensional Analysis, Power Optimization

Difficulty Level: Advanced

Estimated Time: 40 minutes

Summary: This problem explores the optimization of nozzle diameter for a Jetlev Flyer propulsion system. Given fixed system mass and pipe dimensions, students must analyze how flow rate and power requirements vary with nozzle size and derive a relation for the optimal configuration to achieve maximum height with minimum energy expenditure.

Skills Tested:

- Scaling Laws
- Energy Balances
- Optimization
- Dimensional Reasoning

Source: CBE 30355 Final Exam, 2015 **File:** problem0155.tex

Jetlev Flyer Nozzle Optimization

Given your familiarity with the Jetlev Flyer geometry, you are asked to determine the optimum jet nozzle diameter. The supply pipe diameter is D_p (e.g., pipe area $A_p = \pi D_p^2/4$) and the total mass to be supported (hose, flyer, water, etc.) is M_T (both are fixed). We seek to determine the optimum nozzle diameter D_N (e.g., total nozzle area $A_N = 2 \cdot \pi D_N^2/4$, as there are two nozzles!) that will yield a flyer height H. The frictional losses in the system (pipe friction factor and fitting losses) are approximately given by:

$$h_L = \frac{1}{2}\rho \left(\frac{Q}{A_p}\right)^2 \frac{3.4}{\rho g}$$

a. How does the flow rate Q depend on the nozzle diameter?

b. How does the power required depend on the nozzle diameter?

c. How can you use the results of part **b** to get an equation governing the optimum nozzle diameter for achieving the desired height at minimum power?

(Note: It is possible to show that the ratio D_N/D_p is a simple function of the dimensionless ratio $\dfrac{M_T}{\rho H A_p}$, which, for the flyer, reduces to $D_N/D_p \sim 0.4$, but this takes a bit too much time to do on an exam, alas...)

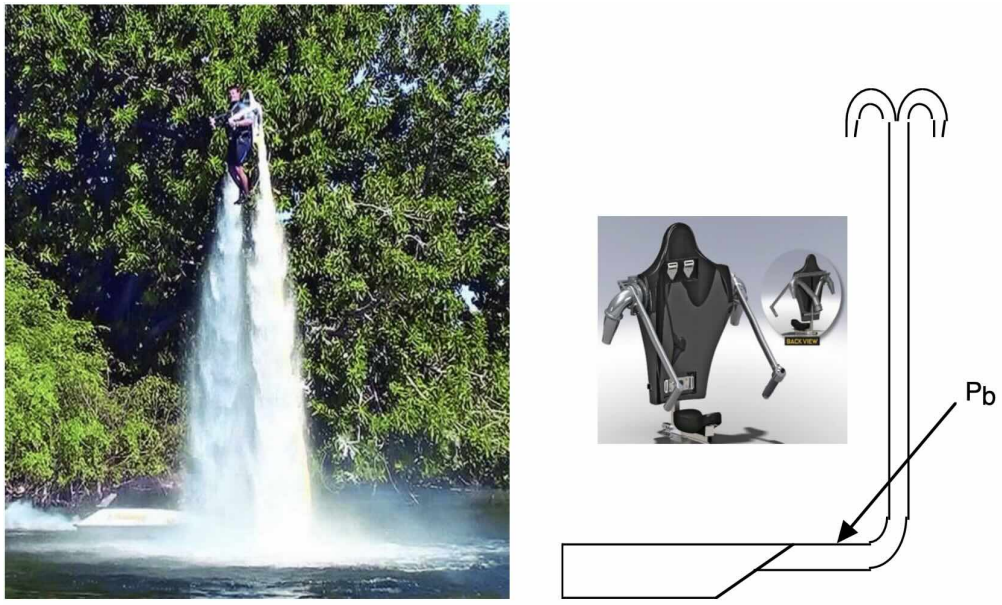


Figure 5: Source: https://jetlev-flyer.com

Integral Momentum Balance on a Fountain System

Subject Tags: integral balances, momentum, jets, support forces

Difficulty: Moderate

Estimated Time: 30 minutes

Summary: This problem asks the student to analyze a gravity-driven fountain using control volume techniques. By applying the integral momentum balance and neglecting energy losses, students derive expressions for the volumetric flow rate and the floating height of a mass supported by a jet. The final result connects the fountain height to physical parameters like mass, geometry, and flow conditions.

Skills Tested:

- Use of integral momentum balance
- Steady-state analysis of fluid forces
- Relation of physical forces to geometry
- Handling simplifying assumptions effectively

Source: CBE 30355 First Hour Exam, 2017, solved **File:** problem0064.tex

https://diffusivepress.org/courseware/cbe30355/

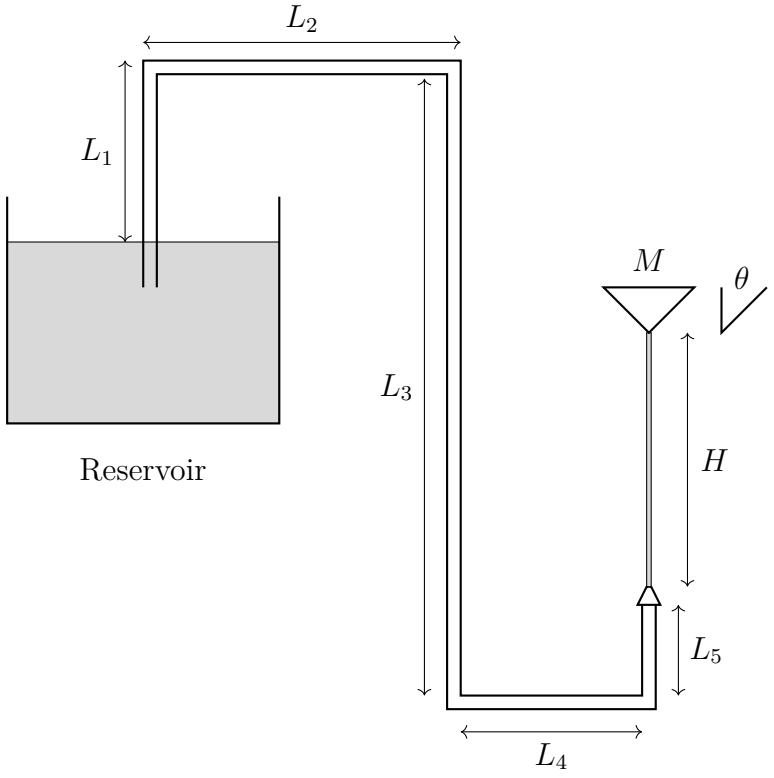
Integral Momentum Balance on a Fountain System

Consider the gravity-driven fountain depicted below. Water from a reservoir travels through a pipe to feed a fountain with a nozzle of area A. The water from the jet then supports a mass M, where the "deflector" is a cone with internal angle 2θ. The objective here is to analyze this fountain and figure out how high the mass floats above the fountain nozzle.

a. Neglecting all losses (actually not a bad approximation if the nozzle area is a lot smaller than the pipe diameter), develop an expression for the flow rate of the fluid through the jet.

b. If M is really small, what is the maximum height it can reach at steady-state?

c. Using an integral momentum balance, develop an expression for the height of the mass M as a function of the cone angle and the other parameters in the problem.



Integral Momentum Balances – Vertical Water Jet Supporting a Mass

Subject Tags: integral momentum balance; vertical jet; force balance; Bernoulli's equation; minimum flow rate

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: In this problem, a vertical jet of water supports a plate with mass M. The student must apply an integral momentum balance to calculate the steady-state height of the plate, given the flow rate and nozzle area, and determine the minimum flow rate necessary to support the mass.

Skills Tested:

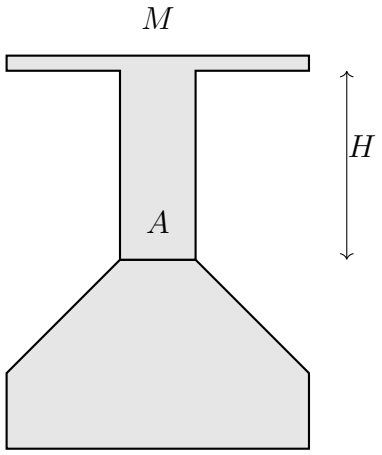
- Applying control volume momentum balance
- Using Bernoulli's equation to relate velocity and pressure
- Evaluating steady-state equilibrium conditions
- Relating flow rate and jet dynamics to support a force

Source: CBE 30355 First Hour Exam, 2019, solved **File:** problem0084.tex

https://diffusivepress.org/courseware/cbe30355/

Integral Momentum Balances – Vertical Water Jet Supporting a Mass

A mass M is sitting on a flat plate as depicted below, balanced on a vertical jet of water. If the volumetric flow rate of the water jet is Q and the area of the jet nozzle is A, we can do a force balance on the plate. Don't forget that the water slows down and spreads out as it travels upwards, but the flow rate is the same! Neglecting all losses (e.g., use Bernoulli's Equation!), calculate the steady-state height H of the plate, and determine the minimum flow rate for which the plate can be supported.



Hydroelectric Power Generation on the St. Joseph River

Subject Tags: hydroelectricity; energy conversion; fluid mechanics; power output; unit conversion

Difficulty Level: Moderate

Estimated Time: 30–40 minutes total (a: 20–25 min, b: 10–15 min)

Summary: This problem involves estimating the electrical power output of a hydroelectric installation on the St. Joseph River, considering flow rate, turbine efficiency, and head height. Students must apply energy conservation principles and perform proper unit conversions. The second part requires cost-benefit estimation over a year using typical electricity prices.

Skills Tested:

- Fluid flow rate analysis and energy per unit time
- Application of gravitational potential energy
- Unit conversions (cfs to kg/s, Watts to MW, etc.)
- Estimating annual energy production and monetary value

Source: CBE 30355 Midterm Exam, sp2021, solved **File:** problem0094.tex

Hydroelectric Power Generation on the St. Joseph River

The University is installing a hydroelectric generation system on the St. Joseph River at Seitz Park on the East Race (you may have noticed all the construction down there). The amount of power that can be generated depends on the flow rate, the total head (change in elevation), and turbine efficiency. The average flow rate of the river is 3000 cfs (cubic feet per second), of which 600 cfs in total is reserved for the waterfall, East and West races, and fish ladder (e.g., other purposes, so you have to subtract this off!). The change in elevation is approximately 3 m.

a. If the turbine efficiency is 80% (a typical value for this size installation), what is the expected electrical power output of the system in MW? (Unit conversion: 1 cubic ft = 28.3 liters). **(Hint: think of the available energy/time from the fluid flow!)**

b. The current cost of electricity in Indiana is $0.12/kWh. Based on this, what would be the expected value to the University of the electricity produced in one year?

Conservation of Momentum and Bernoulli Analysis for Firefighting Nozzle

Subject Tags: Bernoulli equation; mechanical energy; fluid dynamics; firefighting; nozzle optimization

Difficulty Level: Moderate

Estimated Time: 20–30 minutes (a: 15 min, b: 5–10 min)

Summary: This problem applies the Bernoulli equation and conservation of energy to determine the optimal nozzle diameter for maximum water flow in firefighting applications. Neglecting all losses, the first part uses energy constraints to compute the nozzle diameter that achieves maximum volumetric flow rate. The second part requires a conceptual understanding of how losses affect the system and how that alters the optimal diameter.

- Application of Bernoulli's equation
- Power-to-flow rate relationships
- Optimal diameter determination
- Qualitative impact of losses

Source: CBE 30355 Midterm Exam, 2022, solved **File:** problem0096.tex

Conservation of Momentum and Bernoulli Analysis for Firefighting Nozzle

You are trying to put out a fire in a building at a height H. You have a pump which is capable of supplying water with a useful mechanical energy/time (e.g., effective pumping power) of P_W. For this problem, neglect all losses (e.g., use Bernoulli's Equation!).

a. You want to put the fire out quickly, so you want a large flow rate Q. For the parameters of the problem, what diameter nozzle D will achieve the maximum flow rate?

b. There are always losses, and you still need to put out the fire! Should you use a nozzle larger or smaller than the one calculated in part a? Briefly justify your answer.

Conservation of Momentum in the DART Asteroid Test

Subject Tags: momentum conservation; elastic collision; inelastic collision; asteroid deflection; impact dynamics

Difficulty Level: Moderate

Estimated Time: 40–45 minutes total (a: 10 min, b: 10 min, c: 20–25 min)

Summary: This problem applies the principle of momentum conservation to analyze the outcome of a kinetic impactor (NASA's DART mission) striking an asteroid under three different interaction scenarios: completely inelastic, elastic rebound, and high-energy ejecta-driven recoil. Each case explores how much velocity change is imparted to the asteroid and illustrates the long-term potential of small changes in trajectory.

Skills Tested:

- Momentum conservation in 1D systems
- Analysis of inelastic and elastic collisions
- Application of kinetic energy principles to ejecta dynamics

Source: CBE 30355 Midterm Exam, 2022, solved **File:** problem0097.tex

https://diffusivepress.org/courseware/cbe30355/

Conservation of Momentum in the DART Asteroid Test

Just a couple weeks ago NASA tested a concept for potentially saving the Earth: using a kinetic impactor to "nudge" an asteroid on a collision course with the planet. The particular asteroid (Dimorphos) is no danger to us, but it provides a very clever way to test the concept. This isn't fluid mechanics (although there certainly was some granular flow!) but it *is* conservation of momentum. In this problem we will examine the Double Asteroid Redirection Test.

The kinetic impactor had a mass of $m_d = 500\,\text{kg}$ and the target had a mass $M = 5 \times 10^9\,\text{kg}$, so the ratio is 10^7. The relative velocity of the impactor was $U_d = 6600\,\text{m/s}$, so the energy of impact E was pretty big ($1.1 \times 10^{10}\,\text{J}$, or about 3 tons of high explosive). Using this, calculate the change in velocity ΔU of the asteroid target under the following scenarios:

a. The impactor is simply swallowed up and absorbed by Dimorphos.

b. The impactor bounces off (backwards) as if it were a perfectly elastic collision (note: there's no way this would happen at 6.6 km/s – 15,000 mph!).

c. The huge energy of the collision causes a crater to form which ejects a mass m_e 1000 times greater than the mass of the impactor. While this calculation is really complex (and is why they spent $350M to do the test!) for this problem assume that the impact energy E is converted to the kinetic energy of the ejected mass, that all the ejected material has the same velocity, and that it is all traveling back in the direction the impactor came from with a velocity much greater than the escape velocity of the asteroid (which is very small – it's not that big a rock!). Note that these are all rather optimistic assumptions (it effectively represents the upper bound), but it does illustrate the effect they are looking for.

While all the velocities you calculate are really small, if you were to do it to an asteroid that you detected enough years in advance of impact (they're trying to map all earth-crossing asteroids for this very reason!) you could actually deflect it enough to miss the earth.

Conservation of Momentum for Asteroid Deflection

Subject Tags: Momentum Conservation, Asteroid Deflection, Nuclear Explosion, Energy Conversion

Difficulty: Moderate

Estimated Time: 30 minutes

This problem explores the physics of asteroid deflection using a nuclear explosion, focusing on momentum conservation. It involves drawing a control volume sketch, relating energy yield to velocity change of the asteroid, and comparing strategies for deploying the nuclear device.

Skills Tested:

- Sketching control volumes for physical systems
- Applying momentum conservation principles
- Relating kinetic energy to velocity change
- Analyzing and comparing deflection strategies quantitatively

Source: CBE 30355 Midterm Exam, 2024, solved File: problem0104.tex

Conservation of Momentum for Asteroid Deflection

Dinosaur killing asteroid deflection has been much in the news lately. While there are many options if you find it early enough, if it is picked up too late you basically have to use nuclear weapons to give you enough change in momentum to do any good (and even that's hard). Suppose the energy yield of your weapon is E.

You have a couple of options for how you can use it: You can blow it up on the surface, giving a small mass of ejected material M_e a very high velocity, or you can blow it up deeper in (e.g., ground penetrating charge, or land and bury it) and eject a larger mass with a lower velocity.

In this problem you are going to examine these options.

a. Draw a sketch of your system surrounded by a control volume.

b. The total momentum of the system is conserved. If all the energy of the explosion is converted to the kinetic energy of the ejected material, and if it is all ejected in the same direction, how does the change in velocity of the asteroid of mass $M_a \gg M_e$ depend on the parameters? (Note: we will ignore the gravitational attraction of the asteroid, assuming that the ejected velocity is much greater than the escape velocity – this would actually place a constraint on M_e.)

c. Based on your expressions, what is the better way to use your nuke?

Thinning Water Stream from Faucet

Subject Tags: Bernoulli's Equation; Momentum Balance; Fluid Jets; Free Jets

Difficulty Level: Moderate

Estimated Time: 25 minutes (10 for part a, 15 for part b)

Summary:
This problem examines a common but revealing phenomenon: the steady thinning of a laminar stream of water falling from a faucet. Students are asked to apply Bernoulli's equation and the continuity equation to determine how the jet diameter changes with vertical distance. In the second part, they apply the integral momentum theorem to calculate the force exerted by the falling stream on a cup below, requiring simplifications and physical insight.

Skills Tested:

- Application of Bernoulli's equation to vertical flow
- Use of the continuity equation in a free jet
- Control volume analysis using the integral momentum equation

Source: CBE 30355 Midterm Exam, 2023, solved **File:** problem0099.tex

Thinning Water Stream from Faucet

You've all seen it: a laminar stream of water emerges from a water faucet and, as it falls downwards, gets thinner and thinner. Here we analyze this problem.

a. If the initial velocity of the water emerging from the faucet is U_0 and the stream diameter is D_0, what is the diameter of the stream when it reaches a cup a distance H below the faucet?

b. What is the force exerted by the momentum of the stream on the (overflowing) cup? Make any simplifications you deem necessary, but justify them.

Persian Water Clock and Sinking Time Analysis

Subject Tags: Hydrostatics; Archimedes' Law; Bernoulli's Equation; Fluid Inflow; Transient Flow

Difficulty Level: Moderate

Estimated Time: 40 minutes (10 for Bernoulli, 10 for differential equation, 20 for solution)

Summary:
This problem models an ancient Persian water clock composed of a massless cylindrical can with a small hole in the bottom and a weight attached underneath. The system is driven by hydrostatic pressure differences, and the inflow through the hole is governed by Bernoulli's equation. The goal is to derive the velocity of inflow, formulate and solve a differential equation for the time-dependent height, and determine the sinking time, accounting for frictional losses via a discharge coefficient.

Skills Tested:

- Application of Bernoulli's equation to inflow velocity
- Modeling pressure differences due to hydrostatics and weight
- Formulation of an ODE for the time-varying height
- Integration of transient flow equations

Source: CBE 30355 Midterm Exam, 2024, solved **File:** problem0101.tex

Persian Water Clock and Sinking Time Analysis

Time keeping has played an important role in human civilization, and many techniques have been developed over the millennia. An early approach was the Persian Water Clock, in which a bowl with a small hole in the bottom was floated in a reservoir. When the bowl filled sufficiently it abruptly sank, with the sinking time providing one measured unit of time (an attendant would then refill it and keep track of the number of times it was repeated). Here we analyze such a system.

Consider the massless cylindrical can of radius R and height H depicted below (in this case a tennis ball can). A small hole of radius R_0 is drilled into the bottom of the can, and a mass $M_w = \rho_w V_w$ of density ρ_w and volume V_w is attached to the bottom. Because of the weight, the water level in the can is a height Δh below the level in the surrounding reservoir (in this case a glass vase). This causes water (density ρ) to flow into the can through the hole in the bottom. The top of the can projects a distance h above the water level in the reservoir, and this will decrease as the can fills.

a. How does the inflow velocity U depend on the parameters of the problem? Assume that there are no losses in the system.

b. Write down a differential equation and boundary conditions for how the projecting height h varies with time.

c. Solve the problem to determine the sinking time. Note that this works quite well, although the time needs to be adjusted by the inverse of the discharge coefficient (about 0.6) to account for frictional losses in the hole.

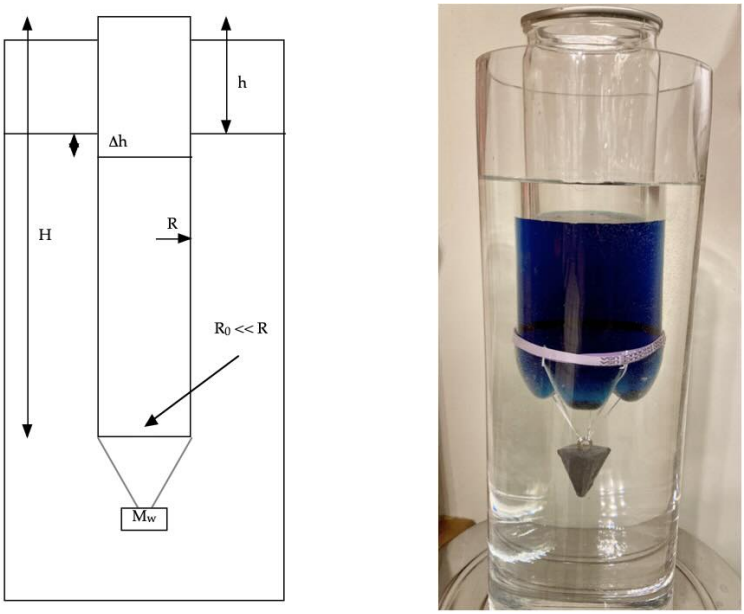


Figure 6: The Persian Water Clock

Pelton Wheel Energy Extraction

Subject Tags: Turbines, Momentum Balance, Energy Efficiency, Control Volume Analysis

Difficulty: Moderate

Estimated Time: 40 minutes

This problem analyzes the mechanical energy transfer in a Pelton wheel where water jets strike the vanes of a rotating wheel. By applying a control volume approach and using conservation of momentum, the optimal operating condition for maximum power extraction is identified. The corresponding mechanical efficiency is calculated, and the fate of the non-recovered energy is discussed.

- Application of control volume analysis
- Linear momentum conservation in rotating systems
- Determination of optimal angular velocity
- Efficiency assessment and energy loss interpretation

Source: CBE 30355 First Hour Exam, 2004 **File:** problem0113.tex

Maximum Power and Efficiency of a Pelton Wheel

Consider the Pelton wheel, the simple turbine discussed in class, depicted below. A water jet moving with velocity U_j and volumetric flow rate Q impinges on the vanes of a wheel of radius R turning with angular velocity Ω.

If the collision of the water with the vane is completely inelastic (e.g., it leaves with a velocity equal to the velocity of the vane, rather than bouncing off), determine the conditions under which the rate of work done on the wheel is a maximum. What is the efficiency of mechanical energy utilization under these conditions? What happens to the lost mechanical energy?

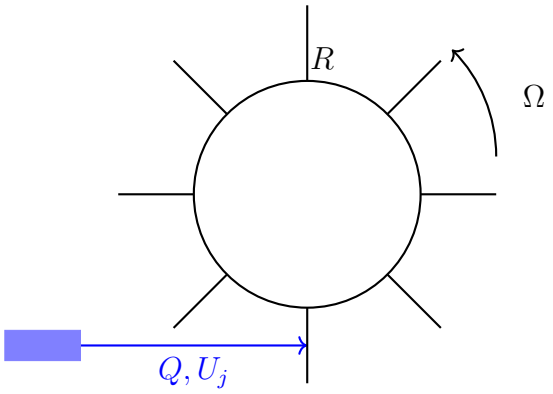


Figure 7: Pelton wheel with incoming water jet.

Hint: This problem is very easy if you draw a control volume surrounding the wheel, and properly account for convection of momentum in and out!

Vacuum Cannon Projectile Velocity

Tags: Unsteady Flow, Pressure-Driven Acceleration, Gas Dynamics, Momentum Conservation

Difficulty: Moderate

Estimated Time: 30 minutes

A popular student demonstration is the vacuum cannon, in which a projectile is placed in a sealed and evacuated tube. When one end of the tube is opened, atmospheric pressure accelerates the projectile along the tube. This problem explores the maximum velocity that can be achieved under ideal and non-ideal conditions.

Skills Tested:

- Energy and Momentum Balances
- Idealized Modeling
- Dimensional Analysis

Source: CBE 30355 First Hour Exam, 2009 **File:** problem0134.tex

Vacuum Cannon Projectile Velocity

A popular student demonstration is the vacuum cannon, depicted below. A projectile of mass M is placed in a tube of radius R and length L, sealed with a layer of tape at each end and evacuated using a vacuum pump. The cannon is activated by popping the seal at one end, allowing atmospheric pressure to drive the projectile out of the other end at high velocity.

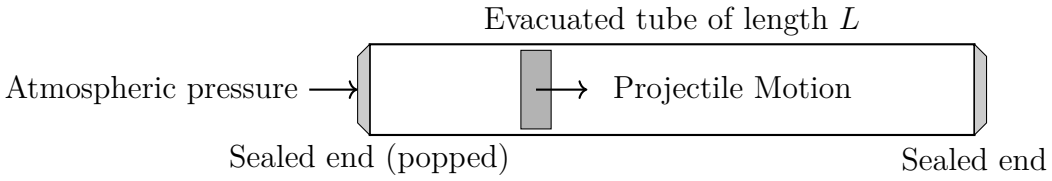


Figure 8: Schematic of vacuum cannon.

a. Neglecting all losses (and the inertia of the air), what is the maximum possible velocity of the projectile at exit?

b. If the atmospheric pressure is 10^6 dyne/cm^2, the tube radius is 2.54 cm, the length is 100 cm and the mass is 1 kg, what is the numerical value of this velocity?

c. Estimate the maximum possible velocity of a very small mass projectile.
(Hint: you have to include the inertia of the air! Take the density of air to be 1.2×10^{-3} *g/cm*^{3}*.)*

Undershot Waterwheel Pumping from the Edisto River

Subject Tags: Control Volume Analysis, Mechanical Power, Hydraulics, Torque Balance

Difficulty Level: Moderate

Estimated Time: 45 minutes

Summary: This problem considers an undershot waterwheel located in Edisto Memorial Gardens in South Carolina, used to pump water from a river into a nearby pond. Flowing water pushes against the vanes, turning the wheel and lifting water via buckets. The problem requires deriving an approximate flow rate expression based on assumptions, then optimizing the total bucket volume to maximize efficiency.

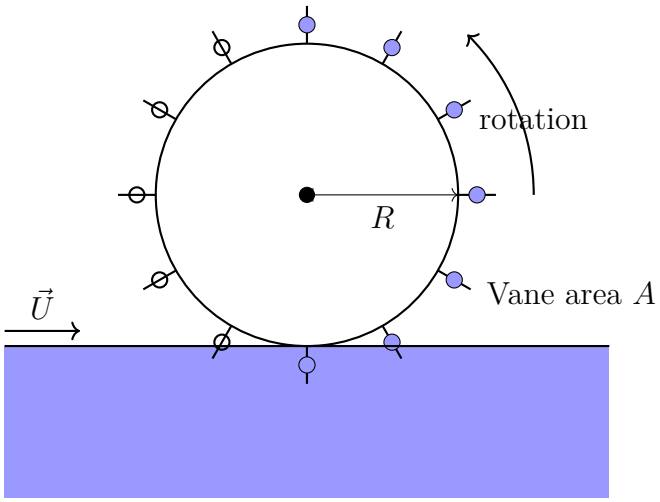
Skills Tested:

- Force and momentum analysis on vanes
- Power extraction from flow
- Torque balance and mechanical constraints
- Optimization under physical assumptions

Source: CBE 30355 First Hour Exam, 2010 **File:** problem0137.tex

Undershot Waterwheel Pumping from the Edisto River

Off I-26 in Orangeburg, SC (on the route to Hilton Head) is Edisto Memorial Gardens. Probably the most photographed feature of the gardens is a 70-year-old waterwheel that was once used to pump water from the slow-moving Edisto blackwater river to a nearby pond. Such undershot waterwheels were often used to irrigate rice paddies, for example. The operation is simple: the flowing water applies a force to the vanes of area A, causing the wheel to turn. As each vane enters the water, a bucket fills with water and then empties at the top of the arc. Your mission is to determine the flow rate and optimal bucket size.



Undershot Waterwheel in the Edisto River

a. This problem isn't as "clean" mathematically as the idealized Pelton wheel solved in class, but the physical principle is the same. It is easiest to work with the total volume of all the buckets V_T, as then the number of vanes doesn't play a large role (assuming there are a lot of them, and one of area A is perpendicular to the stream at all times). Making any further assumptions necessary (but state them clearly so I can see how you get your answer!), develop an approximate equation for the flow rate Q of this pump.

b. What is the optimum value of V_T for a fixed river speed U? What is the maximum value of V_T for which the equation obtained in part **a** is valid?

Hint: The torque exerted on the wheel due to the filled buckets on one side (total filled volume $V_T/2$) is approximately $\rho g V_T R/\pi$.

Hydroplaning Threshold Velocity via Dimensional Analysis

Subject Tags: Dimensional Analysis, Hydroplaning, Tire Pressure, Fluid Forces, Vehicle Safety

Difficulty Level: Moderate

Estimated Time: 25 minutes

In this problem, you will use dimensional reasoning and basic physical insight to estimate the velocity at which a car is likely to hydroplane. The analysis involves comparing the pressure in the tire with the pressure generated by the motion of the water under the tire, and estimating when the supporting force from water overcomes the vehicle's weight.

Skills Tested:

- Dimensional analysis and physical scaling
- Estimating fluid dynamic pressures
- Reasoning with pressure-velocity relationships
- Application of physical intuition to real-world safety questions

Source: CBE 30355 Second Hour Exam, 2014 **File:** problem0148.tex

Hydroplaning Threshold Velocity via Dimensional Analysis

Dimensional Analysis: Some of your classmates demonstrated the phenomenon of hydroplaning. Here we examine where that could occur. You are traveling in a car which has (fairly bald) tires inflated to a pressure of 30 psig. It hits a puddle of water on the road that is deeper than the remaining tread.

Estimate the velocity at which you would expect it to hydroplane (e.g., when would all the tires lose frictional surface contact with the road). Don't just guess, show your reasoning!

Unidirectional Flows

In this section are gathered one of the largest group of problems: flows in only one direction. Flows through capillaries, slits, microchannels, etc., are examined as well as more complicated problems such as lid-driven cavity flow, viscous dampers, etc. Many of the problems are similar, of course: flow down an incline is done several different ways, for example.

Unidirectional Flows – Tube Filling Under Viscous Flow

Subject Tags: fluid mechanics, unidirectional flow, scaling

Difficulty Level: Advanced

Estimated Time: 40 minutes

Summary: A vertical tube open at the bottom and initially sealed at the top is submerged into a viscous fluid. Upon removing the seal, the tube begins to fill due to hydrostatic pressure. This problem investigates the unsteady viscous flow during the filling process, including deriving governing equations and evaluating the conditions under which inertial effects may be neglected.

Skills Tested:

- Application of pressure-driven unidirectional flow assumptions
- Derivation of unsteady filling dynamics in a tube
- Dimensional analysis and nondimensionalization
- Scaling arguments to assess inertial effects

Source: CBE 30355 Midterm Exam, 2021, solved **File:** problem0001.tex

Unidirectional Filling of Submerged Tube

A tube of radius a is submerged vertically to a depth H in a viscous fluid. Initially, the top is sealed with your finger so the tube remains filled with air. At time $t = 0$, you remove your finger and the liquid begins to rise in the tube. Assume the flow inside the tube is unidirectional, neglect capillarity and entrance effects.

a. When you first take your finger off, what is the initial driving force for the flow in the tube?

b. Assuming the fluid is sufficiently viscous such that inertial terms can be neglected, and that at time t the fluid height is $h(t)$, develop an expression for the area-averaged velocity in the tube.

c. Use your result from part (b) to derive a differential equation governing $h(t)$. Render the equation dimensionless to determine a characteristic filling time. (Note: the tube asymptotically approaches full height; solving the equation shows it reaches 84% fill at a nondimensional time of one.)

d. In part (b), inertial effects were ignored. Use a scaling analysis to estimate the minimum viscosity μ for which this approximation is valid.

The following cylindrical Navier–Stokes relations may be helpful:

$$\frac{1}{r}\frac{\partial}{\partial r}(ru_r) + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial u_z}{\partial t} + u_r\frac{\partial u_z}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_z}{\partial \theta} + u_z\frac{\partial u_z}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 u_z}{\partial \theta^2} + \frac{\partial^2 u_z}{\partial z^2}\right] + \rho g_z$$

Poiseuille Flow in a Syringe and Needle System

Subject Tags: laminar flow, Poiseuille flow, control volume, pressure-driven flow, syringe dynamics

Difficulty: Moderate

Estimated Time: 40 minutes

Summary: In this problem, students analyze the flow through a syringe and its attached needle. By applying mass conservation and assuming laminar, fully developed flow through the needle, students calculate the velocity ratios and the plunger force required to empty the syringe in a fixed time. They also evaluate the realism of flow assumptions and perform numerical estimation based on given dimensions and physical properties.

Skills Tested:

- Application of mass conservation
- Use of Poiseuille's law to relate pressure drop and flow rate
- Dimensional substitution and numerical evaluation
- Critical evaluation of flow assumptions

Source: CBE 30355 First Hour Exam, 2017, solved **File:** problem0065.tex

https://diffusivepress.org/courseware/cbe30355/

Poiseuille Flow in a Syringe and Needle System

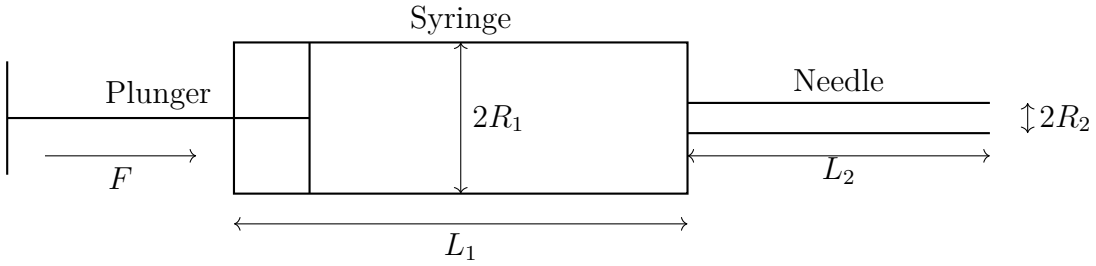
A syringe with radius R_1 and length L_1 is being emptied through a needle with radius $R_2 \ll R_1$ and length L_2 in time T, as depicted below.

a. Using a mass balance, determine the ratio of the average velocity in the needle to that in the syringe.

b. Assuming all losses to be in the needle, and assuming fully developed laminar flow, derive an expression for the force on the plunger necessary to empty the syringe in time T.

c. If $R_1 = 0.5\,\text{cm}$, $R_2 = 0.05\,\text{cm}$, $L_1 = 4\,\text{cm}$, $L_2 = 2\,\text{cm}$, $T = 4\,\text{s}$, and the working fluid has the same viscosity as water, determine the numerical value of this force.

d. Quantitatively comment on the validity of the assumption of fully developed laminar flow in the needle.



The equations of motion in cylindrical coordinates are given by:

$$\rho\left(\frac{\partial v_r}{\partial t}+v_r\frac{\partial v_r}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta}-\frac{v_\theta^2}{r}+v_z\frac{\partial v_r}{\partial z}\right)=-\frac{\partial p}{\partial r}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial(rv_r)}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_r}{\partial \theta^2}-\frac{2}{r^2}\frac{\partial v_\theta}{\partial \theta}+\frac{\partial^2 v_r}{\partial z^2}\right]+\rho g_r$$

$$\rho\left(\frac{\partial v_\theta}{\partial t}+v_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}+\frac{v_r v_\theta}{r}+v_z\frac{\partial v_\theta}{\partial z}\right)=-\frac{1}{r}\frac{\partial p}{\partial \theta}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial\,(rv_\theta)}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2}+\frac{2}{r^2}\frac{\partial v_r}{\partial \theta}+\frac{\partial^2 v_\theta}{\partial z^2}\right]+\rho g_\theta$$

$$\rho\left(\frac{\partial v_z}{\partial t}+v_r\frac{\partial v_z}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta}+v_z\frac{\partial v_z}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2}+\frac{\partial^2 v_z}{\partial z^2}\right]+\rho g_z$$

$$\frac{\partial \rho}{\partial t}+\frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r)+\frac{1}{r}\frac{\partial}{\partial \theta}(\rho v_\theta)+\frac{\partial}{\partial z}(\rho v_z)=0$$

Flow Resistance in Membranes: Unidirectional Flows

Subject Tags: membrane flow; pressure drop; Poiseuille flow; membrane porosity; flow resistance

Difficulty Level: Moderate

Estimated Time: 30–40 minutes total (a: 10 min, b: 10 min, c: 5 min, d: 10–15 min)

Summary: This problem models a microfiltration membrane as an array of cylindrical pores and explores the pressure drop associated with flow through the pores. It involves deriving an expression using Hagen–Poiseuille flow, performing numerical calculations with unit conversions, checking laminar flow conditions, and evaluating design parameters for asymmetric membranes.

Skills Tested:

- Fluid mechanics in porous media
- Hagen–Poiseuille flow application
- Dimensional consistency and unit conversion
- Analysis of laminar flow regime
- Membrane design considerations

Source: CBE 30355 Midterm Exam, sp2021, solved **File:** problem0091.tex

Flow Resistance in Membranes: Unidirectional Flows

A microfiltration membrane of thickness h is often modeled as an array of cylindrical pores of radius a. The membrane is further characterized by a porosity (fraction open area) of ϵ. We are interested in the pressure drop Δp required for a flow rate Q/A (flow rate per unit area, encompassing both the pores and solid).

a. For this model of membrane flow, determine the relationship between Δp and Q/A.

b. For a pore radius of $2\,\mu$m, membrane thickness of $1\,$mm, porosity $\epsilon = 0.5$, and the properties of water, determine the pressure drop necessary to achieve a flow rate of $60\,\text{ml}/(\text{min}\cdot\text{cm}^2)$. Express your result in atmospheres.

Hint: $1\,\text{dyne}/\text{cm}^2 = 0.99\times10^{-6}\,\text{atm}$, and $1\,\mu\text{m} = 10^{-4}\,\text{cm}$.

c. Your calculation in part (b) assumed unidirectional, laminar flow. Quantitatively comment on whether this is appropriate for cylindrical pores of this size at this flow rate.

d. For many applications a pore radius of $2\,\mu$m is too large (e.g., bacteria could pass through). We often use asymmetric membranes with two layers: a support layer (with parameters like those in part b) and a thinner surface layer with much smaller holes. If the surface layer has pore radii of $0.25\,\mu$m, how thin must it be for the pressure drop to no more than double (i.e., for the thin layer to have the same pressure drop as the thicker layer with the larger $2\,\mu$m pores), assuming the same flow rate? Give your answer in microns.

The following equations may be helpful:

$$\frac{1}{r}\frac{\partial\left(ru_r\right)}{\partial r}+\frac{1}{r}\frac{\partial u_\theta}{\partial\theta}+\frac{\partial u_z}{\partial z}=0$$

$$\rho\left(\frac{\partial u_z}{\partial t}+u_r\frac{\partial u_z}{\partial r}+\frac{u_\theta}{r}\frac{\partial u_z}{\partial\theta}+u_z\frac{\partial u_z}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 u_z}{\partial\theta^2}+\frac{\partial^2 u_z}{\partial z^2}\right]+\rho g_z$$

Unidirectional Flows in an Inclined Semi-Circular Trough

Subject Tags: unidirectional flow; Newtonian fluid; shear stress; rheology; inclined trough

Difficulty Level: Moderate

Estimated Time: 30–40 minutes total (a: 15 min, b: 15–20 min, c: 5 min)

Summary: This problem examines the flow of a Newtonian fluid in an inclined semi-circular trough used in rheological experiments. Students are asked to determine the required flow rate to brim the trough, evaluate the resulting shear stress distribution, and predict qualitative effects when the fluid viscosity is altered or a suspension is used.

Skills Tested:

- Momentum balance in cylindrical coordinates
- Unidirectional flow assumptions and simplifications
- Shear stress profile determination
- Reasoning about non-Newtonian behavior in suspension flows

Source: CBE 30355 Midterm Exam, 2022, solved **File:** problem0095.tex

Unidirectional Flows in an Inclined Semi-Circular Trough

As was discussed in class, Tanner's group from the University of Sydney used an inclined half-filled semi-circular trough to study the rheology of suspensions. In this problem we look at the operating conditions of this system.

a. If the trough is of radius R and the angle of inclination to the horizontal is θ, what is the flow rate Q required to fill the trough to the brim (i.e., to the edge of the semicircle)? Assume unidirectional Newtonian flow for simplicity.

b. The shear stress is given by

$$\tau_{zr} = \tau_{rz} = \mu \left(\frac{\partial u_z}{\partial r} + \frac{\partial u_r}{\partial z} \right)$$

What is the stress distribution in the trough? If the viscosity of the fluid is doubled (still just filled to the brim!), what happens to the shear stress? (Be careful!)

c. (2 points extra credit). When a suspension (rather than a Newtonian fluid) flowed down the trough, something weird happened. What was it and why did it occur?

The following equations may be helpful:

$$\frac{1}{r}\frac{\partial \left(r u_r\right)}{\partial r} + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0$$

$$\rho \left(\frac{\partial u_z}{\partial t} + u_r \frac{\partial u_z}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_z}{\partial \theta} + u_z \frac{\partial u_z}{\partial z} \right) = -\frac{\partial p}{\partial z} + \mu \left[\frac{1}{r}\frac{\partial}{\partial r}\left(r \frac{\partial u_z}{\partial r} \right) + \frac{1}{r^2}\frac{\partial^2 u_z}{\partial \theta^2} + \frac{\partial^2 u_z}{\partial z^2} \right] + \rho g_z$$

Coil Pump Performance and Flow Analysis

Tags: Rotating Flow, Laminar Flow, Energy Balance, Reynolds Number

Difficulty: Advanced

Estimated Time: 45 minutes

In this problem we explore the Coil Pump, a variant still used in irrigation systems around the world. A coiled tube rotates on a horizontal drum, half-filled with fluid. Gravity and friction interact to set the equilibrium tilt of the liquid slug in each coil. The goal is to analyze the performance and limits of this device under steady conditions and laminar flow assumptions.

Skills Tested:

- Cylindrical Coordinates
- Reference Frame Shift
- Pressure Gradient Effects
- Frictional Loss Estimation

Source: CBE 30355 Second Hour Exam, 2009 **File:** problem0135.tex

Coil Pump Performance and Flow Analysis

Unidirectional Flows: An interesting student demonstration is the Archimedes Screw, an ancient pumping technique. In this problem we explore the Coil Pump, a variant which is still used in irrigation systems in parts of the world. Consider the geometry depicted below:

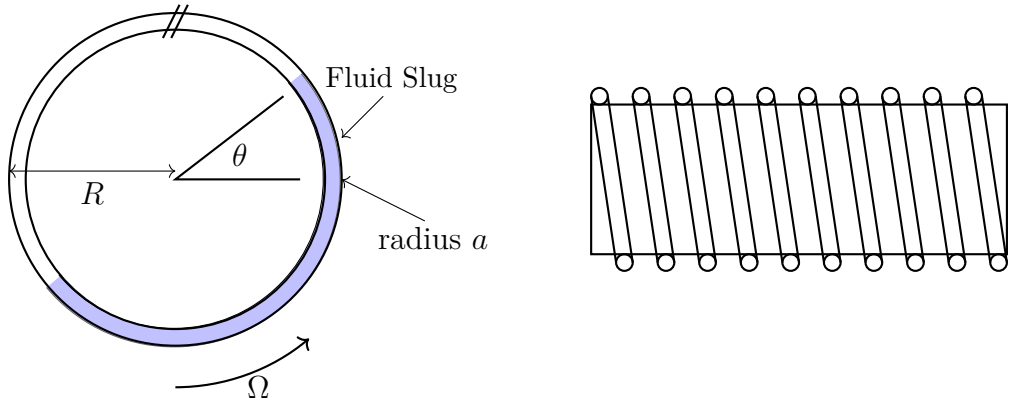


Figure 9: Schematic of a coil pump: tubing coiled around a rotating horizontal drum. Each loop is half-filled with fluid.

A long tube of radius a is coiled around a horizontal drum of radius R (we take $a/R \ll 1$). As the drum rotates with angular velocity Ω, the end loop of the coil is half-filled with fluid and air from a scoop, and this slug propagates down the coil. This leaves each loop of the coil half-filled with fluid. Gravity drags the fluid down towards the bottom of each loop of the coil, and causes the fluid to spiral its way out to the exit. Because of frictional losses, however, the fluid slug in each loop is dragged up a bit, so that instead of being level it is at some angle θ from horizontal. Here we analyze the steady-state performance in the absence of any back pressure.

a. What is the nominal liquid flow rate of the pump as a function of the parameters of the problem? This is easy!

b. If the flow is laminar, we can approximate the frictional resistance as that due to unidirectional flow through a tube. Using this, determine the angle θ the slug in each loop acquires relative to horizontal as a function of the angular velocity. Work in a reference frame where the slug is stationary (as depicted above). The z-momentum equation may be helpful:

$$\rho\left(\frac{\partial v_z}{\partial t}+v_r\frac{\partial v_z}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta}+v_z\frac{\partial v_z}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2}+\frac{\partial^2 v_z}{\partial z^2}\right]+\rho g z$$

c. The maximum possible flow rate occurs when θ reaches $\pi/2$. What is the maximum angular velocity and flow rate?

d. The key assumption leading to the answers in part **b** and **c** is laminar flow, which is valid only for Re < 2100 (based on tube diameter). If $R = 10\,\text{cm}$, $a = 1\,\text{cm}$, and the working fluid is water, calculate whether this assumption is correct. If the working fluid is corn oil ($\nu \sim 80$ centistokes) what will this answer be? Are we underestimating or overestimating the maximum flow rate?

Unidirectional Flows

Subject Tags: Capillary Flow, Lubrication Theory, Unsteady Flow, Surface Tension

Difficulty Level: Moderate

Estimated Time: 45 minutes

Summary: A fingerstick test draws blood into a narrow capillary by surface tension. This problem requires derivation and solution of the governing equation for the unsteady filling of the capillary tube, leading to an analytical expression for the filling time and a final numerical estimate.

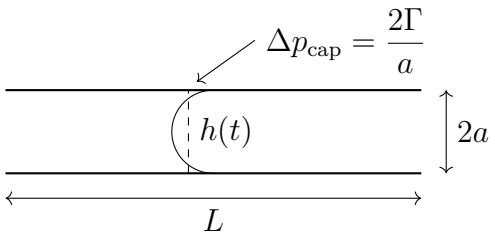
Skills Tested:

- Governing equation formulation with moving boundary
- Lubrication approximation for unidirectional viscous flow
- Dimensional analysis and similarity
- Solving ODEs for time-dependent interface motion

Source: CBE 30355 Final Exam, 2018 **File:** problem0161.tex

Capillary Filling of a Tube by Surface Tension

In the annual University sponsored health screenings for faculty and staff, technicians run a blood test to measure cholesterol and glucose levels. What they do is poke a hole in your finger to get a drop of blood and then draw the blood into a capillary which is then inserted into the analysis machine. The blood is drawn into the capillary by surface tension as depicted below. If the surface tension of blood is $\Gamma = 56$ dynes/cm, the viscosity of blood is 4 cP, the radius of the capillary is $a = 50$ μm, and the length $L = 5$ cm, how long does it take for the tube to fill?



(Hint: Inertial effects are negligible for this problem, and the pressure differential is applied only over the filled length h of the capillary, which changes in time at a rate equal to the average axial velocity!)

a. Write down the governing equations and boundary conditions, as well as the equation governing the time dependent filled length h.

b. Render the equations dimensionless to determine how the filling time depends on the parameters of the problem.

c. Solve the problem to determine when the capillary is completely filled in terms of the parameters of the problem.

d. Plug in the numbers to get the final numerical value.

$$\rho\left(\frac{\partial v_z}{\partial t} + v_r\frac{\partial v_z}{\partial r} + \frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta} + v_z\frac{\partial v_z}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2} + \frac{\partial^2 v_z}{\partial z^2}\right] + \rho g_z$$

$$\frac{\partial \rho}{\partial t} + \frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r) + \frac{1}{r}\frac{\partial}{\partial \theta}(\rho v_\theta) + \frac{\partial}{\partial z}(\rho v_z) = 0$$

Flow Rate in a Channel from Pressure Gradient

Subject Tags: Laminar Flow; Pressure Gradient; Channel Flow; Unidirectional Flow; Navier-Stokes

Difficulty: Simple

Estimated Time: 15 minutes

Summary: This problem considers unidirectional, steady, laminar flow in a planar channel of width $2b$. The objective is to compute the flow rate per unit depth out of the plane, Q/W, as a function of the imposed pressure gradient in the x-direction using appropriate simplifications to the Navier-Stokes equations.

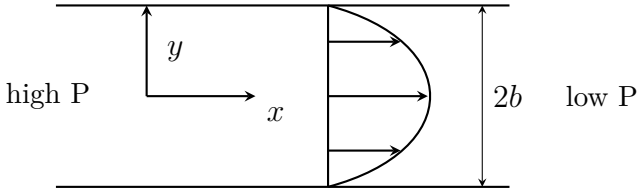
Skills Tested:

- Application of Navier-Stokes equations to unidirectional flow
- Solving second-order ODEs for velocity profiles
- Computing volumetric flow rate from velocity profiles

Source: CBE 30355 Final Exam, 2001 **File:** problem0013.tex

Flow in a Channel from Pressure Gradient

Consider unidirectional, laminar, steady flow in a channel of width $2b$ as depicted below.



What is the flow rate per unit extension out of the plane of the paper, Q/W, as a function of the pressure gradient in the x-direction?

Capillary Rise in a Narrow Channel

Subject Tags: Capillarity, Surface Tension, Hydrostatics, Lubrication Theory

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: This problem investigates the rise of fluid in a narrow 2-D channel due to surface tension-induced capillary pressure. The static and dynamic behaviors of the fluid column are explored by applying hydrostatics and lubrication theory, respectively.

Skills Tested:

- Application of capillary pressure concepts
- Hydrostatic pressure balance
- Simplified Navier–Stokes analysis at low Reynolds number
- Formulation of governing differential equations

Source: CBE 30355 First Exam, 2003 **File:** problem0105.tex

Capillary Rise in a Narrow Channel

Earlier this semester we briefly looked at capillary pressure effects, whose magnitudes are characterized by the surface tension σ. In this problem we examine the capillary rise of a column of fluid in a narrow 2-D channel. Consider the system depicted below. Fluid enters the thin gap of width $2b$. If the fluid wets the surfaces of the channel (basically, if a drop of the fluid would spread out over the material composing the channel surfaces), then the curvature of the interface will cause a capillary pressure drop between the fluid and the air of

$$\Delta p_{\text{cap}} = \frac{\sigma}{b}.$$

Because the pressure in the fluid behind the meniscus is now less than the atmospheric pressure, it sucks fluid up the gap from the reservoir. It is this phenomenon we wish to investigate.

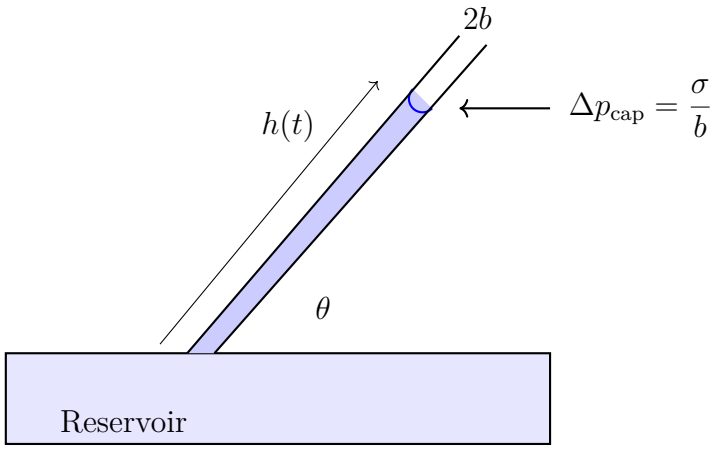


Figure 10: Capillary rise in a narrow 2-D channel

a. Using hydrostatics, determine the steady-state capillary rise in the channel (e.g., the length of the column of liquid when all motion ceases).

b. Now for the transient problem. For a thin gap we can ignore the end effects at the entrance and at the meniscus (e.g., we take the flow to be unidirectional). We can also ignore all inertial terms in the equation of motion, as the Reynolds number is very small. With these approximations, derive a simple differential equation which governs the length of the column of fluid with respect to time.
(Hint: think of plane-Poiseuille flow, flow rates, and mass balances)

Drainage of a Viscous Fluid Through a Narrow Slit

Subject Tags: Navier-Stokes, Unidirectional Flow, Viscous Flow, Slit Drainage

Difficulty Level: Moderate

Estimated Time: 45 minutes

Consider a trough of width W draining through a slit of width $2b$ and length L (in the flow direction), as depicted below. The trough initially contains a viscous fluid of density ρ and viscosity μ filled to a height H_0. The problem is two-dimensional with negligible variation into the page. Because the slit is much narrower than the trough ($2b/W \ll 1$) and narrow compared to its length ($2b/L \ll 1$), viscous resistance in the slit dominates, and the flow can be considered unidirectional.

Skills Tested:

- Formulating driving forces for flow
- Solving Navier-Stokes equations under lubrication approximation
- Developing and interpreting velocity profiles
- Deriving differential equations describing drainage rates

Source: CBE 30355 First Hour Exam, 2005 **File:** problem0120.tex

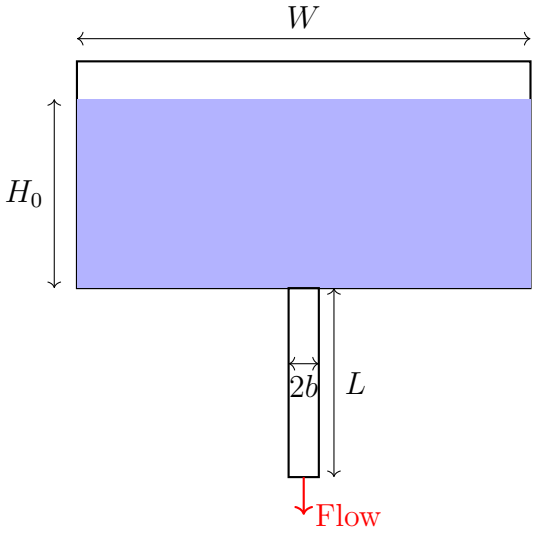
Drainage Through a Narrow Slit

Consider the trough of width W which drains through a slit of width $2b$ and length (in the flow direction) L as depicted below. The trough is initially filled with liquid of density ρ and viscosity μ to a height H_0. You may neglect any variation into the page (e.g., the problem is two-dimensional). The fluid is assumed to be very viscous (e.g., Karo syrup), and thus inertial effects are negligible. Because the slit is much narrower than the trough ($2b/W \ll 1$), the slit dominates viscous resistance to flow (remember the milkshake and straws?). Finally, because the slit is narrow relative to its length ($2b/L \ll 1$), the flow may be considered unidirectional.

a. What is the magnitude of the initial driving force for the drainage?

b. Solve the Navier-Stokes equations for the velocity profile in the slit and the initial drainage rate for fluid draining out of the trough. Assume unidirectional flow!

c. Develop a differential equation describing the height $H(t)$ of the fluid in the trough as a function of time.



Capillary Filling of a Microchannel

Subject Tags: Capillarity, Microfluidics, Plane-Poiseuille Flow, Viscous Flow

Difficulty Level: Moderate

Estimated Time: 45 minutes

Summary: In this problem, we analyze capillary filling of a rectangular microchannel via the balance of capillary and viscous forces. The flow is assumed to be unidirectional and governed by plane-Poiseuille flow. Using this, the governing equations and fill time are derived, and scaling arguments are used to determine parameter dependencies.

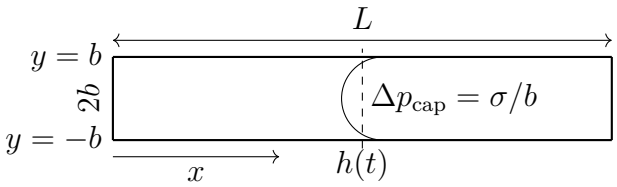
- Derivation of pressure-driven flow in a narrow channel
- Use of mass conservation to link velocity and fill length
- Dimensional analysis of governing equations
- Integration of ODEs for fill time

Source: CBE 30355 Final Exam, 2006 **File:** problem0127.tex

Capillary Filling of a Microchannel

Analysis of blood chemistry is now being done by putting a tiny drop of blood onto the entrance of a microchannel, and the blood is drawn into the channel via capillary action. This is the same effect that allows fuel to be drawn up the wick of a lamp, for example. In this problem we examine the length of time necessary for the channel to fill.

Consider the channel of depth 2 b and length L given below. The curvature of the fluid interface causes the pressure behind the meniscus to be less than atmospheric pressure by an amount $\Delta p_{cap} = \sigma/b$. Because the channel dimensions are very small, inertial effects are negligible and the force due to the capillary pressure is balanced by viscous effects only.



a. Using this information, determine the equations governing the velocity distribution in the channel, and the length h of the column of fluid in the channel as a function of time. Make any approximations or simplifications necessary to get a reasonable problem. (Hint: think of plane-Poiseuille flow, flow rates, and mass balances)

b. Using scaling analysis, render the problem dimensionless and determine the dependence of the fill time on all the parameters of the problem.

c. Solve for the actual time to fill the channel from $h(0) = 0$ to $h(t) = L$.

(As a side note, the analysis you do above works very well provided the channel depth is greater than around 50μm. For microchannels thinner than this value, some fascinating migration effects of the red blood cells become significant, and you have to do a much more sophisticated analysis of the problem!)

Lid-Driven Cavity Flow and Pressure Gradient

Subject Tags: Navier-Stokes; shear flow; pressure gradient; lid-driven cavity

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: This problem explores lid-driven cavity flow where the top plate moves to induce shear flow, while stationary side walls (wipers) induce a pressure-driven counter flow. Students are asked to analyze the resulting velocity profile using the Navier-Stokes equation, boundary conditions, and the constraint of zero net flow to determine the induced pressure gradient.

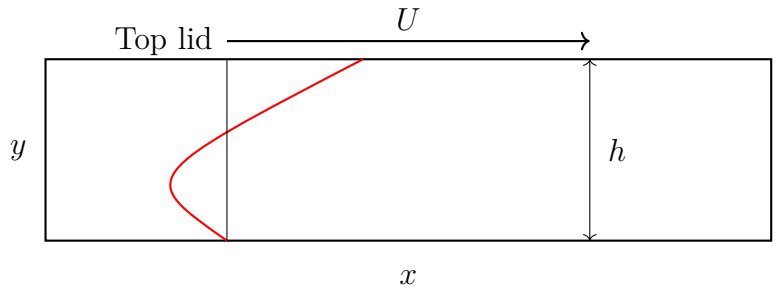
Skills Tested:

- Formulation of Navier-Stokes equations for simplified geometries
- Identification of dominant terms and boundary conditions
- Analytical integration of differential equations
- Application of flow constraints

Source: CBE 30355 First Hour Exam, 2002 **File:** problem0017.tex

Lid-Driven Cavity Flow and Pressure Gradient

Consider the flow system depicted below. This type of flow is called lid-driven cavity flow, in which the upper plate drives a shear flow from left to right, and the wipers (the ends of the box or cavity) induce a pressure gradient in the x-direction which causes a pressure-driven back flow. If we focus our attention on the middle of the box (far from either wiper), and if the box is thin, then the flow is approximately unidirectional in the x-direction (e.g., no component in the y or z directions).



a. Write down the Navier-Stokes equation for momentum in the direction of flow for the central region and show which terms are non-zero. Write down the appropriate boundary conditions.

b. Integrate the differential equation and apply the boundary conditions to obtain the velocity profile in terms of the unknown pressure gradient.

c. The presence of the wipers forces the net flow (e.g., the integrated flow rate) in the x-direction to be zero. Use this condition to determine the pressure gradient induced by the wipers.

Lubricated Thrust Bearing Analysis

Subject Tags: Lubrication theory; Thrust bearing; Scaling analysis; Navier-Stokes limit

Difficulty Level: Advanced

Estimated Time: 40 minutes (15 min setup, 10 min scaling, 15 min solution)

Summary: In this problem, you examine the behavior of a 2-D lubricated thrust bearing with a wiper seal at one end. The fluid is dragged into the narrow gap by the motion of the bottom plate, creating a pressure gradient and a net force. You are asked to derive the governing equations, perform a lubrication limit scaling analysis, and calculate the force per unit width in the bearing.

Skills Tested:

- Application of lubrication theory ($h/L \ll 1$)
- Use of integral mass conservation in thin-film flows
- Identification of dominant balance in asymptotic limits
- Dimensional analysis for scaling arguments
- Analytical solution of pressure-driven viscous flow

Source: CBE 30355 Second Hour Exam, 2017, solved **File:** problem0068.tex

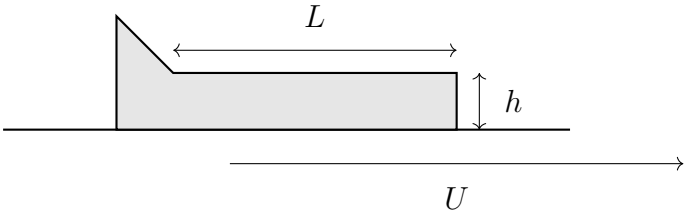
Lubricated Thrust Bearing

A lubricated thrust bearing is depicted below. The lower surface moves from left to right with velocity U, dragging fluid into the gap of constant height h as shown. A wiper at $x = L$ prevents fluid from exiting the bearing, producing a pressure gradient and recirculation within the domain. We seek the thrust per unit extension into the third dimension (i.e., F/W) resulting from the bearing operation.

a. Write down the governing equations and boundary conditions in the limit $h/L \ll 1$. *Hint: one of the equations will be an integral relation.*

b. Perform a scaling analysis of the equations from part (a) to determine how the thrust per unit width F/W scales with the system parameters in the lubrication limit.

c. Solve for the force F/W produced by the bearing.



Unidirectional Flow in a Viscous Damper

Subject Tags: viscous flow; lubrication approximation; symmetry; no-flux condition; damping

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: A viscous damper uses fluid shear and pressure gradients to resist motion. This problem analyzes steady unidirectional flow in a narrow gap where the central plate moves with velocity U but no net fluid exits. Students derive the velocity profile, apply integral constraints, and compute the total resisting force.

Skills Tested:

- Formulating and solving simplified Navier-Stokes equations
- Applying boundary and symmetry conditions
- Enforcing integral constraints like no net flow
- Calculating forces from stress distributions

Source: CBE 30355 First Hour Exam, 2019, solved **File:** problem0082.tex

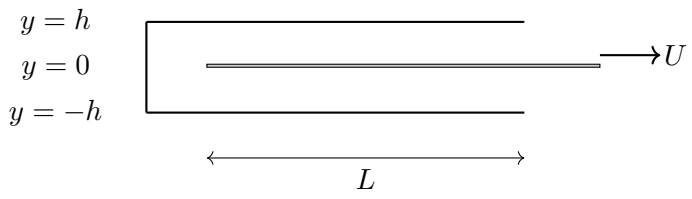
Unidirectional Flow in a Viscous Damper

A viscous damper uses viscous dissipation (e.g., the stress on a plate) to damp vibrations. The geometry is depicted below. The plate at the center has a velocity U, and because the end is closed, there is no net flow of the fluid — which produces a pressure gradient. Assuming steady unidirectional flow, we want to calculate the force on the plate as a function of the viscosity of the fluid, the plate velocity U, the length in the slot L, the fluid gap width h, and the width (into the paper) W.

a. Write down the equations governing the velocity distribution with these assumptions as well as the boundary conditions (and the integral no-flux condition!). The sides are mirror images, so you can just work with one side of the plate.

b. Solve for the velocity distribution.

c. Get the force on the plate. Note that the force is just double what you get for one side.



Unidirectional Flows

Subject Tags: lubrication approximation; unidirectional flow; pressure gradient; tangential force; normal force

Difficulty Level: Advanced

Estimated Time: 30 minutes

Summary: This problem concerns unidirectional flow in a narrow slot with a moving central plate. The configuration changes from open to closed ends, leading to dramatically different velocity fields and pressure distributions. Students must compute the velocity profiles, wall shear, and pressure forces in both configurations, applying the lubrication approximation.

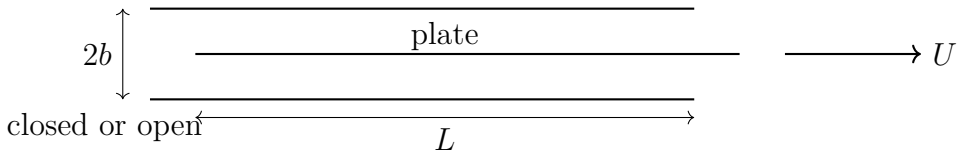
Skills Tested:

- Stokes flow velocity field derivation
- Tangential force via wall shear stress
- Pressure field determination from continuity
- Calculation of normal forces from pressure

Source: CBE 30355 Second Hour Exam, 2019, solved **File:** problem0086.tex

Unidirectional Flows

A plate occupies the center of a slot of width $2b$ and length L (with extension into the paper W) as depicted below. Each side is independent, so you can calculate things just looking at one side of the plate. The plate is pulled out with velocity U. The ratio $b/L \ll 1$ so you can assume unidirectional flow.



a. If the slot is open on both ends, no pressure gradient can be produced. For this case, calculate the velocity profile in the slot and tangential force on the plate.

b. If the slot is closed at one end, the net flow goes to zero and a pressure gradient is produced, significantly increasing the force on the plate (a much more efficient damper!). Calculate the velocity distribution and force on the plate for this case.

c. For the closed end case there is also a normal force on the walls of the slot (it tends to collapse towards the plate if not supported). Calculate this force. Just use the top wall of the slot; the force on the bottom would be the negative of this.

Creeping Flow in a Lid-Driven Cavity

Subject Tags: Creeping Flow, Lubrication Theory, Pressure Gradient, Stokes Equation

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: This problem examines the behavior of a low Reynolds number (creeping) flow within a rectangular cavity where the top boundary moves while the bottom is stationary. A pressure-driven counterflow develops due to confinement. The thin-gap limit is first analyzed to find the velocity and pressure distribution, and the governing PDE and boundary conditions for a full 2D solution are later identified.

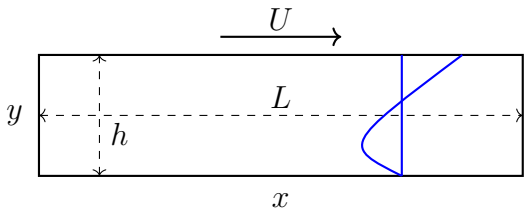
Skills Tested:

- Lubrication approximation in thin geometries
- Stokes flow velocity profiles
- Identification of appropriate governing equations
- Application of correct boundary conditions

Source: CBE 30355 Second Hour Exam, 2003 **File:** problem0108.tex

Creeping Flow in a Lid-Driven Cavity

In this problem we examine the lid-driven cavity flow depicted below. The top plate is dragged to the right (positive x-direction) with velocity U, while the bottom is fixed and the side walls produce a pressure gradient and resulting backflow to the left (negative x-direction). The flow may be taken to be at zero Reynolds number. Your job is to figure out what is going on.



a. In the limit $h/L \ll 1$ (thin gaps), determine the velocity profile and pressure gradient far from the ends. This means that you ignore the end regions where the flow field gets nasty.

b. What (hint: single!) equation governs the flow when the aspect ratio is $O(1)$? (e.g., h/L is no longer small). What are the appropriate boundary conditions for this equation?

Viscous Dampers and Cavitation

Subject Tags: Creeping Flow, Lubrication Theory, Cavitation, Pressure Drop, Shear Stress

Difficulty Level: Moderate

Estimated Time: 30 minutes

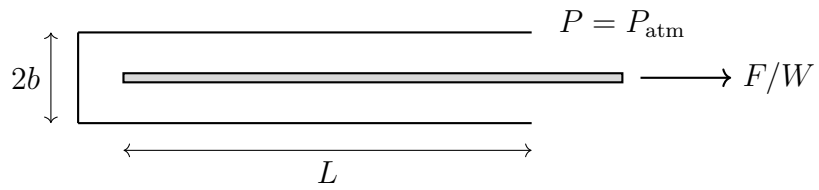
Summary: This problem concerns a viscous damper where a plate is pulled through a fluid-filled cavity at low Reynolds number. Students must apply lubrication theory to estimate the pressure distribution and identify the condition at which cavitation occurs, thereby determining the maximum force that can be applied to the plate.

Skills Tested:

- Application of lubrication theory at low Reynolds number
- Estimating shear stress and pressure gradients in viscous flows
- Understanding cavitation limits based on vapor pressure
- Interpreting force per unit width on moving plates

Source: CBE 30355 First Hour Exam, 2013 **File:** problem0145.tex

Maximum Force Before Cavitation in a Viscous Damper



A plate (of extension W into the paper and negligible thickness) is used as a viscous vibration damper as depicted above. The cavity (of width b on either side of the plate) is filled with a viscous fluid of viscosity μ. In this problem we ignore transients and fluid inertia (e.g., we are at low Reynolds number), and we take all flows to be unidirectional.

The force per unit width F/W pulling the plate outwards is resisted by the resulting fluid flow exerting a shear stress on the plate. If you pull on it too hard, however, the pressure in the fluid at the end of the cavity will get so low that the fluid will cavitate. We assume cavitation occurs when the pressure equals the vapor pressure of the fluid.

For simplicity, we take this vapor pressure to be zero. What is the **maximum** force per unit width F/W that can be applied to the plate **before cavitation occurs**?

Plane Poiseuille Flow in a Microfluidic Channel

Subject Tags: Viscous Flow; Shear Stress; Bioengineering Application; Poiseuille Flow; Flow Rate Estimation

Difficulty: Moderate

Estimated Time: 20 minutes

Summary: This problem explores a classical Poiseuille flow in a narrow rectangular channel designed for bioengineering experiments. The objective is to determine the necessary volumetric flow rate required to maintain a specified shear stress at the wall, using known geometry and fluid properties. The scenario is relevant to microfluidic device design for studies involving cell adhesion.

Skills Tested:

- Application of Poiseuille flow solution
- Use of wall shear stress relationships
- Unit conversions and physical reasoning

Source: CBE 30355 Final Exam, 2002 **File:** problem0024.tex

Plane Poiseuille Flow in a Microfluidic Channel

A problem which is currently being investigated in bioengineering laboratories is the phenomenon of cell adhesion to surfaces in the presence of hydrodynamic stresses. This is very important in the design of biocompatible materials, for example.

To study this, a researcher has built a rectangular flow cell which is $100\,\mu$m deep, 2 mm wide, and 2 cm long. The objective is to have a wall shear stress (e.g., the stress at the lower wall—the 2 mm $\times$ 2 cm surface where cell adhesion is being studied) of

$$\tau_w = 100\,\text{dyne/cm}^2.$$

If the working fluid has the same viscosity as water, what should the volumetric flow rate of the pump supplying the fluid be?

Make any simplifications you deem appropriate, but provide a brief justification for your assumptions.

Plane-Poiseuille Flow in a Microchannel

Subject Tags: fluid mechanics; microfluidics; pressure drop; laminar flow; rectangular channel

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: A fluid is to be pumped through a long narrow rectangular microchannel with given geometric dimensions and flow rate. The goal is to derive an expression for the required pressure drop and calculate its numerical value in three unit systems, using properties of water and appropriate approximations.

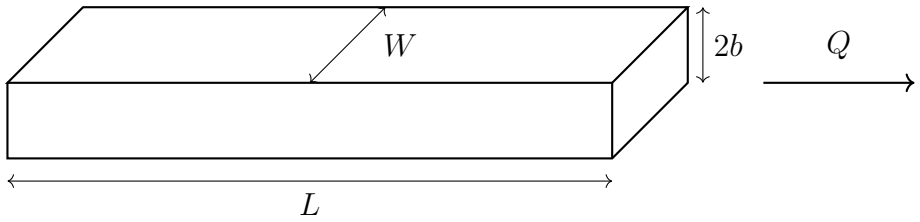
Skills Tested:

- Derivation of pressure drop in plane-Poiseuille flow
- Use of geometric assumptions and approximations
- Conversion of pressure between cgs, atmospheres, and cm H_2O

Source: CBE 30355 First Hour Exam, 2011, solved **File:** problem0029.tex

Plane-Poiseuille Flow in a Microchannel

It is desired to pump fluid of viscosity μ through a microchannel of length L, width W, and depth $2b$ with volumetric flow rate Q, as depicted below. The aspect ratios are such that $L \gg W \gg b$.



a. Making any reasonable approximations, develop an equation for the pressure drop required for the flow rate Q.

b. If $L = 10\,\text{cm}$, $W = 0.1\,\text{cm}$, $b = 0.001\,\text{cm}$ (i.e., $2b = 20\,\mu\text{m}$), and $Q = 0.1\,\mu\text{l/s}$ (i.e., the channel processes 1 ml in a bit under 3 hours), and the working fluid is water, what is the numerical value of the pressure drop in:

- cgs units (dyne/cm^2)
- atmospheres
- cm H_2O (e.g., if you used a water manometer to measure it)

Pressure-Driven Flow in a Microchannel

Subject Tags: Unidirectional Flow, Microfluidics, Pressure Drop, Viscous Flow

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: This problem considers a pressure-driven flow through a rectangular microfluidic channel with water as the working fluid. The driving pressure is provided by a water manometer, and students are asked to determine the resulting volumetric flow rate. A hint emphasizes the dominant role of viscous diffusion in the narrowest channel dimension.

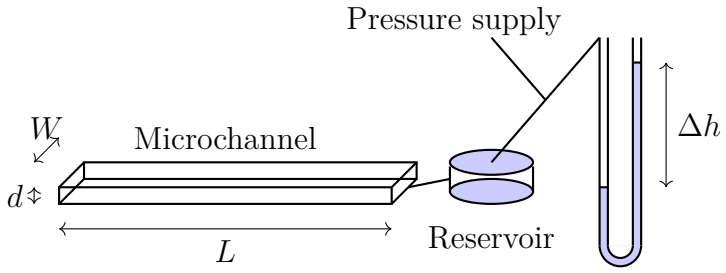
Skills Tested:

- Use of unidirectional flow assumptions
- Viscous pressure-driven flow in rectangular channels
- Pressure head to pressure conversion
- Dimensional reasoning and unit conversion

Source: CBE 30355 First Hour Exam, 2015, solved **File:** problem0052.tex

Pressure-Driven Flow in a Microchannel

In our laboratory we have studied pressure-driven flow in microfluidic channels, and in some cases monitored the pressure using an open-ended water manometer such as discussed in class. Consider the simplified system depicted below:



The channel dimensions are:

- Depth: 50 µm
- Width: 1 mm
- Length: 5 cm

The working fluid for both the manometer and the channel is water. If the manometer reading was 10 cm H_2O (about as high as we could go before the manometer blew out), what would be the flow rate?

Give the answer in microliters/minute.

Hint: For unidirectional flow in any conduit, the flow rate is controlled by viscous diffusion of momentum in the thinnest direction.

Plane Poiseuille Flow: Cell Adhesion and Wall Shear Stress

Subject Tags: pressure-driven flow; plane Poiseuille flow; shear stress; bioengineering; dimensional analysis

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: This problem explores pressure-driven flow in a shallow rectangular geometry typical of biomedical flow cells. Students determine the required pressure to achieve a desired wall shear stress and calculate the flow rate for a water-like fluid. The result is notable for its independence from viscosity, emphasizing the utility of force balances in microfluidic contexts.

Skills Tested:

- Applying force balance in viscous flows
- Interpreting wall shear stress in rectangular geometries
- Understanding Poiseuille flow assumptions
- Converting between units (e.g., dyne/cm^2, µL/min)

Source: CBE 30355 First Hour Exam, 2018, solved **File:** problem0073.tex

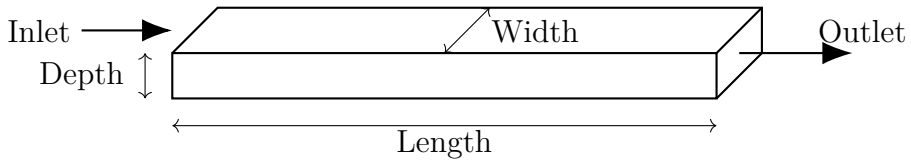
https://diffusivepress.org/courseware/cbe30355/

Plane Poiseuille Flow: Cell Adhesion and Wall Shear Stress

A problem which is currently being investigated in bioengineering laboratories is the phenomenon of cell adhesion to surfaces in the presence of hydrodynamic stresses. This is very important in the design of biocompatible materials, for example. To study this, a researcher has built a rectangular flow cell which is 50 µm deep, 1 mm wide, and 2 cm long. The objective is to have a wall shear stress (e.g., stress at the lower wall — the 1 mm × 2 cm surface — where cell adhesion is being studied) of 5 dyne/cm^2 (cgs units for stress). Due to the ratio of length scales, you can assume unidirectional plane-Poiseuille flow.

a. What is the pressure required of the pump? Using a force balance over the appropriate control volume, show that this is independent of the viscosity of the fluid.

b. If the working fluid has the same viscosity as water, what should the flow rate of the pump supplying the fluid be? Give the answer in µL/min (microliters per minute).



Required Flow Rate in a Microchannel to Generate Shear Stress

Subject Tags: Microchannel Flow, Shear Stress, Flow Rate, Dimensional Analysis, Biomedical Applications

Difficulty Level: Moderate

Estimated Time: 20 minutes

Summary: This problem asks the student to determine the volumetric flow rate necessary to generate a specified wall shear stress in a microfluidic channel with given dimensions. The fluid is water, and the result should be expressed in microliters per minute. This application is relevant to studies of cell adhesion under flow in biomedical engineering.

Skills Tested:

- Use of channel geometry to relate shear stress to flow rate
- Application of Newtonian fluid mechanics in confined flows
- Unit conversions involving length and volumetric flow rate

Source: CBE 30355 Second Hour Exam, 2018, solved **File:** problem0079.tex

Required Flow Rate in a Microchannel to Generate Shear Stress

A microchannel of length 4 cm, width 2 mm, and depth 100 μm is used to study the adhesion of cells to the lower wall. If the required stress at the lower wall is 10 dynes/cm^2 and the working fluid is water, what is the required flow rate?

Give your answer in microliters per minute (μL/min) and show your work!

Oscillatory Unidirectional Flow in a Channel

Subject Tags: unidirectional flow, oscillatory pressure gradient, inertial limit, viscous limit, displacement amplitude

Difficulty Level: Moderate

Estimated Time: 35–45 minutes

Summary: This problem considers oscillatory flow in a narrow channel driven by a sinusoidal pressure gradient. Students analyze the simplified Navier-Stokes equations for this unidirectional flow, perform dimensional analysis to obtain scaling regimes for high and low frequency limits, and compute velocity profiles and displacement amplitudes for each case.

Skills Tested:

- Dimensional analysis and non-dimensionalization
- PDE simplification under physical assumptions
- Identifying and analyzing asymptotic limits
- Computing time-integrated quantities from velocity profiles

Source: CBE 30355 Secpmd Hour Exam, 2011, solved **File:** problem0034.tex

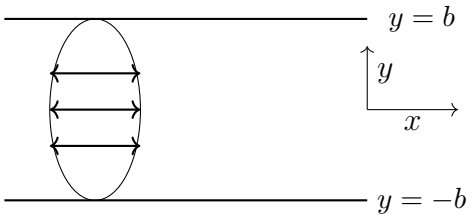
https://diffusivepress.org/courseware/cbe30355/

Oscillatory Unidirectional Flow in a Channel

Consider the unsteady oscillatory flow in a channel of width $2b$ depicted below. The fluid is incompressible, and the flow is unidirectional in the x-direction, with all that implies. We impose an oscillatory pressure gradient:

$$\frac{\partial p}{\partial x} = A\sin(\omega t)$$

where A is the amplitude and ω is the frequency of oscillation.



Flow is unidirectional in x

a. Write down the momentum equation in the x-direction and show which terms are zero. Render the equations dimensionless using $t^* = \omega t$ as the dimensionless time and U_c as an unknown velocity scale. Divide out by A to make the equations dimensionless. The characteristic velocity U_c is determined by balancing the driving force in the problem (the pressure gradient) with either the inertial or viscous term. Recognizing this, determine this characteristic velocity for:

- High frequencies
- Low frequencies

Explicitly identify the single dimensionless group the problem depends on.

b. Solve for the velocity profile in the high frequency limit. What happens to the boundary conditions?

c. Solve for the velocity profile in the low frequency limit.

d. For this sort of problem, we are primarily interested in the amplitude of the displacement of the fluid in the x-direction, which is easily obtained by integrating the velocity over one half period in time. Using the results of parts (b) and (c), determine the amplitude Δx at the centerline of the flow in both the high and low frequency limits.

Unidirectional Flow Between Cylinder and Rod

Subject Tags: Stokes flow; Cylindrical coordinates; Differential equations; Boundary conditions

Difficulty Level: Advanced

Estimated Time: 50 minutes

Summary: This problem analyzes the slow settling of a dense rod inside a fluid-filled vertical cylinder. Assuming the rod is long and the annular gap small, we treat the flow in the gap as unidirectional. The goal is to derive an expression for the volumetric flow rate, state the governing differential equation and boundary conditions, and determine the settling velocity in terms of density difference and fluid parameters.

Skills Tested:

- Setting up velocity profiles in cylindrical geometry
- Applying boundary conditions to viscous flow
- Deriving expressions from integral and differential forms
- Relating motion to buoyant force

Source: CBE 30355 First Hour Exam, 2012, solved **File:** problem0041.tex

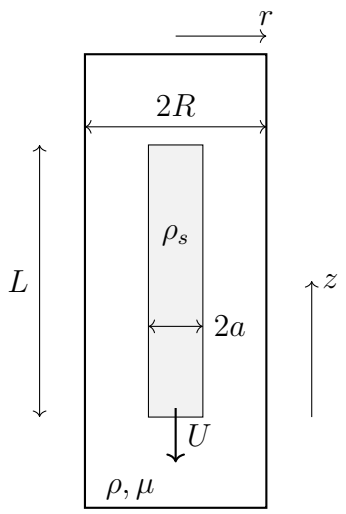
Unidirectional Flow Between Cylinder and Rod

Consider the system depicted below. A closed-end cylinder of radius R is filled with a fluid of density ρ and viscosity μ. A thick rod of radius a, length L, and density $\rho_s > \rho$ settles downwards under the action of gravity with velocity U, displacing fluid. If the length $L \gg a$, the flow in the gap may be regarded as unidirectional in the z-direction.

a. If the rod velocity is U, develop an integral expression for the fluid flow rate in the gap between the rod and the cylinder.

b. Write down the differential equation and boundary conditions (integral and otherwise) governing the velocity distribution in the gap.

c. Show how you can calculate the settling velocity U in terms of the density difference $\rho_s - \rho$. You do not need to evaluate the constants of integration (they are a bit messy), but make it sufficiently detailed that it is clear that you know how to get the final result.



The equations of motion in cylindrical coordinates are given by:

$$\tau_{rz} = \tau_{zr} = \mu\left(\frac{\partial u_z}{\partial r} + \frac{\partial u_r}{\partial z}\right)$$

$$\frac{1}{r}\frac{\partial}{\partial r}(ru_r) + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial u_z}{\partial t} + u_r\frac{\partial u_z}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_z}{\partial \theta} + u_z\frac{\partial u_z}{\partial z}\right) = -\frac{\partial p}{\partial z} - \rho g + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 u_z}{\partial \theta^2} + \frac{\partial^2 u_z}{\partial z^2}\right]$$

Flow Down an Incline

Subject Tags: Viscous flow; inclined plane; Couette flow; stress-free surface; dimensional analysis

Difficulty Level: Moderate

Estimated Time: 25 minutes (10 reading + 15 solution)

Summary: A viscous fluid is placed between two reservoirs connected by an inclined surface. A belt moves up the incline at constant speed U, dragging fluid upward, while gravity pulls the fluid downward. At steady state, the net flow is zero due to balance between gravity and belt motion. The goal is to determine the fluid depth h on the belt that satisfies this condition. The upper surface of the fluid is stress-free, and dimensionless formulation simplifies the analysis.

Skills Tested:

- Governing equations for steady viscous flow
- Applying boundary conditions, including stress-free surfaces
- Performing a force balance in an inclined coordinate system
- Making the problem dimensionless to simplify solution

Source: CBE 30355 First Hour Exam, 2016, solved **File:** problem0054.tex

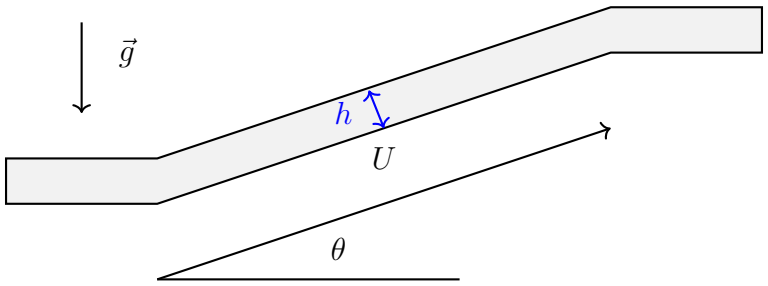
Flow Down an Incline

Two reservoirs containing a viscous fluid are connected by an incline as shown below. On the incline, a belt moves upward with constant velocity U, dragging fluid with it. Gravity acts in the opposite direction, pulling the fluid downward.

Eventually, the system reaches a steady state where the net volumetric flow rate is zero: the upward drag from the belt exactly balances the downward pull from gravity.

For a given belt velocity U, angle θ, and fluid properties μ and ρ, determine the **steady-state fluid depth** h on the moving belt. You may assume the upper surface of the fluid is stress-free.

Hint: The analysis becomes significantly easier if you first non-dimensionalize the governing equations!



Viscosity Measurement by Flow Down an Inclined Tray

Subject Tags: Unidirectional Flow, Scaling, Transient Flow, Viscosity Measurement

Difficulty: Moderate

Estimated Time: 30 minutes

This problem examines unidirectional transient flow of a viscous fluid down an inclined tray to estimate viscosity by tracking the surface velocity of the fluid. The key focus is on governing equations including transient terms, nondimensionalization with characteristic velocity and time scales based on steady state, and scaling arguments to estimate waiting time and fluid properties for validity of the method.

Skills Tested:

- Writing transient unidirectional flow governing equations
- Applying boundary conditions for flow down an incline
- Nondimensionalizing PDEs and identifying characteristic velocity and time scales
- Scaling and dimensional analysis to interpret physical constraints

Source: CBE 30355 Midterm Exam, 2024, solved File: problem0103.tex

Viscosity Measurement by Flow Down an Inclined Tray

In class we demonstrated how you could measure (if not terribly accurately) the viscosity of corn syrup by looking at flow down an incline in a shallow tray. The corn syrup was poured into the tray to a depth d, a marker was placed on the surface of the fluid, and then at time $t = 0$ the tray was tilted to an angle θ relative to the horizontal. The velocity of the marker was measured and used to calculate the kinematic viscosity.

The question here is how long we needed to wait for the velocity of the marker to achieve steady state and what kind of fluid we can use this approach for.

a. Write down the governing equations and boundary conditions for this unidirectional flow problem for a tray at angle θ. Don't forget the transient term!

b. Render the equations dimensionless, determining the characteristic velocity (use steady-state here, as that is what we were trying to measure!) and characteristic time.

c. How long would we have to wait to ignore the transient term? If the tray is of length L, what would be the approximate maximum depth or minimum viscosity we could get away with? (Hint: just combine U_c and t_c and compare to L...)

Inclined Film with Counteracting Shear

Subject Tags: Film Flow, Shear Stress, Inclined Plane, Viscous Flow, Gravity

Difficulty: Moderate

Estimated Time: 35 minutes

This problem involves the analysis of a very viscous film flowing down an inclined plane under the influence of gravity and a counteracting wind shear. The student is asked to determine the condition under which the surface velocity becomes zero, by applying unidirectional flow assumptions and solving the steady-state momentum balance. The role of external shear stress at the free surface is key to the outcome.

- Balance of viscous and gravitational forces
- Shear stress at a free surface
- Velocity profile in inclined flow
- Boundary condition application

Source: CBE 30355 First Hour Exam, 2004 **File:** problem0114.tex

Inclined Viscous Film with Wind Shear

Consider the velocity distribution acquired by the slow flow of a very viscous fluid down an incline, as is depicted below. In contrast to the problem considered in class, here we have a breeze trying to blow the fluid back up the incline, in addition to the gravity trying to draw the fluid down.

We model the effect of the breeze with a constant shear stress τ at the upper surface of the fluid (note: τ will be negative). For some film depth δ, angle θ, density ρ, and viscosity μ, how large does this shear stress have to be for the surface velocity (e.g., the velocity at $y = \delta$) to be zero?

Solve the problem assuming unidirectional flow.

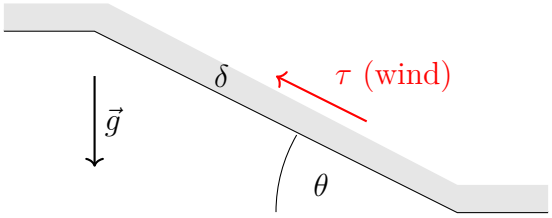


Figure 11: Inclined viscous film of thickness δ with gravity down the slope and wind-induced shear stress τ at the surface.

Breeze Opposes Gravity on Inclined Film

Tags: Film Flow, Shear Stress, Inclined Plane, Viscous Flow, Net Flow Rate

Difficulty: Moderate

Estimated Time: 30 minutes

A thin film of a very viscous fluid flows slowly down an inclined plane under the influence of gravity. A breeze opposes the flow by exerting a constant shear stress τ at the free surface (note that τ is negative). Assuming unidirectional steady flow, determine the value of τ required to make the **net flow rate equal to zero**. That is, what wind strength is needed to halt the average downhill transport of fluid? The film has depth δ, inclination angle θ, density ρ, and dynamic viscosity μ.

Skills Tested:

- Force balance in thin film flows
- Treatment of surface shear stress
- Integration of velocity profiles
- Zero net flow constraint

Source: CBE 30355 First Hour Exam, 2008 **File:** problem0131.tex

Net Zero Flow Condition on Inclined Film with Surface Shear

Consider the velocity distribution acquired by the slow flow of a very viscous fluid down an incline, as is depicted below. In contrast to the problem considered in class, here we have a breeze trying to blow the fluid back up the incline, in addition to gravity trying to draw the fluid down.

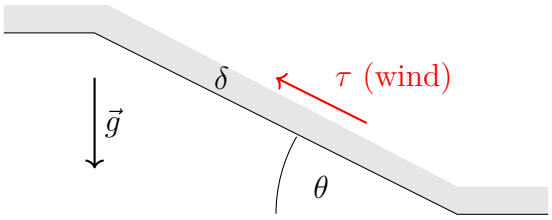


Figure 12: Inclined viscous film of thickness δ with gravity down the slope and wind-induced shear stress τ at the surface.

We model the effect of the breeze with a constant shear stress τ at the upper surface of the fluid (note: τ will be negative). For some film depth δ, angle θ, density ρ, and viscosity μ, how large does this shear stress have to be for the **net fluid velocity** (i.e., the total volumetric flow rate) to be zero?

This would be the point where the fluid stops going down the plane (on average) and starts going back up it. Solve the problem assuming unidirectional flow.

Continuous Coating of a Sheet with a Viscous Film

Subject Tags: Viscous Flow, Film Coating, Lubrication Theory, Optimization

Difficulty Level: Advanced

Estimated Time: 50 minutes

Summary: A sheet is pulled through a bath of viscous fluid and then up an incline at velocity U, coating the sheet with liquid. It then enters a horizontal region where the coating solidifies and moves at the velocity of the belt. This problem asks for an expression for the final coating thickness in the horizontal region, starting from flow and profile analysis on the incline.

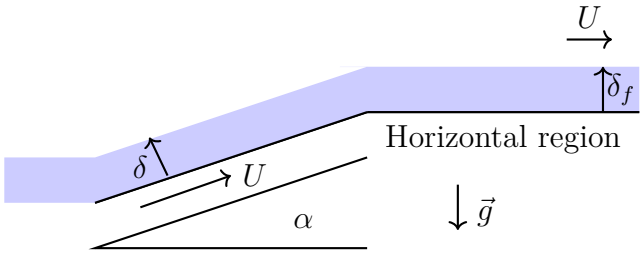
Skills Tested:

- Mass conservation for coating flows
- Relation between flow rate and velocity profile
- Use of lubrication theory and flow optimization
- Determination of extremum conditions to solve for optimal thickness

Source: CBE 30355 Second Hour Exam, 2010 **File:** problem0138.tex

Continuous Coating of a Sheet with a Viscous Film

A variant on conventional dip coating is depicted below. A sheet is coated with a viscous liquid by being drawn through a bath and up an incline at some angle α at velocity U. The coated sheet then enters a horizontal area where the liquid solidifies (e.g., over this region the coating moves with the velocity of the belt). Your mission is to calculate the final thickness of the coating film δ_f in the horizontal area.



a. How does the final film thickness δ_f depend on the net fluid flow up the incline (per unit width "into the board") Q/W and the belt velocity U? (This is easy!)

b. The thickness of the coating fluid on the incline δ will be different from the thickness δ_f in the horizontal area due to the different velocity profiles. Calculate how Q/W depends on δ and the parameters of the problem.

c. If surface tension is negligible, the thickness of the coating film on the incline will be that which results in the maximum Q/W. Using this information, determine this thickness and use it to calculate the final thickness δ_f as a function of the parameters of the problem.

Unidirectional Flows (or the heat transfer analog!)

Subject Tags: heat conduction; interface conditions; thermal resistance; dimensional analogy

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: This problem models the buildup of a solid radioactive layer inside a reactor vessel by analyzing heat conduction through a two-layer wall. By drawing an analogy to unidirectional shear flow, students are asked to derive the unknown layer thickness from measurable thermal parameters. The problem emphasizes the continuity of heat flux and temperature at interfaces.

Skills Tested:

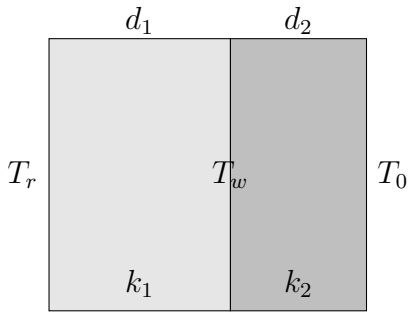
- Heat flux continuity
- Use of analogies with momentum transfer
- Interface matching conditions
- Algebraic derivation of unknowns

Source: CBE 30355 Second Hour Exam, 2017, solved **File:** problem0071.tex

Unidirectional Flows (or the heat transfer analog!)

The waste cleanup process currently being implemented at Idaho National Lab involves a reactor where the waste is calcified for subsequent treatment. While it is supposed to be fluidized, unfortunately the "sand" produced in the reaction tends to form a solid layer on the inside of the reactor vessel, and they need to know when this occurs (it's highly radioactive, so they can't just look inside!).

Last summer I proposed that they look at heat transfer through the vessel surface as a probe of the thickness of the solid layer, resulting in the system depicted below:



The two conductivities k_1 and k_2, the three temperatures T_r, T_w, and T_0, and the outer layer thickness d_2 are all known or measurable.

a. Using this, determine what the thickness of the solid layer d_1 deposited on the wall is in terms of these values.

b. The equations governing heat transfer in a solid are:

$$\rho C_p \frac{\partial T}{\partial t} = k\nabla^2 T, \quad \vec{q} = -k\nabla T$$

Note: This problem is identical to the unidirectional flow problem where the temperatures are equivalent to velocities, the conductivities are identical to viscosities, and the heat flux $\vec{q}$ is analogous to the shear stress. Remember what has to be continuous at an interface...

Electroosmotic Mobility in Microchannels

Subject Tags: Electroosmosis, Fluid Mechanics, Microfluidics, Dimensionless Analysis

Difficulty: Moderate

Estimated Time: 30 minutes

This problem analyzes electroosmotic flow driven by tangential electric fields over charged surfaces in microchannels. The flow velocity profile arises from a balance between viscous forces and a body force due to the electric field acting on a charge density exponentially decaying from the surface. The problem involves formulating governing equations, nondimensionalizing, solving for velocity profiles, and interpreting electroosmotic mobility.

Skills Tested:

- Writing governing differential equations including body forces
- Applying boundary conditions in fluid mechanics problems
- Nondimensionalization of PDEs/ODEs
- Analytical solution of linear ODEs
- Physical interpretation of dimensionless parameters and flow quantities

Source: CBE 30355 Midterm Exam, 2024, solved File: problem0102.tex

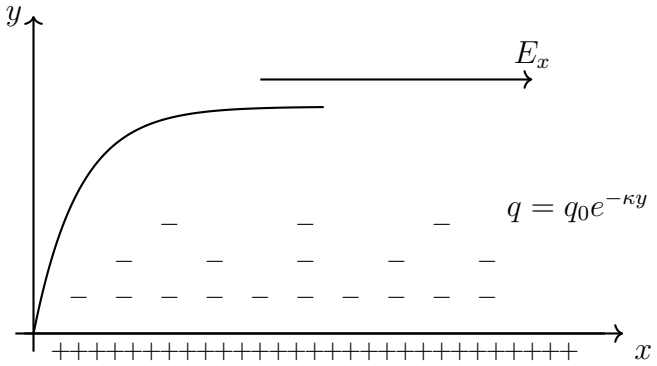
Electroosmotic Mobility Problem

Consider the surface depicted below. In an aqueous system ionic species are often adsorbed onto the surface (these are fixed charges), and counter-ions are distributed in a very thin diffuse layer above the surface (this occurs due to diffusion). The net free charge density q decays exponentially away from the surface, e.g.,

$$q(y) = q_0 e^{-\kappa y}$$

where q_0 is the charge density at $y = 0$ and the constant κ^{-1} is known as the Debye length (usually just a few microns or less!). If you have a tangential electric field E_x (e.g., in the x-direction), you get a body force (just like gravity!) in the x-direction of magnitude $E_x q(y)$ that dies away exponentially far from the surface.

The boundary conditions are no-slip at the surface $y = 0$ and zero shear stress far away (which because the Debye length is so short isn't very far at all!).



Using this:

a. Write down the equation governing the velocity profile $u(y)$ in the x-direction including this body force.

b. Render the equation dimensionless and solve it to get the velocity profile.

c. The electroosmotic mobility is just the velocity far away divided by the electric field strength. What is it?

Parallel-Plate Flow

Tags: Creeping Flow, Rotating Plate, Viscosity Measurement, Experimental Error

Difficulty: Moderate

Estimated Time: 40 minutes

A controlled stress rheometer operates by imposing a known torque on a rotating upper plate and measuring the resulting angular velocity. The fluid's viscosity can then be inferred from the relationship between torque and angular velocity. This problem explores the derivation of that relationship under low Reynolds number conditions and the correction for systematic error in gap width measurements.

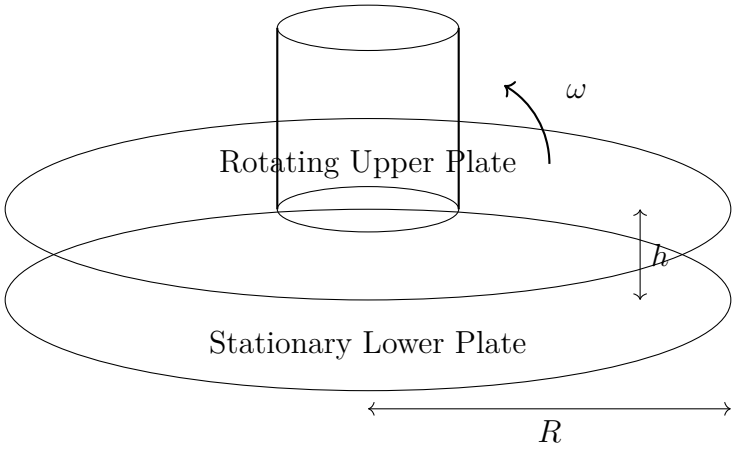
Skills Tested:

- Stokes Flow Approximation
- Torque Balance in Cylindrical Coordinates
- Experimental Design and Error Correction

Source: CBE 30355 Final Exam, 2007 **File:** problem0129.tex

Parallel-Plate Flow

A controlled stress rheometer is depicted below. It works by imposing some specified torque to the upper plate, and then measuring the resulting angular velocity. The viscosity of the fluid between the rotating upper plate and stationary lower plate is proportional to the ratio of the torque and the resulting angular velocity.



a. Solve for the relationship between the torque T and the angular velocity ω in terms of the gap width h, the plate radius R, and the fluid viscosity μ. For this system it is appropriate to assume low Reynolds number flow, greatly simplifying the problem!

b. Experimentally, the measured gap width often has some error in it, principally due to errors in the "zero" as you examined in a problem earlier this semester. If the actual gap width is $h = h_{\text{meas}} + \Delta h$, explicitly show how you can determine the correct viscosity by measuring ω for two different measured gap widths h_1 and h_2 at the same torque T.

You may find the following equations helpful:

$$\frac{1}{r}\frac{\partial(rv_r)}{\partial r} + \frac{1}{r}\frac{\partial v_\theta}{\partial \theta} + \frac{\partial v_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial v_\theta}{\partial t} + v_r\frac{\partial v_\theta}{\partial r} + \frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta} + \frac{v_r v_\theta}{r} + v_z\frac{\partial v_\theta}{\partial z}\right) = \mu\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(rv_\theta)\right) + \frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2} + \frac{\partial^2 v_\theta}{\partial z^2} + \frac{2}{r^2}\frac{\partial v_r}{\partial \theta}\right)$$

$$T = \int_A r\tau_{z\theta}\, dA \quad \text{where} \quad \tau_{z\theta} = \tau_{\theta z} = \mu\left[\frac{\partial v_\theta}{\partial z} + \frac{1}{r}\frac{\partial v_z}{\partial \theta}\right]$$

Lubrication Flow Under a Plate

Subject Tags: Lubrication Theory, Radial Flow, Thin Film Approximation, Force Balance

Difficulty Level: Moderate

Estimated Time: 35 minutes

Summary: A circular plate is supported above a surface by a thin radial film of viscous fluid entering at the center. You are asked to determine the equilibrium height (gap width h) based on a balance between the viscous support force and the weight of the plate. The lubrication approximation is assumed to apply.

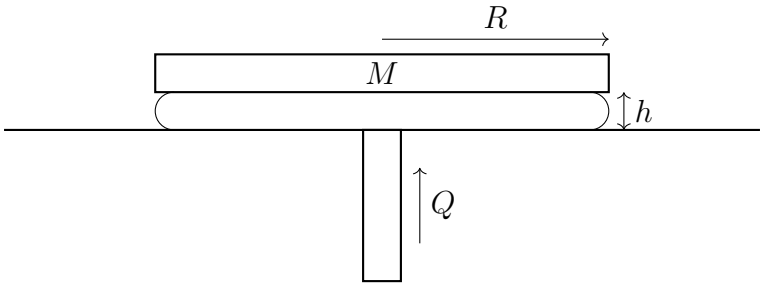
Skills Tested:

- Flow Analysis in Lubrication Limit
- Integral Force Approximation
- Use of Dominant Balance in Equations of Motion
- Application of Pressure-Force Relationships

Source: CBE 30355 Final Exam, 2017 **File:** problem0158.tex

Lubrication Flow Under a Plate

A plate of radius R is supported by the radial flow of a viscous liquid supplied through a hole in the center at a volumetric flow rate Q, as depicted below. If the mass of the plate is M, determine the gap width h between the two surfaces.



Note: recall from Calculus II that

$$\int r \ln(r)\, dr = \frac{1}{2} r^2 \ln(r) - \frac{1}{4} r^2.$$

Also, the diameter of the hole supplying the lubricant doesn't matter, as long as it is small (although it does affect the pressure in the supplying pipe to get a particular flow rate, of course).

The following equations may be helpful (note: the vast majority of these terms may be neglected!):

$$\frac{1}{r}\frac{\partial}{\partial r}(r u_r) + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial u_r}{\partial t} + u_r\frac{\partial u_r}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_r}{\partial \theta} - \frac{u_\theta^2}{r} + u_z\frac{\partial u_r}{\partial z}\right) = -\frac{\partial p}{\partial r} + \mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial (r u_r)}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 u_r}{\partial \theta^2} - \frac{2}{r^2}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial^2 u_r}{\partial z^2}\right] + \rho g_r$$

Falling Slug Rheometry in Cylindrical Geometries

Subject Tags: Lubrication Theory, Low Reynolds Number, Stokes Flow, Sedimentation, Wall Effects

Difficulty Level: Advanced

Estimated Time: 45 minutes

Summary: This problem explores wall effects in low Reynolds number sedimentation by analyzing a cylindrical slug falling in a narrow annular gap. Simplified geometries isolate the influence of shear and pressure in both open and closed-end cylinders, offering insight into the dominant contributions under lubrication flow conditions.

Skills Tested:

- Application of Archimedes' principle
- Shear stress calculation in narrow gaps
- Lubrication theory for pressure-driven flow
- Force balance under Stokes regime
- Asymptotic reasoning for $h \ll a$

Source: CBE 30355 Final Exam, sp2021 **File:** problem0167.tex

Falling Slug Rheometry in Cylindrical Geometries

In the class demonstration of falling ball rheometry, it was shown that for a finite sphere diameter/cylinder diameter ratio, the proximity of the walls had a large effect on the sedimentation velocity. This arose from two effects: First, the proximity of the wall increased the shear stress at the surface of the sphere due to the imposition of a no-slip boundary nearby. Second, the cylinder bottom caused there to be a backflow in the gap between the sphere and the cylinder walls due to the downward motion of the sphere.

This wall effect is very complicated to derive. However, here we look at a vastly simpler problem: what is the sedimentation velocity of a cylindrical slug of length L and radius a centered in a cylinder of radius R, where the gap between the two is $h = R - a \ll a$?

a. If the slug has a density ρ_s and the fluid has a density ρ, what is the net gravitational force on the slug? (Hint: don't forget Archimedes' Law...)

b. Now consider the case where the cylinder is open at both ends (say it is completely immersed in the fluid—not exactly practical!). No pressure gradient would be produced by the slug motion in this case. If the slug moves with velocity U_s, calculate the force on the slug due to shear in the gap and, by equating this force with the gravitational force calculated in (a), determine how U_s depends on the parameters of the problem.

c. Now we do the more realistic problem of a closed-end cylinder. The downward displacement of the slug leads to an upward displacement of the fluid in the gap and a large pressure difference due to this lubrication flow. While the force due to the shear stress in the gap contributes (and is even greater than that calculated in (b)), by far the dominant contribution for $h \ll a$ is the pressure difference acting on the ends of the slug. Using this insight, calculate the sedimentation velocity of the slug in a closed-end cylinder to leading order in h/a.

Stratified Shear Flow Between Plates

Subject Tags: fluid mechanics, stratified flow, Navier-Stokes, shear stress, boundary conditions

Difficulty Level: Moderate

Estimated Time: 25 minutes

A two-layer fluid is sheared between two infinite parallel plates. The upper plate moves at velocity U, while the lower plate is fixed. Each fluid layer has a distinct viscosity and thickness: μ_1, h_1 for the lower layer and μ_2, h_2 for the upper layer. This problem asks for momentum balance equations, velocity and shear stress distributions, and volumetric flow rates for each fluid. Proper continuity at the interface must be enforced.

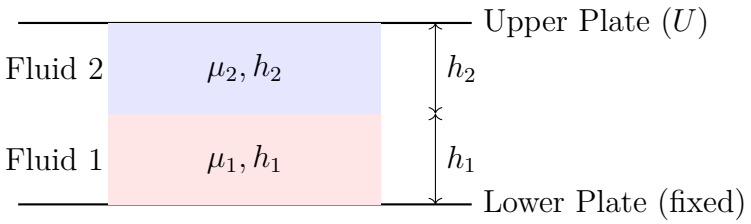
Skills Tested:

- Formulating reduced Navier-Stokes equations for laminar flow
- Applying continuity of velocity and shear stress
- Determining velocity profiles in layered shear flow
- Computing force and flow rate per unit width

Source: CBE 30355 1st Hour Exam, 2001 **File:** problem0005.tex

Two-Layer Shear Flow Between Plates

Consider the stratified fluid depicted below which is being sheared between two infinite parallel plates. If the upper plate is constrained to move with velocity U, the lower plate is fixed, and the viscosities and thicknesses of the two layers are μ_1, μ_2, and h_1, h_2, respectively, calculate the following:



a. Write down the Navier-Stokes equation for momentum in the direction of flow in each region and show which terms are non-zero.

b. What are the velocity and shear stress distributions? What is the force on the lower plate? Remember that both velocity and shear stress must be continuous functions of position. This determines the conditions at the interface between the two fluid layers.

c. Calculate the flow rate of each fluid (per width W out of the plane of the paper).

Yield Stress Measurement in Ranch Dressing

Subject Tags: Non-Newtonian Fluids, Yield Stress, Pressure-Driven Flow, Manometry

Difficulty Level: Moderate

Estimated Time: 20 minutes

Summary: This problem explores a simple experimental setup for determining the yield stress of a food product—ranch dressing—using a pressure gradient measured via a water manometer. The student is asked to derive a formula for yield stress based on system geometry and to compute a specific value given numerical data.

Skills Tested:

- Deriving expressions for yield stress in cylindrical coordinates
- Applying manometry to fluid statics
- Substituting and converting physical dimensions
- Understanding rheological behavior of non-Newtonian fluids

Source: CBE 30355 Final Exam, 2021 **File:** problem0170.tex

Yield Stress Measurement in Ranch Dressing

Food rheology is very important for all sorts of reasons. One desirable trait (for certain products) is a yield stress, the minimum stress at which a fluid will flow (zero for Newtonian fluids such as air, water, and honey). In class it was demonstrated how a tube, a pressure source, and a water manometer could be used to determine the yield stress of ranch dressing.

a. If the tube filled with the dressing is of radius a and length L, and the dressing first starts to flow when the height difference of the water in the manometer monitoring the pressure reaches a value h, develop a formula for the yield stress in terms of these parameters.

b. If the radius is 1 mm, the length is 10 cm, and the height difference is 5 cm, what is the quantitative value of this yield stress?

Quasi-Parallel Flows and Lubrication

In this section are gathered problems dealing with viscous flows in two dimensions, but where there is a separation of length scales (e.g., quasi-parallel flow or lubrication flows). These are closely related to the unidirectional flow problem, of course, but allow examination of things like flow through a porous pipe or channel, or the drainage of a film from the surface of a cylinder. The key is that the flow in the "thin" direction is of $O(\epsilon)$ (thinness ratio) and is calculated from the continuity equation.

Lubrication Flow in a Collapsing Wedge

Subject Tags: Lubrication Approximation; Dimensional Analysis; Wedge Flow; Pressure Distribution; Torque Calculation

Difficulty: Advanced

Estimated Time: 45 minutes

Summary: This problem examines lubrication flow in a collapsing wedge with a small opening angle. Students are asked to derive the governing equations and boundary conditions, perform a dimensional scaling analysis to determine torque dependence on the angle rate of change, and compute the pressure distribution and resulting torque.

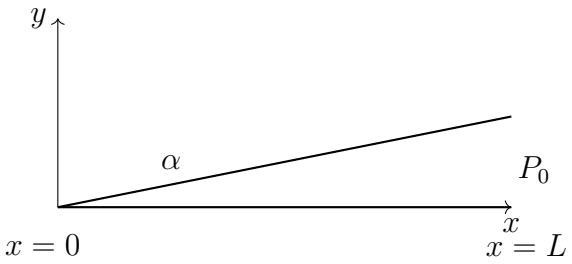
Skills Tested:

- Governing Equation Setup for Lubrication Flows
- Application of Small-Angle Approximations
- Dimensional and Inspectional Analysis
- Torque Calculation from Pressure Fields

Source: CBE 30355 2nd Hour Exam, 2001 **File:** problem0011.tex

Lubrication Flow in a Collapsing Wedge

Lubrication Flow in a Collapsing Wedge. For homework you solved the 2-D streamfunction flow for a collapsing wedge as depicted below. In this problem, however, we look at the simpler lubrication case where the angle of the wedge is very small (e.g., $\alpha \ll 1$).



a. Set up the equations and boundary conditions governing this problem in the lubrication limit.

Hint: Use a length scale $H = \alpha L$ and velocity scale $V_c = L\,\frac{d\alpha}{dt}$ in the y direction, and recall that from trigonometry $\tan\alpha \approx \sin\alpha \approx \alpha$ for $\alpha \ll 1$. Remember that the fluid displaced by the closing wedge must squish out in the x-direction.

b. Using dimensional and inspectional analysis, determine the appropriate scaling for the torque on the hinge as a function of $\frac{d\alpha}{dt}$ and all the relevant physical parameters. The torque per extension in the third dimension exerted by the fluid on the plate is given by:

$$\frac{T}{W} = \int_0^L (P - P_0) x \, dx$$

c. Solve for the pressure distribution and numerical value of this torque.

Lubrication and Cavitation in Separating Plates

Subject Tags: Lubrication theory; Cavitation; Pressure distribution; Torque balance

Difficulty Level: Advanced

Estimated Time: 40 minutes

Summary: Cavitation occurs in lubrication flows when local pressures fall below the vapor pressure of the liquid. In this problem, the gap between two diverging plates is governed by a small angle θ and changes with time. Lubrication theory is used to determine where cavitation will occur and to calculate the force required to separate the plates at a given angular rate.

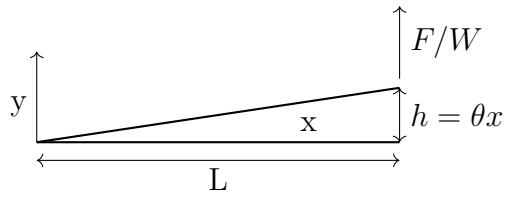
Skills Tested:

- Lubrication approximation and velocity profile construction
- Pressure distribution and cavitation location
- Torque balance and force derivation

Source: CBE 30355 Secpmd Hour Exam, 2011, solved **File:** problem0032.tex

Lubrication and Cavitation in Separating Plates

Cavitation is the phenomenon which occurs when the absolute pressure in a liquid reaches the vapor pressure, and the liquid boils. For most liquids this is pretty close to zero. While usually seen in inertial flows, it is actually most easily visualized in viscous lubrication. Consider the geometry depicted below:



Two plates of length L in contact at one edge are separated by a very small angle θ. For small angles the gap between the plates is given by

$$h(x) = \theta x$$

where x is the distance from the vertex. The plates are pulled apart at some rate $\dfrac{d\theta}{dt}$.

a. Using lubrication analysis, determine the location x_c where the pressure falls to zero absolute (e.g., a gauge pressure of $-p_{\text{atm}}$) and the fluid cavitates.

b. What is the force per unit width into the paper F/W (exerted at the outer edge) necessary to pry the plates apart for some angular velocity $\dfrac{d\theta}{dt}$? (Hint: Think of torque balances! You may leave this in terms of an integral if you wish.)

Lubrication: Closure of a Hinged Plate

Subject Tags: Lubrication theory; Scaling analysis; Torque balance; Mass conservation

Difficulty: Advanced

Estimated Time: 45 minutes

Summary: A plate hinged at one end is being squeezed shut under an applied force per unit width. For small opening angles, lubrication theory applies. This problem asks for governing equations, a scaling analysis of closure rate Ω, and an integral solution for the torque balance.

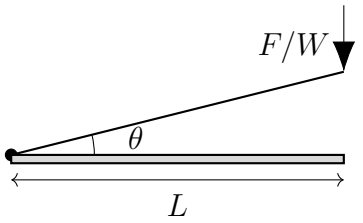
Skills Tested:

- Application of lubrication approximation
- Use of scaling arguments to estimate behavior
- Formulation of torque and mass balances
- Integral representation of mechanical constraints

Source: CBE 30355 Second Hour Exam, 2012, solved **File:** problem0044.tex

Lubrication: Closure of a Hinged Plate

A hinged plate of extension W into the page and of length L, as depicted below, is being squeezed shut with a force per unit width F/W. The angle of the plate with respect to the horizontal is small ($\theta \ll 1$). We wish to determine the closure rate $\Omega = -\dfrac{d\theta}{dt}$ as a function of the parameters of the problem.



a. Write down the governing differential equations, boundary conditions, mass balance, and torque balance about the hinge necessary to solve the problem in the lubrication limit.

b. Using a scaling analysis of the equations in part (a), determine how Ω depends on the parameters of the problem to within an unknown constant.

c. Solve the problem. You may leave the final result for the torque balance in an integral form if you wish.

Lubrication Theory: Detachment of a Strip from a Plane

Subject Tags: lubrication theory; scaling analysis; pressure distribution; dimensionless equations

Difficulty: Advanced

Estimated Time: 40 minutes

Summary: This problem explores the lubrication-theory-based analysis of a long strip detaching from a plane. Students are first asked to simplify the governing equations under the lubrication limit, then to use scaling arguments to estimate the separation behavior, and finally to solve for the pressure distribution and separation time explicitly.

Skills Tested:

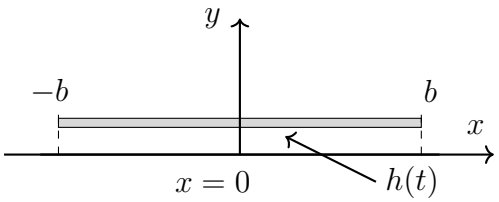
- Lubrication theory and order-of-magnitude analysis
- Dimensional analysis and non-dimensionalization
- Solving simplified Navier-Stokes equations under limiting assumptions

Source: CBE 30355 Second Hour Exam, 2002 **File:** problem0021.tex

Detachment of a Strip from a Plane

Lubrication Theory: In class we demonstrated the detachment of a disk from a plane. In this problem we solve the corresponding 2-D problem of the detachment of a long strip of width $2b$ from a plane. If we apply an upward force per unit length (extension into the paper—the direction that doesn't matter) of F/L, what is the time for the strip to separate from the plane given an initial separation distance $h = h_0$?

A diagram of the geometry is shown below:



a. Write down the continuity equation and the x-momentum equation, render the equations dimensionless, and determine which terms are significant in the lubrication limit.

b. Using inspectional analysis, derive the equation governing the separation as a function of time to within some unknown integral (e.g., don't solve the lubrication problem yet—just do the scaling).

c. Solve the lubrication equations to explicitly obtain the pressure distribution and separation time.
(This part takes a lot of time, so don't do it until you've worked on the other questions!)

Lubrication Force on Cone in Contact with Plane

Subject Tags: Lubrication Theory, Scaling, Momentum Balance, Cylindrical Coordinates

Difficulty: Advanced

Estimated Time: 50 minutes

This problem examines the lubrication flow between a stationary cone and a moving plane. By invoking the lubrication approximation in cylindrical coordinates and applying appropriate boundary conditions, the student must determine the pressure distribution and the total force required to separate the cone from the surface. Dimensional analysis reveals the scaling of the required force with physical parameters.

- Application of lubrication approximation in cylindrical coordinates
- Use of symmetry and thin film scaling
- Dimensional analysis and scaling of pressure
- Calculation of force from pressure distribution

Source: CBE 30355 Second Hour Exam, 2004 **File:** problem0115.tex

Lubrication Force on Cone in Contact with Plane

Lubrication: A cone of radius R and angle α is initially in contact with a plane as depicted below. We want to use lubrication theory to determine the force F necessary to pull the cone off the plane with some velocity V in the limit of small α. In this limit, the gap width is simply αr.

(Hint: If R is the length scale in the radial direction, what is the length scale in the z-direction???)

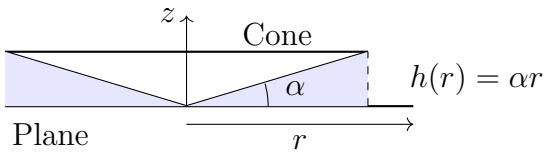


Figure 13: Geometry of cone in contact with plane in lubrication limit.

a. The radial momentum equation and continuity equation are given below. Making use of symmetry, eliminate terms which are identically zero, and write down the equations and boundary conditions which govern this problem.

$$\frac{1}{r}\frac{\partial}{\partial r}(ru_r) + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial u_r}{\partial t} + u_r\frac{\partial u_r}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_r}{\partial \theta} - \frac{u_\theta^2}{r} + u_z\frac{\partial u_r}{\partial z}\right) = -\frac{\partial p}{\partial r} + \mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial (ru_r)}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 u_r}{\partial \theta^2} - \frac{2}{r^2}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial^2 u_r}{\partial z^2}\right] +$$

b. Render the equations dimensionless in the lubrication limit, and show which terms are important. By determining the scaling for pressure (and rendering the equation for the required force dimensionless as well), show how the force scales with the parameters R, V, μ, and α.

c. Solve the problem.

Lubrication: Detachment of a Sphere from a Plane

Tags: Lubrication Theory, Creeping Flow, Scaling, Pressure Distribution, Reynolds Equation

Difficulty: Advanced

Estimated Time: 40 minutes

A sphere of radius a is moving vertically away from a flat wall, with the minimum gap between them given by h_0. In the narrow gap limit, where $h_0/a \ll 1$, lubrication theory provides an approximate description of the fluid mechanics in the gap. The gap thickness $h(r)$ as a function of radial position r (in cylindrical coordinates) is approximately given by:

$$h(r) = h_0 + \frac{r^2}{2a}$$

In this problem, you will derive the pressure distribution and fluid force under creeping flow conditions and estimate the required force to maintain a given velocity V of the sphere.

Skills Tested:

- Application of lubrication theory
- Derivation of pressure-driven flow equations
- Use of scaling arguments for force estimation
- Integral formulation and evaluation

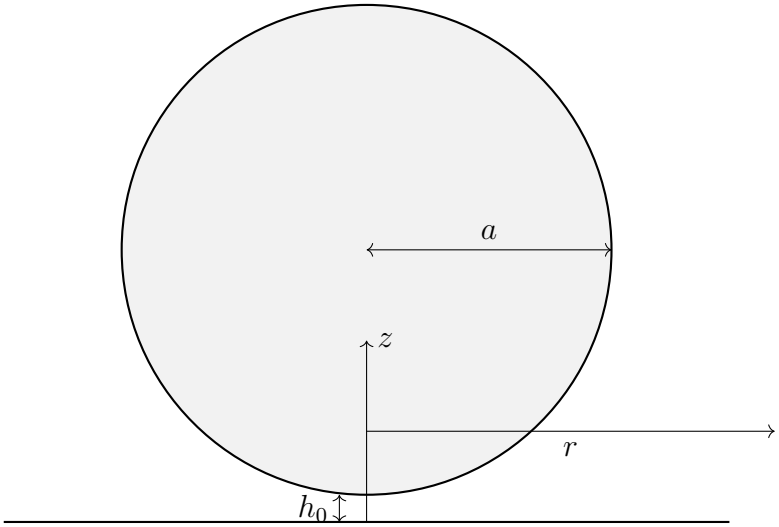
Source: CBE 30355 Second Hour Exam, 2008 **File:** problem0132.tex

Sphere Separating from a Plane in the Lubrication Limit

Consider the geometry depicted below. A sphere of radius a is separated from a plane by a distance h_0 such that $h_0/a \ll 1$. In the narrow gap limit, the gap width $h(r)$ is given approximately by

$$h(r) = h_0 + \frac{r^2}{2a}$$

where r is the radial distance from the point of minimum separation.



a. Determine the equations which govern the pressure distribution and force on the sphere in the lubrication limit.

b. Using scaling analysis, determine the force necessary for the sphere to have a vertical velocity V for small h_0/a, to within some unknown numerical value.
(Note: the appropriate length scale for r is *not* the radius of the sphere!)

c. Solve for the numerical value. You can leave the dimensionless force and pressure in terms of integrals if you wish, but they are actually easy to evaluate if you do it right...

Spin Coating and Thin Film Dynamics

Subject Tags: Lubrication theory, Scaling analysis, Spin coating, Unsteady flow

Difficulty: Advanced

Estimated Time: 45 minutes

Summary: Spin coating is widely used to produce thin films on surfaces by depositing a viscous fluid and rotating the substrate at high speed. This problem analyzes the velocity profile and film thickness evolution in such a process. It involves formulating the governing equations, applying appropriate boundary conditions, estimating time scales using scaling analysis, and solving the equations to extract the full time-dependent solution.

Skills Tested:

- Deriving PDEs in cylindrical coordinates under thin-film approximation
- Applying boundary conditions in unsteady lubrication theory
- Scaling analysis and asymptotic behavior
- Solving simplified Navier–Stokes equations

Source: CBE 30355 Final Exam, 2013, solved **File:** problem0048.tex

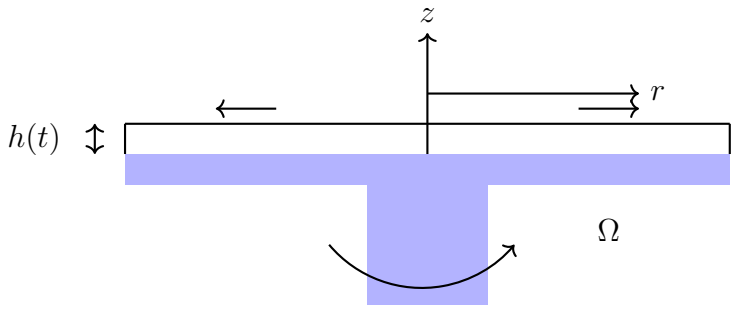
Spin Coating and Thin Film Dynamics

Spin coating is a common way to produce a thin film on a surface: a disk of radius R is coated with a viscous liquid of kinematic viscosity ν, and rotated with angular velocity Ω. Centrifugal forces thin the fluid layer over time, leaving a final thickness $h_f \ll R$.

a. For thin fluid layers the rotational velocity is just $u_r = \Omega r$ over the entire film (i.e., it moves with the rotational velocity of the disk). Using this, determine the differential equations and boundary conditions which govern the radial and vertical velocity distribution in the film, and the equation for the change in film thickness over time. (Hint: How does the change in thickness with time relate to u_z at $z = h(t)$?) It is appropriate to use h_f, the final thickness, as the vertical length scale.

b. Using scaling analysis, determine how long the spin coating process takes as a function of the parameters of the problem to within the usual unknown $\mathcal{O}(1)$ constant.

c. Explicitly solve for the velocity distribution and the height as a function of time to get the $\mathcal{O}(1)$ constant. You may take the initial height to be H, where $H/h_f \gg 1$.



$$\frac{1}{r}\frac{\partial}{\partial r}(ru_r) + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial u_r}{\partial t} + u_r\frac{\partial u_r}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_r}{\partial \theta} - \frac{u_\theta^2}{r} + u_z\frac{\partial u_r}{\partial z}\right) = -\frac{\partial p}{\partial r} + \mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial (ru_r)}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 u_r}{\partial \theta^2} - \frac{2}{r^2}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial^2 u_r}{\partial z^2}\right] + \rho g_r$$

Scaling and Quasi-Parallel Flow in a Porous Fiber

Subject Tags: Scaling analysis; Quasi-parallel approximation; Momentum equations; Continuity; Axial and radial flow

Difficulty Level: Advanced

Estimated Time: 35 minutes (15 scaling, 5 mass balance, 15 full solution)

Summary: This problem models fluid perfusion through a long, narrow, hollow fiber used to deliver nutrients into a tissue medium. Students are required to apply scaling analysis and quasi-parallel approximations to derive dimensionless equations, identify boundary conditions, and solve for velocity fields and pressure gradients using fundamental conservation laws. The axial flow is modified by wall injection, and simplifying assumptions aid in tractable analysis.

Skills Tested:

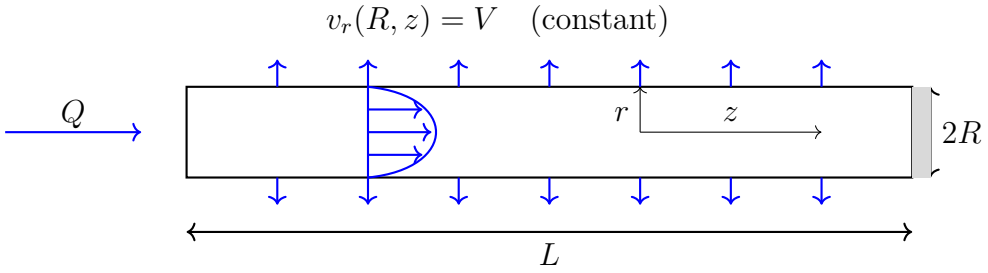
- Dimensional analysis and scaling of PDEs
- Application of quasi-parallel flow approximation
- Mass conservation under radial perfusion
- Analytical solution of simplified momentum and continuity equations
- Coupling radial and axial velocities

Source: CBE 30355 Second Hour Exam, 2016, solved **File:** problem0057.tex

Scaling and Quasi-Parallel Flow in a Porous Fiber

It is desired to use a hollow, porous fiber to perfuse nutrient into a tissue medium. The inner diameter of the fiber is $2R$, and its length is L, with the slenderness condition $R/L \ll 1$. The end of the fiber is sealed, so all fluid entering at $z = 0$ exits radially through the porous walls.

Although the radial perfusion velocity is normally a function of axial position z, we make the simplifying assumption that the radial velocity at the fiber wall is constant and independent of z, i.e.,



a. Render the governing equations dimensionless by appropriate scaling. By scaling the momentum and continuity equations, determine the simplified dimensionless equations and boundary conditions which govern the axial and radial velocity and pressure in the limit $R/L \ll 1$ and $VR/\nu \ll 1$.

b. Using conservation of mass (or volume, since the fluid is incompressible), determine the average axial velocity as a function of z.

c. Using your knowledge of flow through a pipe, solve for the velocity distribution in both the r and z directions and the pressure gradient.

Hint: After solving for v_z and the pressure gradient, you can obtain v_r from the continuity equation. You can significantly shorten the calculations by recalling the ratio of centerline to average axial velocity for this geometry!

d. The following governing equations are provided (note: many terms are negligible or zero in this context):

$$\frac{1}{r}\frac{\partial(rv_r)}{\partial r} + \frac{1}{r}\frac{\partial v_\theta}{\partial \theta} + \frac{\partial v_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial v_z}{\partial t} + v_r\frac{\partial v_z}{\partial r} + \frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta} + v_z\frac{\partial v_z}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left(\frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2} + \frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right) + \frac{\partial^2 v_z}{\partial z^2}\right) + \rho g_z$$

Quasi-parallel flows

Subject Tags: quasi-parallel flow; porous boundary; microfluidics; streamfunction; plasma separation

Difficulty: Advanced

Estimated Time: 40 minutes

Summary: This problem models plasma skimming in a microfluidic device with a porous top wall using the quasi-parallel flow approximation. It leads students through formulating and nondimensionalizing the governing equations, solving for both components of the velocity field, and computing the streamfunction. The system mimics physiological separation processes, requiring understanding of fluid element trajectories.

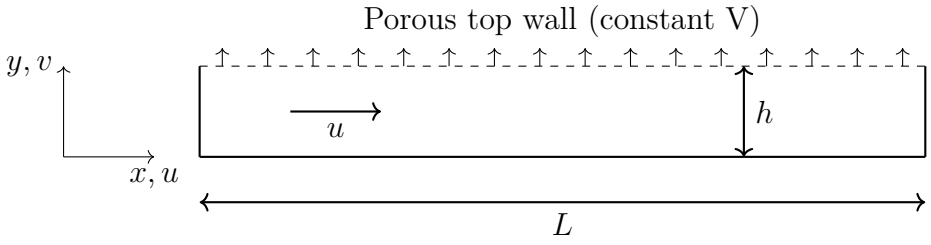
Skills Tested:

- Derivation of governing equations under the quasi-parallel approximation
- Application of boundary conditions in a porous geometry
- Nondimensionalization
- Velocity profile derivation
- Streamfunction computation

Source: CBE 30355 Final Exam, 2016, solved **File:** problem0060.tex

Quasi-parallel flows

Earlier this semester I was asked to evaluate a microfluidic plasma skimming device that was under consideration for licensing. While the complete system was more complicated, at its core the separator consisted of a channel where the top wall was porous as depicted below. Whole blood is drawn into the channel from the left end by the removal of plasma through the porous plate at the top. To determine the fate of the red blood cells, it is necessary to know the complete velocity profile and stream function (the cells tend to settle away from the upper surface, allowing cell-free plasma to be drawn off – but you have to know the velocity profile of the fluid to figure out if they settle fast enough). Here we determine this velocity profile!



a. By making the quasi-parallel flow approximation (and using a mass balance), determine the equations and boundary conditions governing the velocities in the x and y directions. Render these equations dimensionless.

b. Solve these equations to determine the dimensionless velocity along the channel. If you remember the relation between centerline and average velocities in this geometry you can use it.

c. Solve for the dimensionless velocity in the y direction.

d. Determine the streamfunction ψ, taking the value of ψ to be zero at the bottom of the channel.

Contours of constant ψ could be plotted to determine the trajectories that the fluid elements follow, and the trajectories of the settling red blood cells (integrating $dx/dt = u$ and $dy/dt = v - U_s$) would be used to determine their fate in the system, but don't do this here!

Drainage Flows

Subject Tags: Film Flow, Gravity-Driven Flow, Scaling Analysis, Differential Equations

Difficulty Level: Advanced

Estimated Time: 45 minutes

Summary: This problem considers the drainage of a thin viscous film from a vertical cylinder, approximated in the flat-earth limit to simplify the analysis. The task involves solving for the velocity profile, deriving the evolution equation for the film thickness, and analyzing its behavior through nondimensionalization and singularity formation.

Skills Tested:

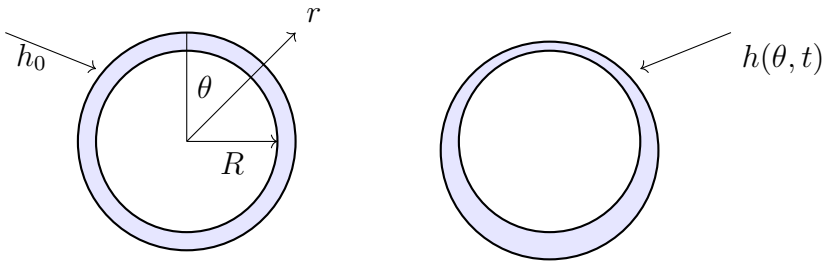
- Velocity Profile Derivation
- PDE Formulation
- Nondimensionalization
- Blow-up Time Calculation

Source: CBE 30355 Second Hour Exam, 2015 **File:** problem0153.tex

Drainage Flows

Earlier this semester your classmates demonstrated the drainage of a viscous film from an apple. Here we examine the simpler problem of drainage from a cylinder of radius R coated with a layer of fluid of density ρ and viscosity μ, and with initial thickness $h_0 \ll R$. We wish to determine the evolution of the thickness of the layer as a function of θ and t.

a. Redraw your coordinates for some value of θ in the flat earth limit (e.g., Cartesian coordinates!) and solve for the velocity profile in the draining film. Remember that g is now a function of θ! This should otherwise be identical to the falling film problem we solved in class. Note that $h \neq h_0$ as the film drainage evolves; rather, h_0 is just its initial condition!



b. Recognizing that the time derivative of the film thickness is just the radial velocity (or y velocity in your "flat earth" coordinate system) evaluated at $h(\theta, t)$, develop the equation that h must obey.

c. By scaling this equation, determine the characteristic drainage time t_c and render the problem dimensionless.

d. Solve for the time for the first drip to form (e.g., the solution blows up) at $\theta = \pi$ (the bottom of the cylinder).

I've provided the CE in cylindrical coordinates below, but it is not necessary for this problem because $h_0/R \ll 1$. It is much easier to do it in appropriate Cartesian coordinates, at least for the radial derivative!

$$\frac{1}{r}\frac{\partial}{\partial r}(rv_r) + \frac{1}{r}\frac{\partial v_\theta}{\partial \theta} + \frac{\partial v_z}{\partial z} = 0$$

Two-Dimensional Flows

This is the smallest category. The two problems listed involve the solution to the streamfunction in polar coordinates via separation of variables. In the class the classic Taylor wiper problem is solved, and for homework students look at a collapsing wedge. These two problems deal with drainage through a slit in a plane and the flow produced by drawing out an elastic sheet. Both cases yield different values for the radial dependence of the streamfunction.

Streamfunction in a Triangular Trough

Subject Tags: Streamfunction, Cylindrical Coordinates, Boundary Conditions, Creeping Flow

Difficulty: Advanced

Estimated Time: 60 minutes

This problem examines steady creeping flow in a two-dimensional triangular wedge filled with a very viscous fluid. An elastic sheet positioned at the centerline is stretched radially, inducing flow. The student is tasked with formulating and solving for the streamfunction that satisfies the boundary conditions. This requires the use of separation of variables and proper application of boundary conditions in cylindrical coordinates.

- Cylindrical coordinate system and velocity-streamfunction relationships
- Use of boundary conditions to construct the PDE and streamfunction form
- Reduction to an ordinary differential equation
- Analytical solution and determination of constants

Source: CBE 30355 Second Hour Exam, 2004 **File:** problem0116.tex

Creeping Flow in a Triangular Wedge

Consider the two-dimensional geometry depicted below. A very viscous fluid is contained in a triangular trough with opening angle $2\theta_0$. The walls are fixed (no-slip), and an elastic sheet lies along the centerline ($\theta = 0$).

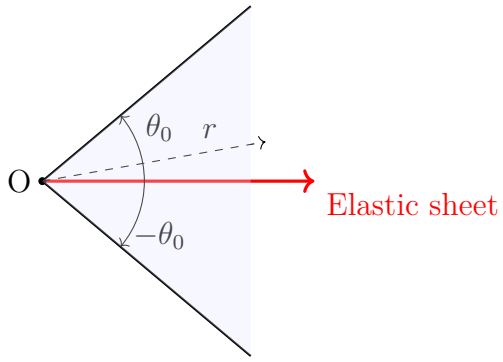


Figure 14: Geometry of creeping flow in a triangular wedge. The elastic sheet stretches radially at $\theta = 0$, inducing flow in the viscous fluid.

We wish to calculate the streamfunction $\psi(r, \theta)$ for the creeping flow induced by stretching the sheet radially, such that:

$$v_r(\theta = 0) = \lambda r$$

The velocity components in terms of the streamfunction in cylindrical coordinates are:

$$v_r = -\frac{1}{r}\frac{\partial \psi}{\partial \theta}, \qquad v_\theta = \frac{\partial \psi}{\partial r}$$

The Laplacian in cylindrical coordinates for scalar functions is:

$$\nabla^2 \psi = \frac{\partial^2 \psi}{\partial r^2} + \frac{1}{r}\frac{\partial \psi}{\partial r} + \frac{1}{r^2}\frac{\partial^2 \psi}{\partial \theta^2}$$

a. Set up the problem as completely as possible, including all relevant boundary conditions.

b. Use the boundary conditions to determine an appropriate functional form for the streamfunction $\psi(r, \theta)$. Derive the corresponding ordinary differential equation and associated boundary conditions for the angular part.

c. Solve the ODE and obtain equations for the unknown coefficients.

d. Explicitly determine the coefficients for the special case when $\theta_0 = \frac{\pi}{2}$.

Two Dimensional Drainage from a Slit

Subject Tags: Viscous Flow, Streamfunction, Boundary Conditions, ODEs, Cylindrical Coordinates
Difficulty Level: Moderate
Estimated Time: 45 minutes

A two-dimensional cylindrical geometry problem involving the flow of a viscous fluid draining through a slit is considered. The goal is to determine the streamfunction governing the unsteady viscous flow. The solution will involve boundary conditions, symmetry, and the appropriate application of the streamfunction formulation in cylindrical coordinates.

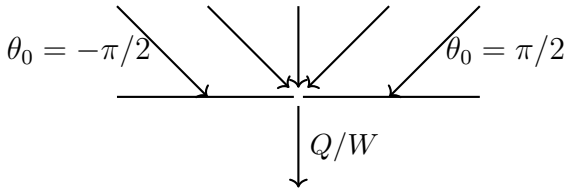
Skills Tested:

- PDE Formulation
- Symmetry Arguments
- Boundary Condition Identification
- Streamfunction Analysis

Source: CBE 30355 Second Hour Exam, 2006 **File:** problem0124.tex

Two Dimensional Drainage from a Slit

Consider the two-dimensional geometry depicted below. A very viscous fluid is draining out of the bottom of a tank through a narrow slit at a rate Q/W per unit extension into the paper.



a. Set up the problem as completely as possible, with all boundary conditions clearly determined.

b. Using the boundary conditions as a guide, determine the form of the streamfunction, and develop the corresponding ODE and boundary conditions. Hint: remember the relationship between the net flow rate through any curve connecting two streamlines, and the value of the streamfunction on the streamlines!

c. Solve for the streamfunction. Make maximum use of symmetry!

Useful Equations in Cylindrical Coordinates:

$$v_\theta = \frac{\partial \psi}{\partial r}, \quad v_r = -\frac{1}{r}\frac{\partial \psi}{\partial \theta}$$

$$\nabla^2 \psi = \frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial \psi}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 \psi}{\partial \theta^2}$$

Boundary Layer Flows

This section contains boundary layer flow problems for mass, momentum, and energy transport, both steady and transient. All are approached similarly, first via scaling to determine the important terms, then application of simple affine stretching to determine the similarity rule and variable. Some are amenable to complete solution, however in most cases they are left at the "set up" stage. These are easily adapted to homework problems where solutions to the boundary layer equations often require numerical solutions.

Thermal Boundary Layers in Shear Flow

Subject Tags: Heat Transfer; Boundary Layers; Similarity Solution; Non-dimensionalization

Difficulty: Advanced

Estimated Time: 45 minutes

Summary: This problem concerns the development of a thermal boundary layer in unidirectional shear flow subject to wall heating. Starting with the thermal energy equation, the problem leads you through non-dimensionalization, determination of thin boundary layer criteria, and derivation of a similarity rule for the resulting self-similar behavior of the temperature field near the wall.

Skills Tested:

- Non-dimensionalization of PDEs
- Boundary layer scaling arguments
- Understanding of steady-state energy transport in viscous flows
- Identification and application of similarity variables

Source: CBE 30355 Final Exam, 2001 **File:** problem0015.tex

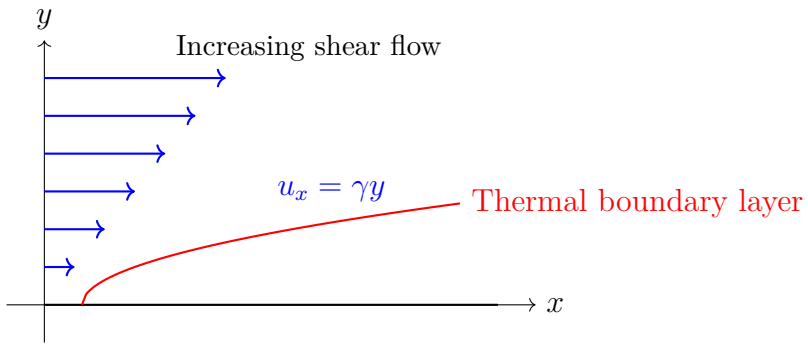
Thermal Boundary Layer Development

Consider the system depicted below. The fluid velocity is unidirectional simple shear flow in the x-direction. The fluid enters with a temperature of zero (e.g., we've already subtracted off some reference temperature) and is heated by a constant wall heat flux q_w. The thermal energy equation is:

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = \alpha \nabla^2 T$$

with boundary conditions:

$$\left.\frac{\partial T}{\partial y}\right|_{y=0,\, x<0} = 0, \quad -k\left.\frac{\partial T}{\partial y}\right|_{y=0,\, x\geq 0} = q_w, \quad T \to 0 \text{ as } y \to \infty$$



a. Render the equations dimensionless using a length scale L in the x-direction, and determine the conditions under which we can expect a thin boundary layer in the y-direction (e.g., how big does L have to be?). Assume steady-state, with no variation in the z-direction.

b. Show that the thermal boundary layer equations yield a self-similar solution, obtaining the similarity rule and similarity variable in canonical form. Using this, determine the temperature at the plate as a function of x to within some undetermined multiplicative constant. Note that you do not need to derive the transformed ODE or solve it.

Uni-directional Startup Flows

Subject Tags: unsteady flow; scaling analysis; similarity solution; Newtonian fluid

Difficulty Level: Advanced

Estimated Time: 45 minutes

Summary: This problem considers unsteady, one-dimensional startup flow induced by an impulsively stressed lower plate. Students are asked to derive the governing equations and boundary conditions, conduct scaling analysis to characterize long- and short-time behavior, and determine a similarity solution via coordinate stretching. This tests mastery of momentum conservation, dimensional analysis, and asymptotic limits in fluid dynamics.

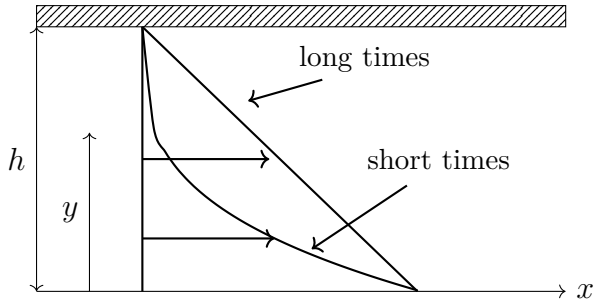
Skills Tested:

- Application of Navier-Stokes in unidirectional unsteady flows
- Scaling and dimensional analysis
- Identification of limiting behavior in PDEs
- Similarity transformations and self-similar solution derivation

Source: CBE 30355 Final Exam, 2002 **File:** problem0027.tex

Uni-directional Startup Flows

Consider the channel depicted below. For all times $t < 0$ the lower plate and fluid are at rest. At $t = 0$, the lower plate is accelerated such that the shear stress at the wall is a constant τ_w independent of time. The flow may be regarded to be unidirectional in the x-direction.



a. Determine the differential equation and boundary conditions governing this problem.

b. At very long times the flow reaches steady-state. Using t_c as this time scale and h as the length scale, how large does t_c have to be for the flow to become steady? (Hint: use scaling analysis and figure out what dimensionless group has to be large or small for the appropriate term to be thrown away). What is the velocity profile in this limit?

c. For very short times a boundary layer develops near the accelerating plate. Using t_c as this time scale, determine the characteristic thickness of this boundary layer (e.g., the new length scale) and the characteristic velocity of the lower plate.

d. Using the technique of coordinate stretching, show that this short-time limit admits a self-similar solution and obtain the similarity rule and variable in canonical form. Determine the velocity of the lower plate as a function of time to within an unknown $O(1)$ multiplicative constant (e.g., the solution of the similarity problem). Approximately when will this solution no longer be valid?

Unsteady Unidirectional Flow

Subject Tags: start-up flow, momentum equation, similarity solutions, asymptotic analysis, boundary layers

Difficulty Level: Advanced

Estimated Time: 60 minutes

Summary: This problem examines the transient development of velocity in a channel due to a suddenly applied pressure gradient. By analyzing both short and long time limits, students practice scaling, asymptotic analysis, and similarity methods. The final goal is to interpret wall shear stress over time in the context of the problem's physical transitions.

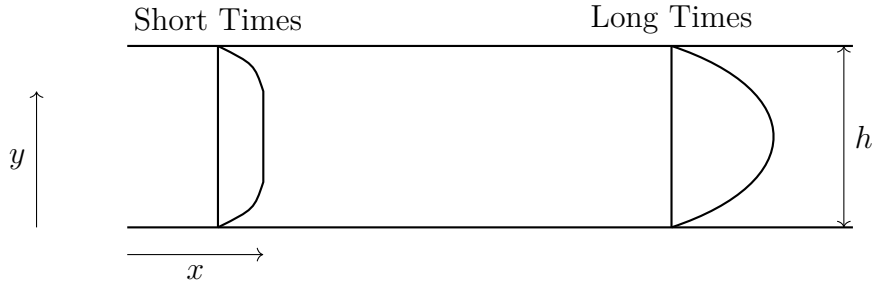
Skills Tested:

- Dimensional analysis and nondimensionalization
- Asymptotic approximations for long and short time behavior
- Solving PDEs in limiting cases
- Deriving and interpreting similarity variables
- Determining wall shear stress over time

Source: CBE 30355 Final Exam, 2011, solved **File:** problem0037.tex

Unsteady Start-up Flow in a Channel

Consider the uni-directional time-dependent start-up flow depicted below. Initially the fluid is at rest in a channel of width h. At time $t = 0$ we impose a pressure gradient $\frac{dp}{dx} = -G$ (a constant) in the x-direction. This causes the fluid to accelerate, eventually reaching some asymptotic velocity distribution. In this problem we use scaling to examine all the interesting parts of the problem.



a. The starting point: Write down the governing differential equation valid for all times and associated boundary conditions, crossing out all the terms which are zero. Do not include gravity.

b. Long times: Render the equations dimensionless, using some undetermined timescale t_c for time. Show that for very large t_c (e.g., if we wait a long time), inertia no longer matters, and obtain the appropriate scaling for the asymptotic velocity. About how long do we have to wait?

c. Long times: Solve for the asymptotic velocity distribution at long times, and determine the shear stress at the lower wall (hint: the velocity distribution should be – really – familiar by now!)

d. Short times: Rescale the velocity for very short times, and show that now the viscous term becomes unimportant. How short does t_c have to be? What is the velocity scaling for short times?

e. Short times: Solve for the now time-dependent velocity profile. (Hint: It's really simple, and you should remember what happens to the B.C.'s when you throw out the viscous term...)

f. Short times: Near each wall we will develop a boundary layer at short times. While there's one at both walls, we'll just look at the one near $y = 0$ for simplicity. Rescale the y coordinate and determine the boundary layer thickness as a function of t_c.

g. Short times: Using either the results of part (f), or via simple affine stretching, show

that the short time boundary layer problem admits a similarity solution. Give the similarity rule and the similarity variable in canonical form, and determine the time dependent wall shear stress to within some unknown multiplicative constant (e.g., the solution of the transformed differential equation that you don't have time to get).

h. All times: Using the results of (b) and (g), sketch up the wall shear stress as a function of time (rendered appropriately dimensionless) in the two asymptotic limits. Note that there is a gap between the two asymptotic solutions which must be bridged via another solution technique. It's actually fairly straightforward to get that one too, but we won't worry about that here... Do point out the domain of validity of the two solutions, however.

Scaling analysis of boundary layer flows

Subject Tags: mass transfer; electrochemical probe; shear stress; scaling analysis; boundary layer

Difficulty: Advanced

Estimated Time: 30 minutes

This problem examines the use of electrochemical probes to infer wall shear stress in turbulent flows. It involves a mass transfer analogy where the diffusion of a reactive species to an electrode provides a signal linked to the local shear rate. You will analyze the scaling laws, temporal resolution, and physical limitations of such probes.

Skills Tested:

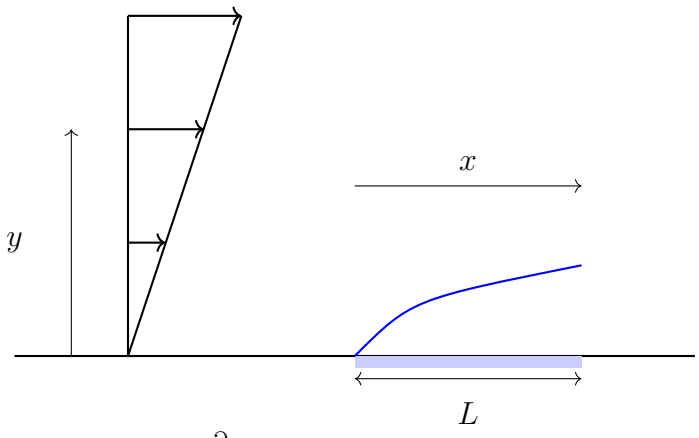
- Scaling analysis of partial differential equations
- Estimation of time scales in unsteady boundary layers
- Physical interpretation of transport analogies
- Application of boundary layer approximations

Source: CBE 30355 Final Exam, 2013, solved **File:** problem0047.tex

Scaling analysis of boundary layer flows

A popular method for measuring instantaneous wall shear stresses in turbulent flows is the use of an electrochemical probe. The fluid is doped with some reactant at concentration c_0 which decomposes electrochemically at an electrode flush with the wall. The reaction is essentially instantaneous, so the concentration of reactant at the electrode is zero, and the reaction rate is determined by the rate with which the species diffuses to the wall.

By measuring the current, we can determine the reaction rate and hence the integrated mass flux to the electrode. The mass flux is related to the shear stress because diffusive transport of mass is so slow relative to momentum transport that the mass transfer boundary layer essentially samples the linear shear flow right at the wall.



$$\frac{\partial c}{\partial t} + \vec{u} \cdot \vec{\nabla} c = D \nabla^2 c$$

$$\mu \left. \frac{\partial u_x}{\partial y} \right|_{y=0} = \tau_w \quad \Rightarrow \quad \vec{u} = \frac{\tau_w}{\mu} y \, \hat{e}_x$$

$$N = -D \left. \frac{\partial c}{\partial y} \right|_{y=0}, \quad I = W \int_0^L N(x) \, dx$$

a. Using scaling analysis, determine how the current per unit width I/W scales with the parameters of the problem (e.g., τ_w, D (the diffusion coefficient), μ, c_0, and L) at steady-state.

b. Estimate the time resolution of such a probe (e.g., how long would it take for the signal to approach steady state in the presence of shear transients). Choose $L = 1$ mm, $D = 10^{-6}$ cm^2/s, $\tau_w = 100$ dynes/cm^2, and use properties of water.

c. The current is only related to the shear stress if the boundary layer approximation holds. In general, we would like to make the probe as short as possible to get the best time resolution possible. What is the shortest length the probe can be?

Scaling Analysis of Boundary Layer Flows

Subject Tags: Mass Transfer, Stagnation Flow, Boundary Layers, Self-Similarity, Scaling Laws

Difficulty Level: Advanced

Estimated Time: 60 minutes

This problem explores mass transfer in the presence of a stagnation-point flow, where a planar jet impinges upon a surface. The flow and concentration fields must be analyzed using scaling arguments, with attention to the emergence of a boundary layer. The final step introduces self-similar variables and culminates in an ODE that characterizes the concentration profile.

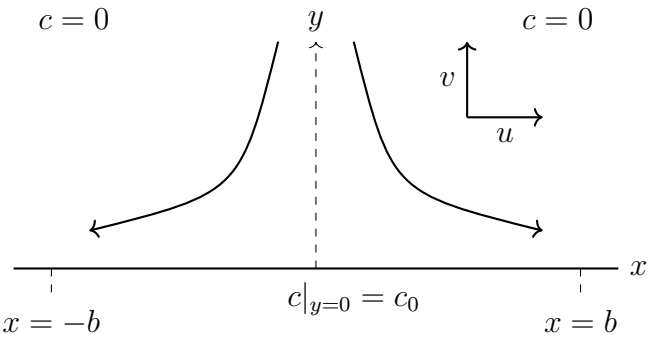
Skills Tested:

- Use of continuity to determine velocity components
- Scaling and boundary-layer approximations in convective-diffusion systems
- Estimation of total mass transfer rate from boundary layer theory
- Similarity transformation and derivation of ODEs for scalar transport

Source: CBE 30355 Final Exam, 2005 **File:** problem0122.tex

Mass Transfer from a Surface in Stagnation Flow

Consider the two-dimensional geometry depicted below. A planar jet impinges on a surface of width $2b$, with solute concentration $c = 0$ far from the surface and $c = c_0$ at the surface.



a. Near the plate, the x-component of velocity is given as $u = Axy$. Using the continuity equation and the boundary condition that $v = 0$ at $y = 0$, determine $v(x, y)$.

b. The concentration distribution $c(x, y)$ satisfies the convection-diffusion equation:

$$u\frac{\partial c}{\partial x} + v\frac{\partial c}{\partial y} = D\left(\frac{\partial^2 c}{\partial x^2} + \frac{\partial^2 c}{\partial y^2}\right)$$

Use scaling arguments to determine when the diffusion in the x-direction can be neglected and determine the characteristic thickness δ of the mass transfer boundary layer.

c. The surface mass flux is given by

$$N = -D\left.\frac{\partial c}{\partial y}\right|_{y=0}$$

Using your result from part (b), determine how the total mass transfer rate from the plate (width $2b$, depth W) scales with the problem parameters A, b, D, and c_0.

d. Using simple affine stretching, show that the problem admits a self-similar solution. Solve the resulting ODE for the concentration distribution. You may leave your final answer in terms of an integral.

Thermal Boundary Layers with Linearly Increasing Wall Flux

Subject Tags: Boundary Layers, Heat Transfer, Similarity Solutions, Dimensional Analysis
Difficulty Level: Advanced
Estimated Time: 45 minutes

Summary:
In this problem, a thermal boundary layer is analyzed for unidirectional shear flow above a flat plate. A wall heat flux increases linearly with distance down the plate. The problem asks for nondimensionalization, identification of conditions for a thin boundary layer, and the derivation of a similarity solution and plate temperature scaling.

- Dimensional Analysis
- Steady-State Heat Transfer
- Similarity Transformations
- Boundary Condition Implementation

Source: CBE 30355 Final Exam, 2006 **File:** problem0126.tex

Thermal Boundary Layers with Linearly Increasing Wall Flux

The thermal energy equation is:

$$\frac{\partial T}{\partial t} + \vec{u} \cdot \nabla T = \alpha \nabla^2 T$$

with boundary conditions:

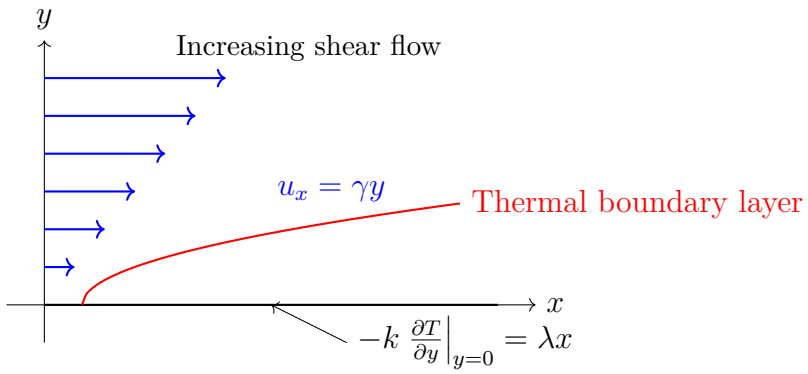
$$-k \left.\frac{\partial T}{\partial y}\right|_{y=0} = \lambda x, \quad \text{for } x \geq 0$$

$$-k \left.\frac{\partial T}{\partial y}\right|_{y=0} = 0, \quad \text{for } x < 0$$

$$T \to 0, \quad \text{as } y \to \infty$$

$$T_{inlet} = 0$$

$$\vec{u} = \gamma y \mathbf{e}_x$$



a. Render the governing equation and boundary conditions dimensionless using a length scale L in the x-direction, and determine the conditions under which we can expect a thin boundary layer in the y-direction. You may assume steady-state, with no variation in the z-direction.

b. Show that the thermal boundary layer equations yield a self-similar solution, obtaining the similarity rule and similarity variable in canonical form. Using this, determine the temperature at the plate as a function of x to within some undetermined multiplicative constant. Note that you do not have to get the transformed ODE or solve it to do this.

Heat Transfer from a Rotating Disk with Boundary Layer Flow

Subject Tags: Boundary Layers, Heat Transfer, Rotating Flow, Self-Similarity

Difficulty Level: Advanced

Estimated Time: 60 minutes

Summary: This problem investigates the heat transfer from a rotating disk in an unbounded fluid, using the velocity distribution obtained from boundary layer theory. The disk is heated with a constant surface heat flux, and the goal is to determine the surface temperature scaling, explore validity conditions, and demonstrate that the governing equations admit a self-similar solution.

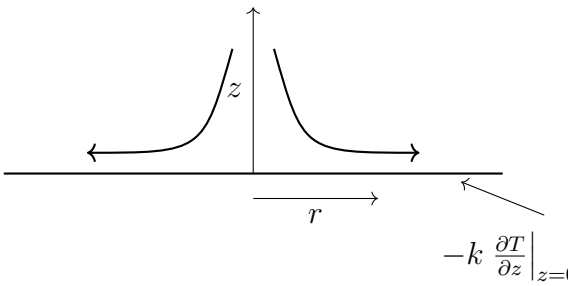
Skills Tested:

- Scaling of the energy equation
- Dimensional analysis for rotating systems
- Reduction of PDEs to ODEs via affine stretching
- Formulation of appropriate boundary conditions

Source: CBE 30355 Final Exam, 2010 **File:** problem0139.tex

Heat Transfer from a Rotating Disk with Boundary Layer Flow

Scaling/Boundary Layers: For homework you examined the boundary layer produced by a rotating disk. In this problem we use this velocity distribution to determine the temperature of a heated, rotating disk. Consider the disk of radius R depicted below, rotating with angular velocity Ω in an unbounded fluid.



The velocity distribution in the r- and z-directions near the disk is given by:

$$u_r = \lambda \left(\frac{\Omega^3}{\nu}\right)^{1/2} zr, \qquad u_z = -\lambda \left(\frac{\Omega^3}{\nu}\right)^{1/2} z^2$$

where λ is a dimensionless constant of order one obtained from a numerical solution to the momentum boundary layer equations.

The disk is heated such that at the surface there is a constant heat flux:

$$-k \left.\frac{\partial T}{\partial z}\right|_{z=0} = q_0 \quad \text{(constant)}$$

a. By scaling the energy equation and boundary conditions in the boundary layer limit, determine the characteristic temperature of the surface of the disk as a function of the parameters of the problem. You may take the temperature far away from the disk to have a reference value of zero (just like pressure in incompressible flow, the absolute value of the temperature doesn't matter as long as material properties are assumed constant). The energy equation for this problem is:

$$\frac{\partial T}{\partial t} + u_r \frac{\partial T}{\partial r} + u_z \frac{\partial T}{\partial z} = \alpha \left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right) + \frac{\partial^2 T}{\partial z^2}\right]$$

b. How fast do we have to rotate the disk for the scaling result obtained in part **a** to be valid?

c. Using affine stretching, show that this (now dimensionless) problem admits a self-similar solution. Obtain the corresponding ODE and boundary conditions.

d. Solve the equation to determine the temperature distribution on the disk. You may leave the final result in terms of an integral.

Heat Transfer from Plate in Impinging Jet Flow

Subject Tags: Scaling, Boundary Layer, Impinging Jet, Heat Transfer, PDE, Similarity Solution

Difficulty Level: Advanced

Estimated Time: 60 minutes

Summary: This problem analyzes heat transfer from a flat plate subjected to an impinging planar jet. Students must derive boundary layer scaling laws, determine the resulting heat loss, reduce the governing PDE to an ODE via similarity transformations, and (optionally) obtain a formal solution. This problem develops physical intuition for convective heat transfer under stagnation-point flow.

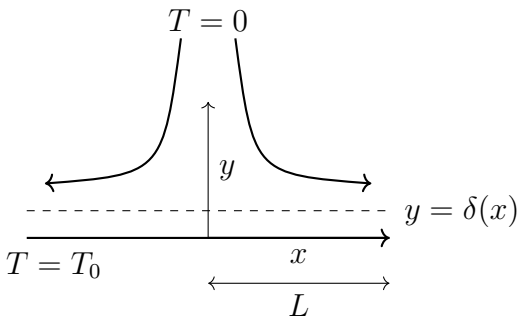
Skills Tested:

- Application of continuity and boundary layer approximations
- Scaling analysis for heat transfer
- Derivation of governing ODEs with boundary conditions
- Use of similarity solutions in thermal boundary layers

Source: CBE 30355 Final Exam, 2012 **File:** problem0143.tex

Impinging Planar Jet and Thermal Boundary Layer

Consider the heat transfer from a plate in the flow due to an impinging planar jet as depicted below. The fluid from the jet (usually produced by flow from a slit) has a temperature $T = 0$ (e.g., we subtract it off as a reference) and impinges on a plate maintained at a temperature T_0. Your mission is to determine the total heat transfer from the plate per unit extension into the paper, Q/W.



a. The velocity in the x-direction near the plate is given by $u = \lambda xy$. Applying no-slip and continuity, what is the velocity v in the y-direction?

b. The heat transfer equations are given below. Scale the equations in the boundary layer limit to determine the boundary layer thickness δ. Scale the expression for the integrated heat flux to determine the total rate of heat loss from the plate to within some unknown $\mathcal{O}(1)$ constant.

$$\rho C_p \left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} \right) = k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right)$$

$$\frac{Q}{W} = 2\int_0^L k \left.\frac{\partial T}{\partial y}\right|_{y=0} dx$$

c. Derive the second order ODE which governs the problem, together with boundary conditions, using appropriate similarity variables.

d. *Bonus:* Solve the differential equation and obtain the heat flux. You may leave the final answer in integral form.

Unsteady Heat Transfer in Solids

Subject Tags: Heat Transfer, Diffusion, Similarity Solutions, Scaling

Difficulty Level: Advanced

Estimated Time: 60 minutes

Summary: A solid of thickness h initially at zero temperature is exposed to a constant heat flux at the bottom surface while the top remains at zero temperature. The goal is to analyze the transient and long-term temperature distribution using governing equations, scaling, and similarity methods.

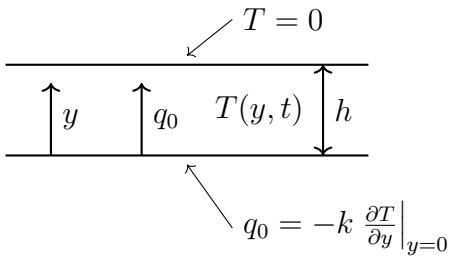
Skills Tested:

- PDE formulation for heat conduction
- Dimensional analysis and scaling
- Derivation of similarity variables
- Identification of asymptotic behaviors

Source: CBE 30355 Final Exam, 2018 **File:** problem0160.tex

Unsteady Heat Transfer in Solids

A material with thermal conductivity k (and thermal diffusivity $\alpha = k/\rho C_p$, the heat transfer analog to the momentum diffusivity ν) is sandwiched between two planes separated by a distance h. Initially the temperature of the entire system is at a reference value of zero, and the surface at $y = h$ is maintained at this temperature for all time. For $t > 0$ there is a constant heat flux q_0 at the lower wall ($y = 0$). Here we examine the resulting temperature distribution.



a. The starting point. Write down the governing differential equation and boundary conditions valid for all times. Remember that heat transfer in solids is mathematically identical to unidirectional flow problems!

b. Long-time behavior. Render the problem dimensionless in the long time limit and determine the asymptotic temperature distribution at long times.

c. Short-time scaling. For very short times the length scale isn't the gap width, but instead should be some boundary layer thickness δ. By scaling the equations in this limit, how do δ and the temperature scale with t?

d. Similarity solution. Using either the results of part c, or via simple affine stretching, show that the short time boundary layer problem admits a similarity solution. Give the similarity rule and the similarity variable in canonical form, and determine the time dependent temperature of the lower wall to within some unknown multiplicative constant (e.g., the solution of the transformed differential equation that you don't have time to get).

e. Breakdown of similarity. The solution in part d will break down after some time t. When will this occur and why?

Scaling of the Boundary Layer Equations for Wedge Flow

Subject Tags: Boundary Layer, Scaling Analysis, Falkner-Skan, Self-Similarity, Shear Stress

Difficulty Level: Advanced

Estimated Time: 45 minutes

Summary: This problem explores the boundary layer development for flow into a wedge, an intermediate case between flat plate and stagnation flow that leads to the Falkner-Skan formulation. The analysis includes scaling of the boundary layer equations, pressure gradient effects, and derivation of similarity variables for the streamfunction.

Skills Tested:

- Scaling of governing equations in the boundary layer limit
- Evaluation of pressure gradient effects
- Boundary layer thickness scaling
- Shear stress dependence
- Derivation of self-similar variables

Source: CBE 30355 Final Exam, 2019 **File:** problem0164.tex

Scaling of the Boundary Layer Equations for Wedge Flow

In class we solved for both flow past a flat plate at zero incidence (the Blasius problem) and stagnation flow such as you would get near the leading edge of a cylinder (flow perpendicular to a plate). These are two limits of the more general flow into a wedge which leads to the Falkner-Skan equation. Here we examine this problem.

Consider the flow into a wedge with internal angle of 120° as depicted below. The inner limit of the Euler flow solution as $y/L \to 0$ has a velocity given by:

$$u = \lambda x^{1/2}$$

where λ is a constant .

a. Determine the pressure gradient in the boundary layer for this flow. Does this promote or retard the growth of the boundary layer thickness?

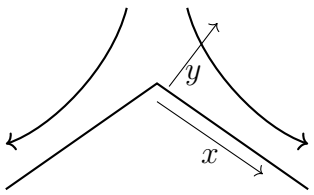
b. Scale the momentum and continuity equations in the boundary layer limit.

c. How does the boundary layer thickness vary with the parameters of the problem (and how does it depend on x, the distance from the leading edge)?

d. How does the shear stress at the surface ($y = 0$) depend on the parameters of the problem (and how does it depend on x)?

e. How far from the leading edge of the wedge do we have to be before the boundary layer solution becomes valid?

f. Using the continuity equation to define the streamfunction, determine the similarity rule for the streamfunction and the similarity variable (in canonical form!) which would yield the self-similar solution for this problem. Note that you are not being asked to derive the Falkner-Skan equation (e.g., the transformed ODE) itself.



Thermal and Momentum Boundary Layers

Subject Tags: Natural Convection, Boundary Layers, Dimensional Analysis, Heat Transfer, Scaling

Difficulty Level: Advanced

Estimated Time: 60 minutes

Summary:
This problem explores the cold drafts created by natural convection near a cold window. By scaling the Navier-Stokes and heat transport equations, we derive the characteristic size of the thermal boundary layer, the induced downward velocity, and the rate of heat loss per unit width of the window—all without solving the equations explicitly. The result is an elegant self-similar solution that characterizes the system in terms of basic physical parameters.

Skills Tested:

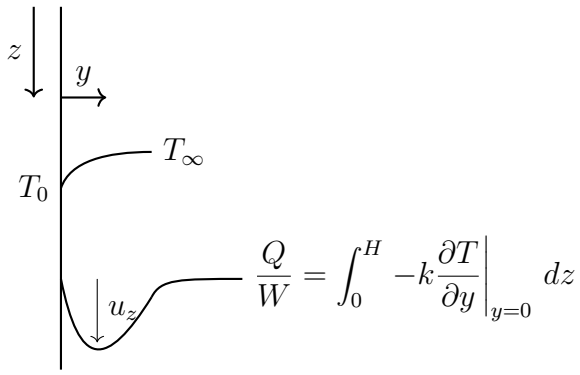
- Scaling of transport equations
- Use of boundary layer approximations
- Dimensional analysis
- Derivation of self-similar variable
- Estimation of convective heat loss

Source: CBE 30355 Final Exam, 2024 **File:** problem0172.tex

Thermal and Momentum Boundary Layers

It's cold outside and that means drafts off of cold windows! The cause is straightforward: the density of air increases as it gets colder according to the ideal gas law, and that causes a negative buoyant force that drags the fluid down. In this problem you will scale the transport equations to determine characteristic magnitude of the heat loss from a cold window from scaling alone.

Consider the window of height H depicted below. We take the temperature of the air in the room to be T_∞ and that of the window to be T_0, so the characteristic temperature difference is just $\Delta T = -(T_0 - T_\infty)$. The Navier-Stokes equation for momentum in the z-direction reduces to:



$$u_y \frac{\partial u_z}{\partial y} + u_z \frac{\partial u_z}{\partial z} = \nu \left(\frac{\partial^2 u_z}{\partial y^2} + \frac{\partial^2 u_z}{\partial z^2} \right) + g\beta(T - T_\infty)$$

$$\frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z} = 0$$

For natural convection problems such as this, the pressure gradient (a source of momentum!) is replaced with the buoyancy term $g\beta(T-T_\infty)$ where β is the coefficient of thermal expansion (just $1/T_\infty$ for an ideal gas, where T_∞ is in °K). The rate of heat loss per unit width Q/W is given by the integral of the temperature gradient at the surface, where k is the thermal conductivity of the air. Scaling these equations (and the continuity equation!) you can determine the boundary layer thickness, the characteristic magnitude of the downward draft velocity, and an estimate of the rate of heat loss from the window without ever solving the problem!

a. Render the z-momentum and continuity equations dimensionless in the boundary layer limit.

b. Determine how the boundary layer thickness and draft velocity depend on the parameters of the problem.

c. How large does H need to be for the boundary layer limit to be valid?

d. In the boundary layer limit the problem admits a self-similar solution (it's not a function of y^* and z^* independently, but rather a function of the similarity variable ξ). From your scaling (or via simple affine stretching – same thing!) determine ξ in canonical form.

e. By scaling the integral relation for the heat loss from the window, determine how Q/W depends on the parameters of the problem.

f. Now for some numbers. If ΔT is 20°C, the room temperature T_∞ is 293 K, and the kinematic viscosity of air is $0.15\,\mathrm{cm}^2/\mathrm{s}$, what are the values of U_c and δ from your scaling for a window of height $H = 1\,\mathrm{m}$?

g. If our window is square (e.g., $W = 1\,\mathrm{m}$ too), and the thermal conductivity of air k is $0.026\,\mathrm{W/m \cdot K}$, what is the scaling for the energy loss in watts? Note that this is just the scaling, of course – it turns out that if you solve the whole problem (including the energy equation, needed to get the temperature profile) your "unknown $O(1)$ constant" is about 0.5 for air (and is a weak function of the Prandtl number).

Dimensional Analysis and Scaling

In this section are gathered the very important problems dealing with dimensional analysis (the Buckingham Π theorem) as well as scaling of the equations to determine approximate magnitude of effects and scale up or scale down laws. The techniques used here are often very close (or identical) to those involved in boundary layer problems, and many of these problems could fit in either category.

Dimensional Analysis of Drop Breakup

Subject Tags: Dimensional Analysis, Drop Dynamics, Fluid Mechanics

Difficulty Level: Moderate

Estimated Time: 35 minutes

Summary: A drop of fluid released into another immiscible fluid forms a toroidal shape and eventually breaks up. Using dimensional analysis, this problem explores how physical parameters such as drop and fluid viscosities and densities influence the height of breakup.

Skills Tested:

- Application of Buckingham Π theorem
- Identifying and interpreting dimensionless groups
- Dimensional reasoning and scaling arguments

Source: CBE 30355 Midterm Exam, 2021, solved **File:** problem0003.tex

Drop Breakup via Dimensional Analysis

A drop of volume V, density ρ_d, and viscosity μ_d is released into a surrounding fluid of viscosity μ_f and density ρ_f, where $\rho_f < \rho_d$. The drop expands into a torus and eventually breaks up after falling a height H.

a. Using the Buckingham Π theorem, determine how many dimensionless groups this problem depends on, and construct a complete and independent set of such groups using the variables:

$$V,\ \rho_d,\ \rho_f,\ \mu_d,\ \mu_f,\ g,\ H$$

b. Since ρ_f and ρ_d are very close in value, strengthen your result by expressing buoyancy in terms of the density difference. That is, replace ρ_d and g with the product $\Delta\rho\, g$, where $\Delta\rho = \rho_d - \rho_f$. Reconstruct the relevant dimensionless groups using this simplification.

c. Empirical observation suggests that the breakup height H is nearly independent of drop volume and drop viscosity. Based on this, how should H vary with the fluid viscosity μ_f? Use your dimensionless groups to justify your conclusion.

Inspectional Analysis of Energy Transport in Flow Past a Heated Sphere

Subject Tags: Dimensional Analysis; Energy Transport; Navier-Stokes; Heat Transfer; Dimensionless Groups

Difficulty: Advanced

Estimated Time: 30 minutes

Summary: This problem explores the analogy between energy and momentum transport by analyzing steady flow of fluid past a heated sphere. Students are asked to nondimensionalize the governing Navier-Stokes and thermal energy equations, as well as boundary conditions, and express the total heat loss from the surface of the sphere in dimensionless form. Key focus is on identifying the relevant dimensionless groups that govern the thermal behavior.

Skills Tested:

- Governing Equation Nondimensionalization
- Application of Boundary Conditions in Dimensionless Form
- Heat Transfer and Energy Conservation in Fluid Flow
- Identification of Dimensionless Groups

Source: CBE 30355 2nd Hour Exam, 2001 **File:** problem0010.tex

Inspectional Analysis of Energy Transport in Flow Past a Heated Sphere

Energy transport is in many ways analogous to momentum transport. Consider flow past a heated sphere. Fluid (density ρ, viscosity μ, thermal conductivity k, and heat capacity per unit volume ρC_p) flows past a sphere of radius a whose surface temperature is maintained at T_1. The fluid far from the sphere is moving with velocity U and has temperature T_0. All material properties are assumed to be constant.

The velocity distribution is governed by the Navier-Stokes equations (neglecting gravity), and the temperature distribution by the thermal energy equation:

$$\rho C_p \left(\frac{\partial T}{\partial t} + \vec{u} \cdot \nabla T \right) = k \nabla^2 T$$

a. Render the equations and boundary conditions which govern the problem dimensionless (remember to render the Navier-Stokes equations dimensionless as well). Use the dimensionless temperature:

$$T^* = \frac{T - T_0}{T_1 - T_0}$$

b. If the heat loss from the sphere is given by:

$$Q = -k \int_{r=a} \nabla T \cdot \hat{n} \, dA$$

render this equation dimensionless as well. Upon what dimensionless groups does the dimensionless heat loss depend?

Inspectional Analysis of Oscillatory Channel Flow

Subject Tags: Unsteady Flow; Dimensional Analysis; Oscillatory Pressure Gradient; Viscous vs. Inertial Regimes

Difficulty Level: Moderate

Estimated Time: 35 minutes

Summary: This problem considers the unsteady, oscillatory flow of an incompressible fluid in a channel with a time-dependent pressure gradient. Students simplify the x-momentum equation, nondimensionalize it using a frequency-based time scale, and apply scaling arguments to find a characteristic velocity. The analysis reveals a key dimensionless group governing the dynamics in different frequency regimes.

Skills Tested:

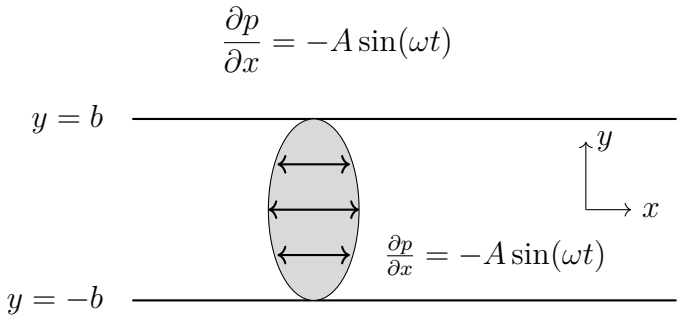
- Simplifying the Navier-Stokes equations for unidirectional flow
- Nondimensionalization using oscillatory scales
- Identifying and interpreting dominant balances
- Determining characteristic velocity scales
- Extracting meaningful dimensionless parameters

Source: CBE 30355 Second Hour Exam, 2016, solved **File:** problem0023.tex

https://diffusivepress.org/courseware/cbe30355/

Oscillatory Flow Between Parallel Plates

Consider the unsteady oscillatory flow in a channel of width $2b$, as shown below. The fluid is incompressible and the flow is unidirectional in the x-direction.



a. Write down the x-momentum equation for this unidirectional, incompressible flow and identify which terms vanish.

b. Nondimensionalize the momentum equation using

$$t^* = \omega t$$

as the dimensionless time variable and U_c as an unknown characteristic velocity scale. Use the pressure gradient amplitude A to normalize the equation.

c. Use inspectional analysis to determine the appropriate velocity scale U_c by balancing terms in the nondimensional momentum equation. Consider both high-frequency and low-frequency limits, and identify the key dimensionless group (a frequency-based Reynolds number) in each case.

Inspectional Analysis of Ship Drag Using a Scale Model

Subject Tags: dimensional analysis, modeling, Reynolds number, Navier-Stokes, similitude

Difficulty Level: Moderate

Estimated Time: 20 minutes

Summary: This problem concerns the use of a 1:100 scale model to predict the drag and required power of a full-size ship using inspectional analysis. Students will nondimensionalize the Navier-Stokes equations, identify key dimensionless groups (Reynolds number), and use similitude to infer the velocity and power requirements for the real system based on model data.

Skills Tested:

- Deriving dimensionless forms of governing equations
- Identifying relevant dimensionless groups
- Applying scaling arguments to similitude problems
- Power estimation via drag force and velocity

Source: CBE 30355 Secpmd Hour Exam, 2011, solved **File:** problem0033.tex

Inspectional Analysis of Ship Drag Using a Scale Model

It is proposed to model the drag on a large ship by towing a 1:100 scale model in a towing tank. You are assigned the task of doing the modeling.

a. Write down the Navier-Stokes equations in vector form and render them dimensionless using the inertial scaling. Identify the two dimensionless groups which appear.

b. If the target velocity of the large ship is U, what should the velocity U_m of the scale model be? (Hint: what dimensionless group are you trying to preserve?)

c. You want to size the power plant required to drive the ship. If the force measured on the towing rope for the model is F_m, what is the required size of the power plant? (As a practical matter, you would also need to account for losses, so the required size would be greater than this value – ignore this effect.)

Dimensional Analysis in Ship Design

Subject Tags: dimensional analysis; similitude; fluid mechanics; scale modeling

Difficulty Level: Moderate

Estimated Time: 15 minutes

Summary: A 1:100 scale model of a ship is used to study hull design in a towing tank. Given the velocity and towing force for the model, the problem explores how to estimate the full-scale velocity for dynamic similarity and the required power output for the full ship. Assumptions of similitude and propeller efficiency are used to derive practical engineering estimates.

Skills Tested:

- Dimensional analysis
- Reynolds number scaling
- Power estimation with efficiency

Source: CBE 30355 Second Hour Exam, 2017, solved **File:** problem0069.tex

https://diffusivepress.org/courseware/cbe30355/

Dimensional Analysis in Ship Design

A 1:100 scale model of a ship is constructed for study of hull design in a towing tank.

a. If the velocity of the model is U_m, what would be the corresponding velocity of the full scale ship?

b. If the force required to move the model at this velocity is measured as F_m, what would be the corresponding required power output of the engines of the full scale ship, assuming 75% efficiency of the propellers (a reasonable value)?

Dimensional Analysis in Ship Design

Subject Tags: Dimensional Analysis, Scaling Laws, Reynolds Number, Power Estimation

Difficulty: Moderate

Estimated Time: 30 minutes

Summary: This problem involves the use of dimensional analysis and scaling arguments to relate the motion and power requirements of a 1:64 scale model of a ship to its full-scale counterpart. The student is asked to compute the velocity and power required at full scale, and to analyze how the Reynolds number changes with scale, with an emphasis on why this matters in experimental modeling.

Skills Tested:

- Application of Froude scaling
- Power scaling from force and velocity
- Dimensional analysis of Reynolds number
- Physical interpretation of scaling effects

Dimensional Analysis in Ship Design

A 1:64 scale model of a ship is constructed for study of hull design in a towing tank.

a. If the velocity of the model is U_m, what would be the corresponding velocity of the full scale ship?

b. If the force required to move the model at this velocity is measured as F_m, what would be the corresponding required power output of the engines of the full scale ship, assuming 75% efficiency of the propellers (a reasonable value)?

c. What is the ratio of the Reynolds number of the model to that of the full scale ship? Why would this matter?

Dimensional Analysis of Drag Scaling for a Ship Model

Subject Tags: Dimensional Analysis, Fluid Mechanics, Drag, Scaling Laws

Difficulty: Moderate

Estimated Time: 25 minutes

Summary:
You are asked to estimate the drag force on a full-scale aircraft carrier using data from a 1-meter-long model. The goal is to determine relevant dimensionless groups, identify how to achieve approximate dynamic similarity in the model experiment, and use that information to relate the drag forces between model and full-scale system.

- Identification of dimensionless groups (e.g., Reynolds and Froude numbers)
- Application of scaling laws from dimensional analysis
- Approximate dynamic similarity considerations
- Drag force scaling

Source: CBE 30355 Second Hour Exam, 2006 **File:** problem0125.tex

Drag Scaling from Ship Model: Dimensional Analysis

You are assigned the task of estimating the drag on a 300 meter long aircraft carrier moving at 20 m/s (roughly 40 knots, or 40 nautical miles per hour). You choose to do this using a towing tank with a 1 meter long model.

a. The dimensionless drag depends on two key dimensionless parameters: what are they, and what is their approximate magnitude for the full-sized ship?

(Hint: If you can't remember them, you can easily get them by appropriate scaling of the Navier-Stokes equations!)

b. Using the concept of approximate (rather than strict) dynamic similarity, determine the velocity at which the model should be towed to best approximate the flow pattern of the full-sized carrier. What are the values of the important dimensionless parameters for the model?

c. What is the ratio of the drag on the carrier to that measured for the model?

Scaling of Freighter Design via Dimensional Analysis

Subject Tags: Dimensional Analysis, Marine Engineering, Power Scaling, Fuel Consumption, Transit Time

Difficulty Level: Moderate

Estimated Time: 30 minutes

This problem explores the use of dimensional analysis and scaling laws to estimate how important design and operational features of freighters—such as transit time, power plant size, and fuel costs—scale with overall ship volume, assuming geometric similarity and uniform operating conditions. It is an excellent test of physical modeling and estimation skills in a real-world engineering context.

Skills Tested:

- Dimensional analysis and similarity scaling
- Estimation of power requirements in fluid transport
- Reasoning about operational costs as functions of system scale
- Linking geometric and physical scaling

Source: CBE 30355 Second Hour Exam, 2014 **File:** problem0149.tex

Scaling of Freighter Design via Dimensional Analysis

Dimensional Analysis: You are responsible for determining the size of the power plant and fuel consumption for a fleet of freighters with similar design (hull shape), but different sizes. All of them are traveling as fast as they can practically go on the same route.

How do the transit time, size of the power plant, and total fuel cost vary with the freighter volume?

Scaling and Blowout Time for a Draining Straw

Subject Tags: Scaling analysis; Reynolds number; Unidirectional flow; Fluid mechanics

Difficulty: Advanced

Estimated Time: 60 minutes

Summary: A straw of length L and radius a is initially filled with fluid and then drained by applying a pressure difference. This problem investigates the unidirectional flow approximation and uses scaling analysis to determine the drainage time in both the high and low Reynolds number limits.

Skills Tested:

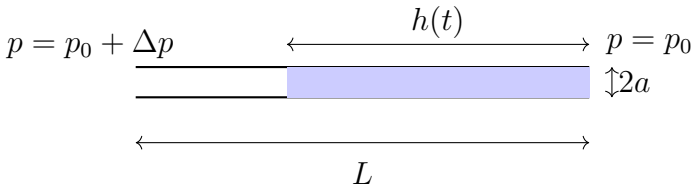
- Derivation of governing equations for unsteady unidirectional flows
- Dimensional analysis and identification of dominant terms
- Scaling arguments in high and low Reynolds number regimes
- Interpretation of boundary/initial conditions across flow regimes

Source: CBE 30355 Second Hour Exam, 2012, solved **File:** problem0045.tex

https://diffusivepress.org/courseware/cbe30355/

Draining of a Fluid-Filled Straw

Consider a horizontal straw of length L and radius a containing a liquid with viscosity μ and density ρ, as depicted below. The liquid is initially at rest. At time $t = 0$, a constant pressure difference Δp is applied to blow the liquid out. Let $h(t)$ be the length of straw filled with fluid at time t. We wish to determine the drainage time T_d for the column to fully evacuate in both the high and low Reynolds number limits.



a. Using the unidirectional flow approximation, write down the differential equation governing the fluid velocity (unsteady, in general), keeping only the non-zero terms. Also write down the relation between the velocity and the change in length $\dfrac{dh}{dt}$ of the fluid column. Include all relevant boundary and initial conditions.

b. Scale the equations for **high** Reynolds numbers and determine the (unknown) characteristic blowout time t_c in this limit. What boundary/initial conditions must be discarded in this limit, and what dimensionless group must be small?

c. Solve the problem to obtain the dimensionless, nonlinear, second-order ODE governing the evolution of the liquid slug length in this limit, along with the initial conditions. (The equation is trivial to solve numerically using MATLAB—the dimensionless blowout time turns out to be $\sqrt{\pi/2}$—but do not solve it here.)

d. Scale the equations for **low** Reynolds numbers and determine the new (unknown) characteristic blowout time t_c in this limit. What boundary/initial conditions must be discarded in this limit, and what dimensionless group must be small?

e. Solve for the actual drainage time in the low Reynolds number limit by solving the dimensionless equation from part (d) and obtain a numerical value for the drainage time.

The following equations may be helpful:

$$\frac{1}{r}\frac{\partial}{\partial r}(ru_r) + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial u_z}{\partial t} + u_r\frac{\partial u_z}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_z}{\partial \theta} + u_z\frac{\partial u_z}{\partial z}\right) = -\frac{\partial p}{\partial z} + \rho g_z + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 u_z}{\partial \theta^2} + \frac{\partial^2 u_z}{\partial z^2}\right]$$

Dimensional Analysis of Dripping Drop Radius

Subject Tags: Dimensional Analysis; Surface Tension; Gravity; Capillarity

Difficulty Level: Moderate

Estimated Time: 20 minutes

Summary: This problem explores the physical relationship between gravitational and surface tension forces for a drop detaching from a porous plate. By constructing a dimensional matrix, students determine the dimensionless groups and use them to estimate the drop radius and volume.

Skills Tested:

- Constructing a dimensional matrix
- Applying the Buckingham Pi theorem
- Estimating physical quantities using scaling
- Unit conversion and volume calculation

Source: CBE 30355 Second Hour Exam, 2016, solved **File:** problem0058.tex

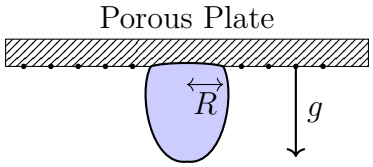
https://diffusivepress.org/courseware/cbe30355/

Dimensional Analysis of Dripping Drop Radius

Under some conditions (slow drips seeping from the underside of a porous plate) the radius R of a drop detaching from the surface is the result of a balance between the surface tension γ (units Force/Length) and gravitational forces.

a. Form the dimensional matrix of the parameters governing this problem, and determine the number of dimensionless groups involved.

b. Using the results of part (a), give a rough estimate of the radius R of a drop of water ($\gamma = 70\,\text{dynes/cm}$) dripping off of a plate and calculate its volume.



Scaling/Unidirectional Flows

Subject Tags: Dimensional Analysis, Unidirectional Flow, Scaling Limits, Viscous Drainage, Reynolds Number

Difficulty Level: Moderate

Estimated Time: 35 minutes (10 setup, 10 scaling, 10 solution, 5 interpretation)

Summary: A vertical straw filled with fluid drains due to gravity when the top is opened. Using both dimensional analysis and unidirectional flow approximations, we explore the time required to drain the straw under high and low Reynolds number limits. This problem requires formulation of governing equations, careful application of boundary conditions, and scaling arguments to derive time scales.

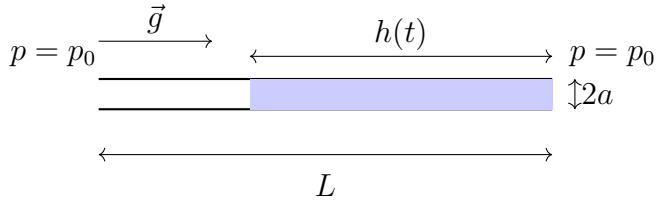
Skills Tested:

- Dimensional analysis and construction of π groups
- Simplification of governing equations using unidirectional flow assumptions
- Reynolds number-based scaling and limit behavior
- Determination of boundary and initial conditions

Source: CBE 30355 Final Exam, 2016, solved **File:** problem0061.tex

Scaling/Unidirectional Flows

Consider a vertical straw of length L and radius a containing a liquid with viscosity μ and density ρ as depicted below (the diagram is sideways so it fits on the page!). The liquid is initially at rest (we put our finger over the top of the straw). At time $t = 0$ we remove our finger and allow the liquid to drain out of the straw due to gravity. The length of the straw filled with fluid at any time t is given by $h(t)$. In this problem we wish to determine the time T_d required to empty the straw — i.e., how long it takes for h to reach zero.



a. A friend argues that T_d is a function of L, a, g, and fluid properties. Use dimensional analysis to determine a dimensionless expression for T_d in terms of these parameters (e.g., construct relevant Π groups).

b. Using the unidirectional flow approximation, write down the differential equation governing the fluid velocity (hint: unsteady, in general), keeping only the non-zero terms. Also write down the relation between the velocity and the change in length with time dh/dt of the column of fluid in the straw of length L. Write down all relevant boundary conditions and initial conditions.

c. Scale the equations for **HIGH** Reynolds numbers, and determine the (unknown) characteristic drainage time t_c in this limit. What boundary/initial condition(s) have to be thrown out in this limit, and what dimensionless group of parameters has to be small?

d. Solve for the drainage time T_d in this limit.

e. Scale the equations for **LOW** Reynolds numbers, and determine the characteristic drainage time t_c in this limit.

The following equations may be helpful:

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r u_r\right) + \frac{\partial u_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial u_z}{\partial t} + u_r\frac{\partial u_z}{\partial r} + u_z\frac{\partial u_z}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right) + \frac{\partial^2 u_z}{\partial z^2}\right] + \rho g$$

Dimensional Analysis

Subject Tags: dimensional analysis; porous media; capillary action; surface tension

Difficulty Level: Moderate

Estimated Time: 20 minutes

Summary: This problem involves a porous block absorbing a wetting fluid. Students are asked to apply dimensional analysis to determine the time scale for fluid absorption based on physical parameters like viscosity, surface tension, pore size, and block thickness. Additional information is used to validate and refine the dimensional prediction.

Skills Tested:

- Dimensional reasoning
- Scaling laws
- Use of empirical observations to refine models

Source: CBE 30355 Second Hour Exam, 2017, solved **File:** problem0070.tex

Dimensional Analysis

A porous block of material of thickness H with characteristic pore radius a is put on a puddle of liquid of viscosity μ and surface tension γ. If the liquid wets the pores, it will be absorbed into the block over some time T.

a. Using dimensional analysis, determine the independent dimensionless parameters that govern the time to fully wet the block.

b. It is observed that the wetting time scales as the square of the block thickness. Use this to strengthen your dimensional analysis.

c. If the pore radius is $10\,\mu$m, the block thickness is $5\,$cm, and the fluid is water (surface tension $\gamma = 70\,$dyn/cm), what is the order of magnitude of the time to wet the block?

Dimensional Analysis/Inertial Flows: Can Crushing by Vapor Condensation

Subject Tags: Dimensional Analysis, Inertial Flow, Vapor Condensation, Pressure, Time Estimation

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: This problem analyzes a classroom demonstration in which a can containing water vapor is crushed after being inverted and immersed in cold water. The rapid condensation of the vapor inside causes water to rush into the can through a hole. The volume of the can and the crushing pressure are provided, along with the hole areas for two different cans—one that crushed and one that didn't. Students are asked to estimate upper and lower bounds for the time it took for the vapor to condense completely.

Skills Tested:

- Dimensional reasoning and scale analysis
- Pressure-driven inertial flow analysis
- Time scale estimation from physical data

Source: CBE 30355 Second Hour Exam, 2018, solved **File:** problem0077.tex

Dimensional Analysis/Inertial Flows: Can Crushing by Vapor Condensation

In class, students demonstrated how a can filled with water vapor could be crushed by flipping it over and immersing it in cold water. The idea was that the vapor in the can would rapidly condense, and the water would have to rush into the hole on the end to fill the volume previously occupied by the vapor. The volume of the cans was about 500 mL, and the crushing pressure was about $\Delta p = 4 \times 10^4 \, \text{dynes/cm}^2$.

Whether or not a can crushed depended on the size of the hole at the top (bottom when inverted in the water). If a can with a hole area of 5 cm^2 crushed while one with an area of 10 cm^2 didn't crush, estimate an upper and lower bound for the time it took for the water vapor in the can to condense when it was immersed in cold water. Don't worry about factors of two.

Dimensional Analysis: Dip Coating with a Viscous Fluid

Subject Tags: Dimensional Analysis, Dip Coating, Surface Tension, Viscous Flow, Film Thickness

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: This problem involves applying dimensional analysis to a classic dip-coating process in which a solid object is withdrawn from a vat of viscous fluid. In part (a), the film thickness is determined in terms of relevant parameters including velocity, viscosity, surface tension, and buoyancy. In part (b), recent experimental observations suggest a maximum film thickness that is independent of surface tension. Students are asked to repeat the analysis under this new condition and estimate the scaling for maximum thickness.

Skills Tested:

- Dimensional analysis and dimensional matrix construction
- Identification of relevant physical variables and their units
- Extraction of characteristic scales and dimensionless groups

Source: CBE 30355 Second Hour Exam, 2018, solved **File:** problem0078.tex

Dimensional Analysis: Dip Coating with a Viscous Fluid

Dip coating (where an object is dipped in and then withdrawn from a vat of fluid, leaving a residue of fluid on the surface) is a commonly used coating technique.

a. For the case of strong surface tension γ (the most commonly encountered case), the thickness of the film h_f is a function of the vertical velocity U, the viscosity μ, the surface tension γ (units of force per length), and the buoyant force ρg (inertia is small, so it doesn't depend on inertia (density) separately). Using dimensional analysis, determine the number of independent dimensionless groups involved in the problem, and show what group(s) the problem depends on.

Hint: This is fairly simple if you use the dimensional matrix to construct characteristic lengths (groups of parameters with units of length) or reference velocities, and then the dimensionless groups are just ratios of these length or velocity scales.

b. An interesting paper (Snoeijer, et al., *Phys. Rev. Lett.*, 2008) demonstrated that under some conditions thicker films can also occur in dip coating. In this case, the thickness no longer depends on γ. Using this information, repeat the dimensional analysis of part (a) and determine this maximum thickness to within an $O(1)$ constant.

Scaling Analysis of Radial Flow Between Disk and Plane

Subject Tags: Lubrication theory, Radial flow, Force scaling, Low Reynolds number, Inertial limit

Difficulty Level: Advanced

Estimated Time: 50 minutes

Summary: This problem examines the fluid dynamics between a disk and a plane in radial flow, under conditions that span from low to high Reynolds number regimes. Starting with a mass balance and moving through both viscous and inertial scaling limits, the student is guided to determine how the net force on the disk depends on geometry, flow rate, and fluid properties. The final step involves predicting the equilibrium gap height where force balances.

Skills Tested:

- Application of mass conservation to axisymmetric flows
- Use of scaling arguments in viscous and inertial regimes
- Lubrication theory and pressure-driven flow analysis
- Dimensional reasoning to estimate physical quantities

Source: CBE 30355 Second Hour Exam, 2018, solved **File:** problem0080.tex

Scaling Theory: Low Reynolds Numbers to High!

Consider a disk of radius R located a distance H above a plane as depicted below. Fluid of viscosity μ and density ρ is pumped into the center of this geometry through a small orifice at a rate Q (volume/time), and flows outward radially. It is observed that at low flow rates the disk is blown away from the plane, while at high flow rates it is sucked downwards (as demonstrated in class). Here we analyze this problem in the limit $H/R \ll 1$.

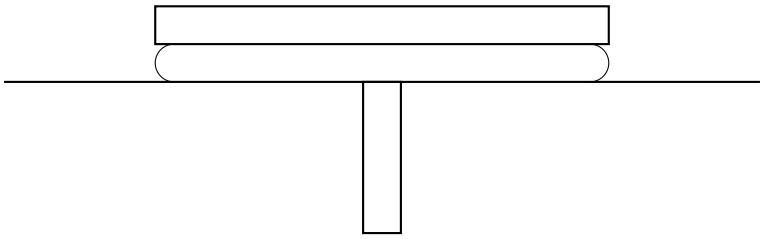
a. Using a mass balance, determine the average radial velocity (averaged over z) and how it depends on r.

b. Via scaling analysis of the equations of motion and a force balance on the plate, determine how the force scales with the parameters of the problem at low flow rates (lubrication limit).

c. Calculate the pressure distribution and force on the plate in this limit. You may leave the force in terms of a dimensionless integral of a known function, although the integral is pretty easy to evaluate to get the final number.

d. Determine via scaling analysis how the force scales with the parameters of the problem at high flow rates (inertial limit). Scaling only here – you would need to know the radius of the feed pipe (not provided) to get the numerical value.

e. If we allow the gap width to change, it will eventually seek a separation where the net force on the plate is zero. Estimate this gap width using scaling analysis.



The following equations may be helpful (note: the vast majority of these terms may be neglected!):

$$\frac{1}{r}\frac{\partial}{\partial r}(ru_r) + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial u_r}{\partial t} + u_r\frac{\partial u_r}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_r}{\partial \theta} - \frac{u_\theta^2}{r} + u_z\frac{\partial u_r}{\partial z}\right) = -\frac{\partial p}{\partial r} + \mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial (ru_r)}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 u_r}{\partial \theta^2} - \frac{2}{r^2}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial^2 u_r}{\partial z^2}\right] + \rho g_r$$

Scaling / Unidirectional Flows

Subject Tags: unsteady flow; scaling analysis; cylindrical coordinates; inertial limit; viscous limit; drainage time

Difficulty Level: Advanced

Estimated Time: 40 minutes

Summary: This problem explores the unsteady draining of a vertical liquid column in a straw under gravity. By applying the unidirectional flow assumption and using both inertial and viscous approximations, the student will determine characteristic timescales and assess the appropriate limit for water. The problem includes formulation, nondimensionalization, and limiting analysis.

Skills Tested:

- Differential equation setup under unidirectional flow assumptions
- Scaling and nondimensionalization techniques
- Analysis of inertial and viscous dominated regimes
- Order-of-magnitude estimation and physical interpretation

Source: CBE 30355 Second Hour Exam, 2019, solved **File:** problem0087.tex

https://diffusivepress.org/courseware/cbe30355/

Scaling / Unidirectional Flows

Consider a vertical straw of length L and radius a containing a liquid with viscosity μ and density ρ. The liquid is initially at rest (e.g., you have put your finger over the end so that no flow occurs). At time $t = 0$, you take your finger off the end and gravity causes the fluid to flow. The length of the straw filled with fluid at any time t is given by $h(t)$. In this problem, we wish to determine the time T_d required to empty the straw in both inertia- and viscous-dominated conditions—that is, how long it takes for h to reach zero.

a. Using the unidirectional flow approximation, write down the differential equation governing the fluid velocity (hint: unsteady, in general), keeping only the non-zero terms. Also write down the relation between the velocity and the change in length with time $\frac{dh}{dt}$ of the column of fluid in the straw of length L. Write down all relevant boundary conditions and initial conditions.

b. Render the equations and boundary conditions dimensionless, scaling the one term which you *know* has to be there to be $O(1)$. Don't forget the equation for h!

c. Scale the equations and boundary conditions for the case where inertia is the dominant physical mechanism, and determine the (unknown) characteristic drainage time t_c in this limit. What boundary/initial condition(s) have to be thrown out in this limit, and what dimensionless group of parameters has to be small?

d. Scale the equations for when viscous effects are dominant, and determine the new (unknown) characteristic drainage time t_c in this limit. What boundary/initial condition(s) have to be thrown out in this limit, and what dimensionless group of parameters has to be small?

e. If the straw is 20 cm long and the radius is 2 mm, which limit is more appropriate for water? Show your work!

The equations of motion in cylindrical coordinates are given below (you don't need all of these terms for this problem!):

$$\frac{1}{r}\frac{\partial}{\partial r}(rv_r) + \frac{1}{r}\frac{\partial v_\theta}{\partial \theta} + \frac{\partial v_z}{\partial z} = 0$$

$$\rho\left(\frac{\partial v_z}{\partial t} + v_r\frac{\partial v_z}{\partial r} + \frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta} + v_z\frac{\partial v_z}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2} + \frac{\partial^2 v_z}{\partial z^2}\right] + \rho g_z$$

$$\rho\left(\frac{\partial v_r}{\partial t} + v_r\frac{\partial v_r}{\partial r} + \frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta} + v_z\frac{\partial v_r}{\partial z} - \frac{v_\theta^2}{r}\right) = -\frac{\partial p}{\partial r} + \mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial (rv_r)}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 v_r}{\partial \theta^2} - \frac{2}{r^2}\frac{\partial v_r}{\partial \theta} + \frac{\partial^2 v_r}{\partial z^2}\right] + \rho g_r$$

Dimensional Analysis: Emulsion Drop Size

Subject Tags: dimensional analysis; emulsions; scaling; shear rate; surface tension

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: In this problem, we analyze a high-shear emulsification process using dimensional analysis. The final droplet size is assumed to depend on surface tension, viscosities, and shear rate. Students will construct a dimensional matrix, determine the number of independent dimensionless groups, and use a scaling estimate to predict the shear rate required to generate small water droplets in an oil phase.

Skills Tested:

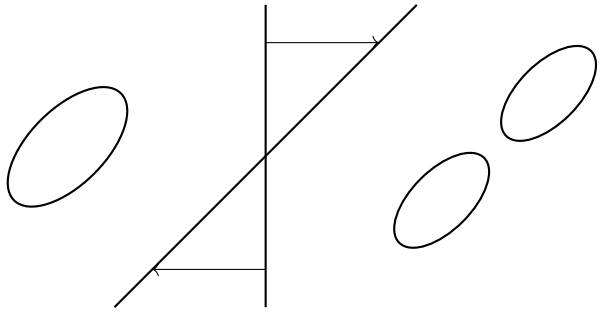
- Dimensional matrix formulation and analysis
- Identification of relevant variables and scaling
- Estimation using physical parameter values
- Interpretation of dominant physical forces in multiphase systems

Source: CBE 30355 Second Hour Exam, 2019, solved **File:** problem0088.tex

https://diffusivepress.org/courseware/cbe30355/

Dimensional Analysis: Emulsion Drop Size

A popular way to make water-in-oil emulsions (basically suspensions of really tiny drops) is by subjecting a mixture of oil and water to high shear rates. The shear stresses in the oil tear apart the drops of water until they reach the desired size. It is observed that the final drop radius a is a function of the surface tension Γ, the shear rate $\dot{\gamma}$, the oil viscosity μ_f, and the drop viscosity μ_d.



a. Form the dimensional matrix of the parameters governing this problem, and determine the number of dimensionless groups involved.

b. If $\Gamma = 7\,\text{dyne/cm}$, $\mu_f = 70\,\text{cP}$, and the drop phase is water, make an estimate of the oil phase shear rate required to achieve a drop radius of $50\,\mu\text{m}$.

Hint: It is the viscosity of the phase outside the drop which primarily controls the process!

Dimensional Analysis and Scaling in Mixing Tanks

Subject Tags: dimensional analysis; dynamic similarity; scale modeling; viscous flow; diffusion scaling

Difficulty Level: Moderate

Estimated Time: 30–40 minutes total (a: 10 min, b: 5 min, c: 10 min, d: 10–15 min)

Summary: This problem involves the use of dimensional analysis and strict dynamic similarity to design a scale model of a highly viscous industrial mixing tank. Key questions include the selection of appropriate model fluid properties, impeller speed, and the scaling of torque and diffusion processes.

Skills Tested:

- Dimensional analysis and similarity principles
- Reynolds number and rotational scaling
- Torque and power scaling with system geometry
- Diffusion coefficient scaling in viscous systems

Source: CBE 30355 Midterm Exam, sp2021, solved **File:** problem0092.tex

Dimensional Analysis and Scaling in Mixing Tanks

It is proposed to examine the mixing properties of the H. B. Fuller PB01 mixing tank using strict dynamic similarity. The tank is 1 m in radius and 3 m high, and the impeller operates at a rotation speed of 40 rpm. The fluids they are mixing are very viscous, with a viscosity of about 200 poise and a density of 1 g/cm^3. It is proposed to use a 1:9 scale model for the study.

a. What should be the properties of the simulant in the scale model?

b. What should be the rotation speed of the impeller in the scale model?

c. If we put an encoder to measure the torque on the impeller shaft of the model, how does that scale to the torque on the impeller of the full scale system?

d. In addition to mixing patterns, it is also desirable to look at interfacial transport (a key problem is getting all the water out of their viscous mixture). A colleague proposes to simulate this by looking at interfacial transport of a molecule in the model system. If the diffusion coefficient of water in the full scale system (with its viscous fluid) is 10^{-6} cm^2/s, what should be the diffusion coefficient of the water simulant in the model system to preserve strict dynamic similarity?

Hint: Think what ratios should be preserved!

Scaling Analysis for a Viscous Mixing Tank

Subject Tags: Dimensional Analysis, Reynolds Number, Froude Number, Dynamic Similarity, Scale Modeling

Difficulty Level: Advanced

Estimated Time: 45 minutes

Summary: This problem explores how to use dimensional analysis and strict dynamic similarity to design a scale model of a mixing tank for highly viscous fluids. The task involves estimating Reynolds and Froude numbers, adjusting fluid properties, determining operating conditions, and calculating power requirements under scaling.

Skills Tested:

- Evaluation of Reynolds and Froude numbers
- Application of similarity laws to fluid property scaling
- Derivation of rotational frequency in model systems
- Power scaling using dimensional arguments

Source: CBE 30355 Final Exam, sp2021 **File:** problem0168.tex

Scaling Analysis for a Viscous Mixing Tank

It is proposed to examine the mixing properties of the H. B. Fuller PB01 mixing tank using strict dynamic similarity. The tank is 1 m in radius and 3 m high, and the impeller operates at a rotation frequency of 40 rpm. The fluids they are mixing are very viscous, with a viscosity of about 200 poise and a density of 1 g/cm^3. It is proposed to use a 1:9 scale model for the study.

a. The mixing impeller pretty much spans the tank radius (to scrape the goo off the sides). Based on this, what is the approximate magnitude of the Reynolds number and the Froude number in this system? (Hint: don't forget you need to convert rpm to radians/s to get a good estimate...)

b. To achieve strict dynamic similarity the simulant fluid used in the scale model must be changed. What fluid property matters, and what should its new value be?

c. What should be the rotational frequency of the impeller in the scale model?

d. In designing the model system it is necessary to figure out what size motor to buy to drive the shaft. Because we are preserving strict dynamic similarity, we can use the power requirement of the full scale system to determine the motor needed for the model. What is the scale down ratio?

Hydrostatics and Submarine Implosion Scaling

Subject Tags: Hydrostatics, Implosion, Scaling, Continuity, Navier-Stokes
Difficulty Level: Advanced
Estimated Time: 45 minutes (15 concept, 15 calculation, 15 scaling)

Summary: This problem explores the hydrostatics and inertial scaling associated with submarine implosions, as exemplified by the Titan submersible tragedy. Students estimate the hydrostatic pressure at depth, compute the explosive-equivalent energy release of an implosion, derive radial velocity fields using the continuity equation, and apply dimensional analysis to the Navier-Stokes equations to determine a characteristic implosion time.

Skills Tested:

- Applying hydrostatic pressure calculations
- Estimating energy release using Archimedes' law
- Using continuity to obtain velocity fields
- Dimensional analysis of spherical Navier-Stokes equations

Source: CBE 30355 Midterm Exam, 2023, solved **File:** problem0098.tex

Hydrostatics and Submarine Implosion Scaling

Submarine implosions are very rare, however as I'm sure you are aware the Titan submersible was destroyed last summer exploring the wreckage of the Titanic. In this problem we are going to examine some aspects of this tragedy. The Titan submersible was a stubby cylinder with an equivalent spherical radius $R_0 = 1.1$ m. It imploded near the Titanic at a depth of 3800 m. The density of seawater is 1.028 g/cm^3.

a. What was the pressure on the hull in atmospheres at the depth of the Titanic (about where it imploded)? Recall 1 atm = 101 kPa and $g = 9.8$ m/s^2.

b. What was the energy released by the implosion? (Hint: remember Archimedes' law and how energy is related to force and distance.) Express your result in kilograms of high explosives, where 1 kg HE $= 4.2 \times 10^6$ J.

c. We are interested in estimating the time necessary for the implosion to occur once the hull is breached. Model the implosion as a spherical bubble of radius $R(t)$ collapsing under pressure. The fluid velocity at $r = R(t)$ is $u_r|_{r=R} = U(t)$. Using the continuity equation, determine the radial velocity $u_r(r,t)$ for all $r > R$ in terms of the (unknown) $U(t)$.

d. The change in radius with time is just $\dfrac{dR}{dt} = U(t)$. Initially, the pressure inside the bubble is 1 atm (negligible), while the pressure at large r is the value you got in part (a) from hydrostatics. Using this, scale the radial component of the Navier-Stokes equations in spherical coordinates to obtain the characteristic implosion time.

Interestingly, if you actually solve the problem (including the adiabatic compression of the bubble affecting the pressure inside as well – it oscillates if it is perfectly symmetric and you ignore all losses) you get a dimensionless implosion time of 0.917 – so the value you get from scaling alone matches the complete solution to within 10%!

The following equations are helpful – just remember that the vast majority of the terms are zero!

$$\frac{\partial \rho}{\partial t} + \frac{1}{r^2}\frac{\partial}{\partial r}(r^2 \rho u_r) + \frac{1}{r\sin\theta}\frac{\partial}{\partial \theta}(\sin\theta \rho u_\theta) + \frac{1}{r\sin\theta}\frac{\partial}{\partial \phi}(\rho u_\phi) = 0$$

$$\rho\left(\frac{\partial u_r}{\partial t} + u_r\frac{\partial u_r}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_r}{\partial \theta} + \frac{u_\phi}{r\sin\theta}\frac{\partial u_r}{\partial \phi} - \frac{u_\theta^2 + u_\phi^2}{r}\right) =$$
$$-\frac{\partial p}{\partial r} + \mu\left[\frac{1}{r^2}\frac{\partial^2}{\partial r^2}\left(r^2 u_r\right) + \frac{1}{r^2 sin\theta}\frac{\partial}{\partial \theta}\left(sin\theta\frac{\partial u_r}{\partial \theta}\right) + \frac{1}{r^2\sin^2\theta}\frac{\partial^2 u_r}{\partial \phi^2}\right] + \rho g_r$$

Kolmogorov Scale and Mixing Simulation Requirements

Subject Tags: Dimensional Analysis; Turbulence; Kolmogorov Scale; CFD Resolution

Difficulty Level: Moderate

Estimated Time: 30 minutes (15 for scaling, 15 for calculation)

Summary:
This problem introduces the concept of the Kolmogorov scale in turbulence and explores how it arises from dimensional analysis of power input, viscosity, and density. In the second part, the student uses given physical properties to estimate the numerical value of the scale in a glycerin-filled tank and determines the implications for the grid resolution required in a three-dimensional computational fluid dynamics (CFD) simulation.

Skills Tested:

- Dimensional analysis with physical parameters
- Application of turbulent scaling arguments
- Numerical estimation of physical length scales
- CFD grid point calculation based on resolution requirement

Source: CBE 30355 Midterm Exam, 2023, solved **File:** problem00100.tex

Kolmogorov Scale and Mixing Simulation Requirements

Mixing in a large vessel is usually characterized by the energy input per unit volume and time (e.g., the power going to the impeller divided by the tank volume). To fully simulate the behavior on a computer you have to solve the flow equations over lengths ranging from the tank size down to something called the Kolmogorov scale η, the length scale at which turbulent fluctuations die away.

a. If the Kolmogorov scale depends on this power input P/V, and the density ρ and viscosity ν of the fluid, how does it scale with the parameters of the problem?

b. You are trying to model a mixing vessel filled with glycerin ($\rho = 1.26\,\mathrm{g/cm^3}$, $\nu = 12\,\mathrm{cm^2/s}$). The tank has a volume of $1\,\mathrm{m^3}$ and the impeller input has a power of $1\,\mathrm{kW}$. What is the approximate numerical value of the Kolmogorov scale? (Note: we're not worried about the $\mathcal{O}(1)$ constant here, although the accepted value is 0.95.) Approximately how many grid points would be required for a 3-D numerical simulation?

High Reynolds Number Flow Down an Incline

Subject Tags: High Reynolds Number, Inclined Flow, Scaling Analysis, Turbulence, Eddy Viscosity

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: This problem examines the behavior of fluid flowing down an inclined plane at high Reynolds number. The inviscid limit is first considered to obtain an expression for the thinning of the film, followed by an estimate of the asymptotic layer thickness when turbulent effects become significant. Eddy viscosity is introduced as a turbulence model, and scaling arguments are used to derive the depth dependence.

Skills Tested:

- High Reynolds number flow assumptions
- Inviscid conservation analysis
- Application of eddy viscosity models
- Dimensional and scaling analysis

Source: CBE 30355 Second Hour Exam, 2003 **File:** problem0107.tex

High Reynolds Number Flow Down an Incline

For homework you analyzed the problem of a fluid flowing down an incline, such as depicted below, for low Re. In this problem we look at the opposite high Re limit.

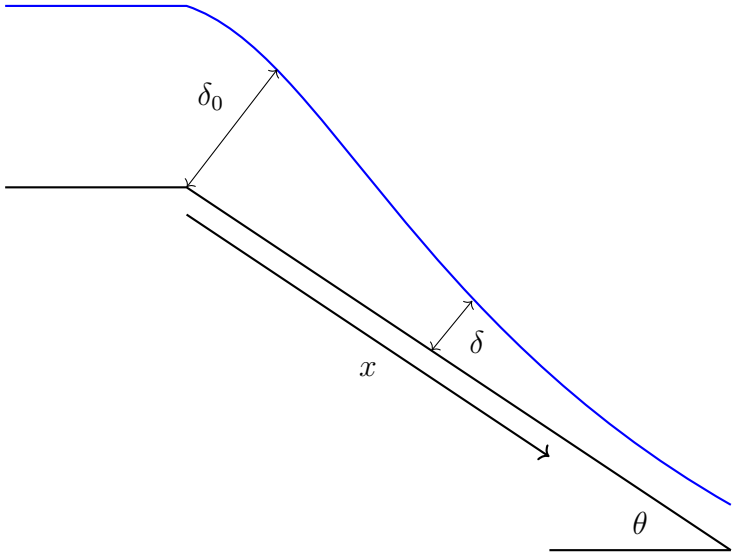


Figure 15: Flow Down an Incline

a. Ignoring all frictional losses, calculate the depth of the fluid as a function of distance down the incline. The initial depth is δ_0 and the flow rate per unit width is Q/W.

b. The fluid will keep getting thinner as it accelerates down the incline. Eventually, however, frictional losses due to turbulence will come into play, and the fluid will reach some asymptotic thickness. We want to estimate this thickness. The effect of turbulence can be modeled as an "eddy viscosity" analogous to the molecular viscosity in the equations of motion, only now the viscosity will be a function of both the depth of the layer and the velocity. For this problem the eddy viscosity can be estimated as:

$$\mu_t \approx \lambda \rho \delta^2 |\frac{u}{\delta}|$$

The parameter λ is just a dimensionless constant, with a value of around 0.1 or so. Replacing the molecular viscosity in the equation of motion with this term, use scaling analysis to estimate the asymptotic depth of the fluid layer. *(You don't have to solve it, just determine how it scales with the various physical parameters).*

Axial Force from Rotating Disk in Lubrication Limit

Subject Tags: Lubrication Theory, Scaling Analysis, Rotating Disk, Navier-Stokes, Dimensionless Form, Pressure Gradient

Difficulty Level: Advanced

Estimated Time: 45 minutes

This problem investigates the strong axial forces generated when a rotating disk approaches a nearby plane in the lubrication limit. You will calculate velocity profiles, identify governing equations and boundary conditions, and apply scaling and integral estimates to obtain both dimensional and quantitative expressions for force. One simplifying assumption is given to allow more accurate estimation.

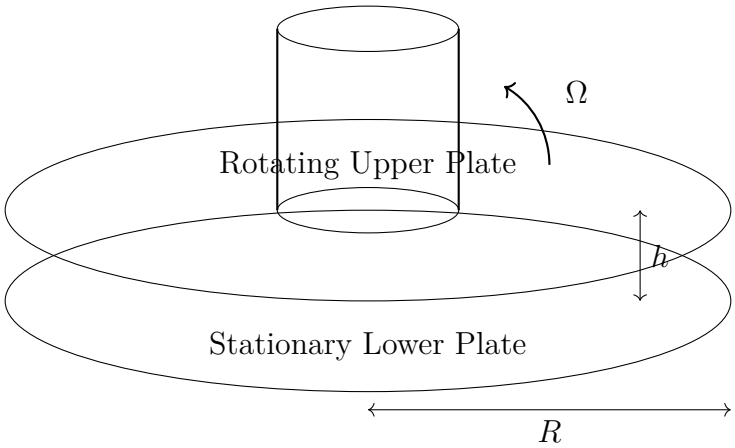
Skills Tested:

- Lubrication theory in cylindrical geometry
- Formulation of radial momentum equations
- Application of appropriate boundary conditions
- Scaling analysis and non-dimensionalization
- Estimation of axial forces using approximate methods

Source: CBE 30355 Final Exam, 2014 **File:** problem0111.tex

Axial Force from Rotating Disk in Lubrication Limit

Lubrication/Scaling: This semester we have examined the flow resulting from a rotating disk. If the disk is far from a surface there is no axial force on it (only torque). If it is very close to a plane, however, the axial force can be very large! Here we examine this in the lubrication limit $H/R \ll 1$ where H is the separation and R is the radius, as is depicted below.



a. Calculate the θ-velocity in this limit (this is simple, and you've done it before!). A classmate asserts that you also should require

$$\frac{H}{\sqrt{\nu/\Omega}} \ll 1.$$

Qualitatively describe what would happen to the velocity profile if this were violated. Be brief!

b. Write down the equation governing the radial velocity and pressure distributions in the lubrication limit. *(Hint: don't forget to include the one inertial term which actually drives the radial flow!)*

c. Write down all the boundary conditions which govern the radial velocity and pressure distributions. *(Hint: one is an integral!)*

d. Scale pressure and radial velocity, rendering the problem dimensionless. Use this to determine the scaling for the axial force on the disk.

e. To a good approximation (only off by about 10%, actually) the radial pressure gradient is just that due to the centrifugal force averaged across the gap. Using this simplifying approximation, determine a better estimate of the axial force on the disk than you can get with scaling alone (e.g., to within 10%).

The following equations may be helpful (particularly if you circle the dominant terms!):

$$\rho\left(\frac{\partial v_r}{\partial t}+v_r\frac{\partial v_r}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta}-\frac{v_\theta^2}{r}+v_z\frac{\partial v_r}{\partial z}\right)=-\frac{\partial p}{\partial r}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial(rv_r)}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_r}{\partial \theta^2}-\frac{2}{r^2}\frac{\partial v_\theta}{\partial \theta}+\frac{\partial^2 v_r}{\partial z^2}\right]+\rho g_r$$

$$\rho\left(\frac{\partial v_\theta}{\partial t}+v_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}+\frac{v_r v_\theta}{r}+v_z\frac{\partial v_\theta}{\partial z}\right)=-\frac{1}{r}\frac{\partial p}{\partial \theta}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial\left(rv_\theta\right)}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2}+\frac{2}{r^2}\frac{\partial v_r}{\partial \theta}+\frac{\partial^2 v_\theta}{\partial z^2}\right]+\rho g_\theta$$

$$\rho\left(\frac{\partial v_z}{\partial t}+v_r\frac{\partial v_z}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta}+v_z\frac{\partial v_z}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2}+\frac{\partial^2 v_z}{\partial z^2}\right]+\rho g_z$$

$$\frac{\partial \rho}{\partial t}+\frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r)+\frac{1}{r}\frac{\partial}{\partial \theta}(\rho v_\theta)+\frac{\partial}{\partial z}(\rho v_z)=0$$

Dimensional Analysis of Shockwave Expansion

Subject Tags: Dimensional Analysis, Scaling Laws, Shockwaves, Explosion Physics

Difficulty Level: Moderate

Estimated Time: 30 minutes

This classic example of dimensional analysis examines the time-dependent expansion of a shockwave resulting from an explosion. The problem explores how the radius of a shockwave depends on key physical parameters like energy, density, and pressure. Students will derive power-law relationships using dimensional arguments and estimate when simplified models break down due to additional physical effects like atmospheric pressure.

Skills Tested:

- Application of the Buckingham Pi Theorem
- Identification of relevant variables and fundamental dimensions
- Construction of dimensionless groups and scaling laws
- Interpretation of when physical simplifications cease to apply

Source: CBE 30355 Second Hour Exam, 2005 **File:** problem0121.tex

Shockwave Radius Scaling

Consider the radius of a shockwave generated by a large explosion, such as an atomic bomb.

a. The radius r of the shockwave in air depends only on the energy (or yield) of the explosion E, the time since detonation t, and the density of the undisturbed gas ρ_0. Using dimensional analysis, determine the radius of the shockwave as a function of time to within an unknown constant.

b. Eventually, the shockwave decelerates due to the ambient pressure P_0 of the surrounding atmosphere. Assuming that the critical radius r_c at which this occurs depends only on the explosion energy E and atmospheric pressure P_0, use dimensional analysis to estimate the corresponding breakdown time t_c when the simple answer from part (a) breaks down.

Surface Tension and Drainage of a Sealed Can

Subject Tags: Dimensional Analysis, Bernoulli Equation, Surface Tension, Unsteady Flow

Difficulty Level: Moderate

Estimated Time: 45 minutes

Summary: This problem explores the classic scenario of draining a can through a small hole, complicated by the presence of surface tension when the top is sealed. You will estimate the critical radius for drainage based on scaling arguments, derive the dynamics of drainage when the can is vented, and estimate when viscosity becomes important.

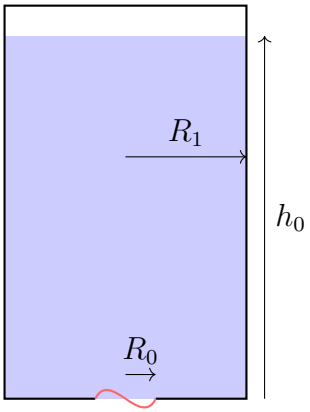
Skills Tested:

- Dimensional analysis with surface tension
- Pressure-driven outflow modeling using Bernoulli's equation
- Derivation and solution of ODEs for unsteady draining
- Estimating when viscous effects dominate

Source: CBE 30355 Final Exam, 2010 **File:** problem0140.tex

Drainage of a Can Through a Small Hole

Earlier this semester students demonstrated the interesting issue of the drainage of a can of liquid through a small hole of radius R_0. Here we examine this phenomenon:



a. If the top of the can is sealed (only the one hole in the bottom!), it is observed that the liquid won't come out at all if the radius of the hole is sufficiently small, and that the critical size doesn't depend on the height or radius of the can either. Adding a tiny amount of soap to the hole caused the liquid to start running out, showing that the drainage was held off by surface tension. Using these observations and dimensional analysis, estimate the critical radius R_0 for water ($\gamma = 70$ dyne/cm).

b. Physically, why shouldn't the critical size of the hole depend on the height of the fluid in the can? No more than a couple of sentences!

c. When a hole was punched into the top of the can, air rushed in and the liquid drained out very quickly. Neglecting viscosity, develop an equation to determine the velocity and flow rate as a function of fluid depth, and use it to calculate the drainage time for initial height h_0. Evaluate this drainage time for the properties of water, $h_0 =$ 10 cm, $R_1 = 5$ cm, $R_0 = 0.25$ cm. The discharge coefficient (actual velocity/velocity with no losses) has a value of about 0.6 for a sharp edged orifice.

d. A colleague argues that you are making a serious mistake by ignoring viscous effects. Estimate how large the viscosity of the fluid in the can would have to be to significantly affect the drainage time.

Dimensional Analysis: Hanford Pulse Jet Mixer Modeling

Subject Tags: Dimensional Analysis, Dynamic Similarity, Scale Modeling, Environmental Engineering

Difficulty Level: Moderate

Estimated Time: 35 minutes

Summary: This problem examines the design of a dynamically similar scale model to simulate a pulse jet mixing system used in nuclear waste treatment. Students are asked to identify appropriate scaling laws based on dynamic similarity and use these to determine model size, pressure, and time scale requirements.

Skills Tested:

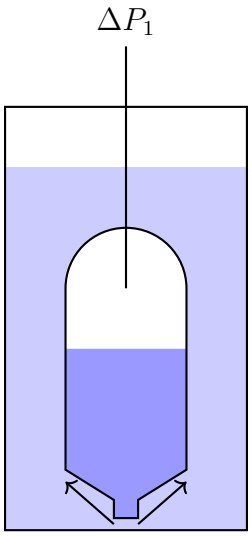
- Constructing and applying dimensionless groups
- Establishing dynamic similarity
- Scaling pressure, volume, and time for model systems
- Using fluid properties for similarity transformations

Source: CBE 30355 Second Hour Exam, 2013 **File:** problem0146.tex

Pulse Jet Mixer Scale Modeling Problem

In what is arguably the largest environmental remediation project in history, the Hanford waste treatment facility is being constructed to separate and vitrify the liquid radioactive waste left over from the fabrication of atomic weapons. A key component of the process is the mixing vessels to homogenize the liquid waste.

The mixing is done via pulse jet mixers, which are basically downward pointing bottles where liquid is first sucked in and then squirted out in an oscillatory manner. Your task is to develop the scaling laws necessary for designing a scale model test system for studying and evaluating the full scale system.



Full Scale System Parameters:

- Tank Volume = 500,000 liters
- Fluid Viscosity = 30 centipoise
- Fluid Density = 1.4 g/cm^3

You want to model this system using water as the working fluid.

a. What should the volume of the model tank be?

b. If the operating pressure of the full scale tank pulse jets is ΔP_1, what should the pressure of the jets in the model system be?

c. If the period of oscillation of the full scale pulse jets is T_1, what should the period of the model scale pulse jets be?

Hint: Don't forget that the free surface in the pulse jets matters, and use strict dynamic similarity!

Inertial Impaction and Face Shields

Subject Tags: Dimensional Analysis, Stokes Flow, Particle Transport, Newton's Second Law, Low Reynolds Number

Difficulty Level: Advanced

Estimated Time: 45 minutes

Summary: This problem models the effectiveness of a face shield in blocking exhaled droplets via inertial impaction. A droplet initially moves with airflow but, when the flow is abruptly stopped (simulating deflection by a shield), its inertia may carry it forward into the shield. The analysis involves dimensional analysis, low Reynolds number (Stokes) drag, and solving an ODE for particle motion.

Skills Tested:

- Dimensional analysis with multiple parameters
- Identification of relevant physical limits (Stokes flow)
- Application of Newton's Second Law to particle motion
- Use of Stokes drag in modeling viscous resistance
- Analytical solution of first-order ODEs

Source: CBE 30355 Final Exam, 2020 **File:** problem0166.tex

Inertial Impaction and Face Shields

Last summer I was asked to evaluate whether a face shield would serve as an adequate substitute for a face mask in protection against the emission or inhalation of droplets. Unfortunately, the answer was no. Because a shield is solid, the only way it can filter out droplets emitted during speech is via inertial impaction: because of their mass they do not exactly follow the streamlines of the air deflected by the shield, and instead (if they are big enough) impact on the surface. Here we examine this phenomenon by looking at a one-dimensional analog.

Consider a spherical droplet of radius a which initially moves with the velocity U_0 of the fluid (the exhaled air). At time $t = 0$ you stop the flow (corresponding to deflection by the face shield in the real problem), but due to its mass the droplet keeps going for a while. Inertial impaction occurs if the final displacement relative to the fluid is greater than the distance to the face shield.

a. The displacement of the droplet x depends on the density of the drop ρ, the density of the fluid (air) ρ_a, the viscosity of the air μ, the radius of the drop a, and the initial velocity U_0. Using dimensional analysis, determine the dimensionless groups the displacement depends on.

b. The result in part (a) isn't terribly useful, as there are too many groups! If the droplets are really small, however (and the ones we are most worried about are really really small!), their motion is governed by Stokes flow (low Re). In this case, the density of the air is negligible and the displacement x is proportional to U_0 (e.g., due to the linearity of the governing flow equations). Use this to strengthen your dimensional analysis down to a single group.

c. This still isn't good enough, as we need to actually solve the problem to get the "$O(1)$" constant. Using Newton's Second Law (e.g., force = mass $\times$ acceleration) and Stokes' Law (hint: remember $6\pi \ldots$) for the drag on a sphere, set up and solve for the time-dependent velocity and displacement of a droplet.

d. If the initial velocity is 100 cm/s, the viscosity of air is 1.81×10^{-4} g/cm $\cdot$ s, and the distance to the face shield is 5 cm, estimate the diameter of the droplets which could be captured by inertial impaction. The problem is that the droplets emitted in speech are smaller than this, and thus they get away...

Pipe Flows

In this section are gathered problems where the correlations for pipe losses are employed for systems which do not involve pumps. These are primarily gravity driven fountain and drain systems.

Hero's Fountain

Subject Tags: Fluid Mechanics; Bernoulli Equation; Head Loss; Friction Factor; Reynolds Number; Pipe Flow

Difficulty: Moderate

Estimated Time: 30 minutes

Summary: This problem involves analysis of Hero's Fountain, a classical device relying on air pressure and water columns to generate a jet. The first part considers an ideal scenario with no losses, while the second part incorporates frictional head losses in pipes and fittings using empirical correlations for laminar and turbulent flow regimes.

Skills Tested:

- Application of Bernoulli's equation
- Evaluation of head loss from pipe and fitting data
- Use of friction factor correlations for different flow regimes
- Dimensional analysis and fluid velocity estimation

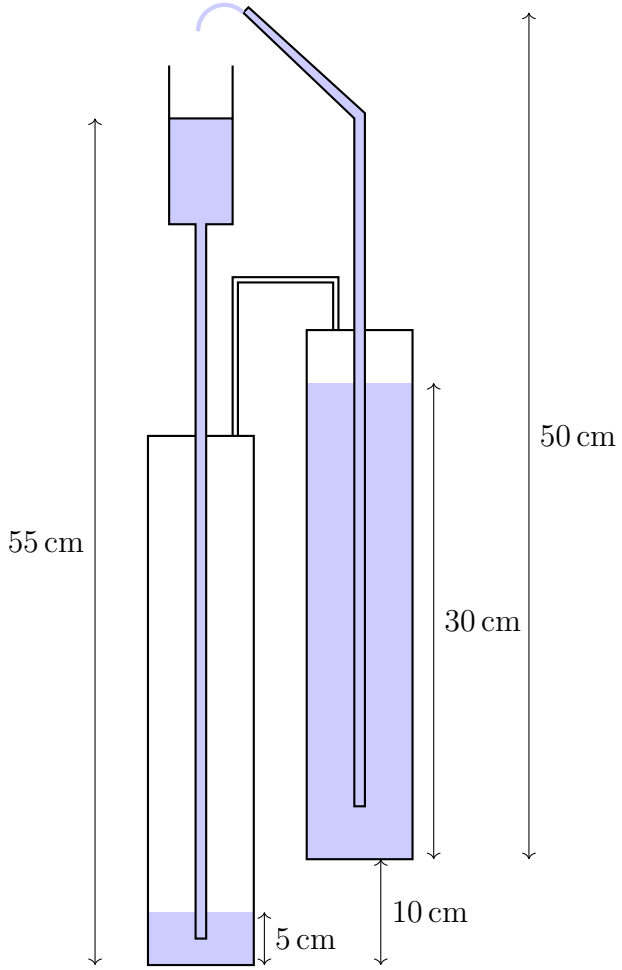
Source: CBE 30355 Final Exam, 2001 **File:** problem0014.tex

Hero's Fountain

In class we demonstrated Hero's Fountain, attributed to Hero of Alexandria a couple of millennia ago. In this problem we analyze its performance.

a. Neglecting all frictional losses, what is the exit velocity and flow rate of the fountain? All pipes are 1 cm ID smooth tubes.

b. Modify your answer by accounting for the head losses in the pipes and fittings. Correlations for friction factors in pipes and fittings are given below. You may take the total length of pipe to be 100 cm.



Head loss formula:

$$h_L = \frac{u^2}{2g}\left(\sum K + \frac{4 f_f L}{D}\right)$$

Friction factor correlations:

- Laminar flow (Re < 2100): $f_f = \frac{16}{\mathrm{Re}}$

- Blasius Formula ($3000 < \mathrm{Re} < 10^5$): $f_f \approx 0.0791\,\mathrm{Re}^{-1/4}$
- Prandtl Resistance Law ($\mathrm{Re} > 3000$):

$$\frac{1}{\sqrt{f_f}} = 4.0\log_{10}(\mathrm{Re}\sqrt{f_f}) - 0.40$$

Fitting K values:

- Sudden contraction: $K = 0.45$
- Sudden expansion: $K = 1.0$
- 45° elbow: $K = 0.35$

A Variant on Hero's Fountain

Subject Tags: Bernoulli equation; Head loss; Pipe flow; Nozzle losses; Fountain dynamics

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: This problem analyzes a variation on Hero's fountain, asking first for an idealized calculation of the fountain height, and then refining the result to include head losses due to pipe friction and fittings. A final part introduces a nozzle and requires an analytic derivation of a loss factor due to area contraction.

Skills Tested:

- Application of Bernoulli's equation
- Friction factor and head loss calculations
- Iterative estimation procedures
- Dimensional reasoning and simplification

Source: CBE 30355 Final Exam, 2011, solved **File:** problem0036.tex

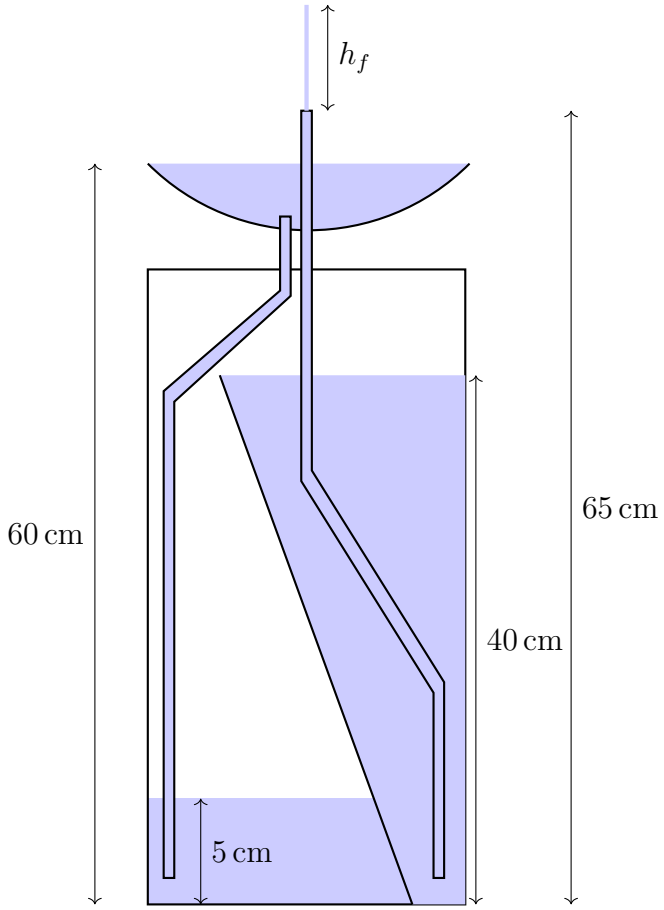
A Variant on Hero's Fountain

a. Neglecting all frictional losses, what is the height h_f of the fountain water jet?

b. Modify your answer by accounting for the head losses in the pipes and fittings. Correlations for friction factors in pipes and fittings are given below. You may take the total length of pipe to be 120 cm of 1 cm ID tubing. You will probably need to do a couple of iterations to get the friction factor right.

c. The result in part b leads to a rather unimpressive fountain. It is proposed to add a nozzle to the outlet to make it go higher. Nozzle friction factors are based on the ratio of areas of the inlet and outlet (β = small area/large area). While such factors are usually empirically measured, this one is easy to get analytically. *(Hint: think Bernoulli's equation and mass balances!)* Neglecting friction, derive the K factor of a nozzle as a function of β.

d. If we use a nozzle with $\beta = 0.5$, what is the new velocity (in the pipes) and fountain height? I just want an estimate: don't re-compute the friction factor from part b!



Useful Equations and Values:

$$h_L = \left(\sum K + \frac{4 f_f L}{D} \right) \frac{u^2}{2g}$$

$$f_f = \begin{cases} \frac{16}{\mathrm{Re}} & \text{for } \mathrm{Re} < 2100 \\ 0.0791\,\mathrm{Re}^{-1/4} & 3000 < \mathrm{Re} < 10^5 \\ \frac{1}{\sqrt{f_f}} = 4.0 \log_{10}(\mathrm{Re}\sqrt{f_f}) - 0.40 & \text{for } \mathrm{Re} > 3000 \end{cases}$$

Fitting K values:

- sudden contraction: 0.45
- sudden expansion: 1.0
- 45° elbow: 0.35

A Simple Fountain

Subject Tags: Bernoulli; Head Loss; Laminar Flow; Friction Factor; Dimensional Analysis

Difficulty Level: Moderate

Estimated Time: 25 minutes

Summary: This problem examines a historically inspired design for a fountain driven by hydrostatic pressure from an elevated reservoir. Students apply Bernoulli's equation to estimate the ideal fountain height and incorporate head losses due to friction and fittings to determine realistic performance. The problem concludes with a brief design analysis to suggest improvements.

Skills Tested:

- Application of Bernoulli's equation
- Estimation of head loss due to friction and fittings
- Use of empirical correlations for friction factor
- Basic engineering design recommendations

Source: CBE 30355 Final Exam, 2002 **File:** problem0026.tex

A Simple Fountain

The ancient Greeks and Romans (as well as many later civilizations, including the famous fountains of Versailles!) often operated fountains by the simple expedient of hydrostatic pressure produced by a reservoir fed by springs. In this case we will examine a very simple design for a small garden fountain driven by a reservoir in a creek.

Consider the pipe system and reservoir depicted below. All pipes are smooth, with a diameter of 2 cm and a total length of 20 m. The reservoir is located 1 m above the pool where we are installing the fountain.

a. Neglecting all losses, what is the maximum height the fountain could reach (think simple here!)?

b. Accounting for frictional losses, what height does it actually get to? (Hint: Not very impressive!)

c. What is the cheapest way we could get a fountain height much closer to the theoretical maximum? Why does this work?

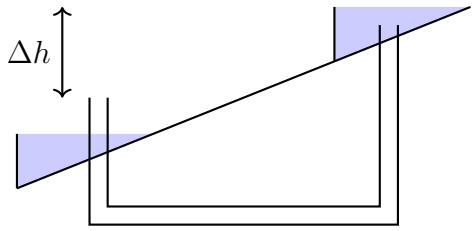


Figure 16: A Gravity Fed Fountain.

Useful equations and data:

$$\Delta h = 1\,\text{m} \quad h_L = \frac{u^2}{2g}\left(\sum K + \frac{4 f_f L}{D}\right)$$

$$f_f = \frac{16}{\text{Re}} \quad \text{(laminar, Re} < 2100) \qquad f_f \approx 0.0791\,\text{Re}^{-1/4} \quad (3000 < \text{Re} < 10^5)$$

$$\frac{1}{\sqrt{f_f}} = 4.0 \log_{10}(\text{Re}\, f_f) - 0.40 \quad ; \quad \text{Re} > 3000$$

Fitting K values:

- sudden contraction: 0.45
- sudden expansion: 1.0
- 90° elbow: 0.90

Siphon Flow and Spray Height

Subject Tags: siphon, Bernoulli, head loss, flow rate, friction factor

Difficulty Level: Moderate

Estimated Time: 40 minutes

Summary: This problem analyzes the flow through a siphon draining a tank. Part (a) assumes ideal flow to determine flow rate and spray height, while part (b) incorporates head losses and friction factor correlations, requiring iterative calculation for accurate results.

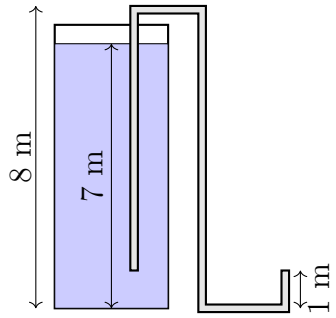
Skills Tested:

- Application of Bernoulli's equation
- Use of empirical correlations for head loss
- Dimensional analysis and unit conversion
- Iterative solution methods for friction factor estimation

Source: CBE 30355 Final Exam, 2013, solved **File:** problem0049.tex

Siphon Flow from a Tank

Here we determine the flow rate of a siphon used to drain the tank depicted below. Water fills a tank (open at the top to the atmosphere) to the height shown, and the siphon consists of 16 meters of 4 cm ID pipe (it is filled with water too!).



a. Neglecting all frictional losses, what is the flow rate and what height (relative to the siphon outlet) would the spray from the siphon outlet reach? Give your answer in liters/s for the flow rate and meters for the spray height.

b. Modify your answer to the flow rate and spray height by accounting for the head losses in the pipes and fittings. Correlations for friction factors in pipes and fittings are given below. You will probably need to do a couple of iterations to get the friction factor right — don't forget the domain of validity of the correlations! It helps to start the iterative calculation of the velocity with a reasonable guess for f_f since you know the other parameters.

$$h_L = \left(\sum K + 4 f_f \frac{L}{D}\right) \frac{u^2}{2g}$$

$$f_f = \begin{cases} \dfrac{16}{\mathrm{Re}}, & \mathrm{Re} < 2100 \\ \left[4.0 \log_{10}\left(\mathrm{Re}\sqrt{f_f}\right) - 0.40\right]^{-2}, & \mathrm{Re} > 3000 \end{cases}$$

Fitting K values:

- Sudden contraction: 0.45
- Sudden expansion: 1.0
- 90° elbow: 0.7

Siphon Drainage Design and Time Estimation

Subject Tags: Unsteady Flow, Bernoulli Equation, Pipe Flow, Friction Factor, Differential Equations

Difficulty: Advanced

Estimated Time: 45 minutes

Summary: This design problem involves determining the appropriate pipe diameter for a siphon system intended to drain a tank within a specified time. The challenge involves setting up and solving a differential equation for fluid height over time using a control volume approach and applying Bernoulli's equation with head losses. The solution includes iterative estimation of the friction factor for turbulent flow.

Skills Tested:

- Application of Bernoulli equation with losses
- Unsteady flow modeling
- Solution of first-order ODEs
- Friction factor estimation and iteration

Source: CBE 30355 Final Exam, 2003 **File:** problem0110.tex

Siphon Drainage Design and Time Estimation

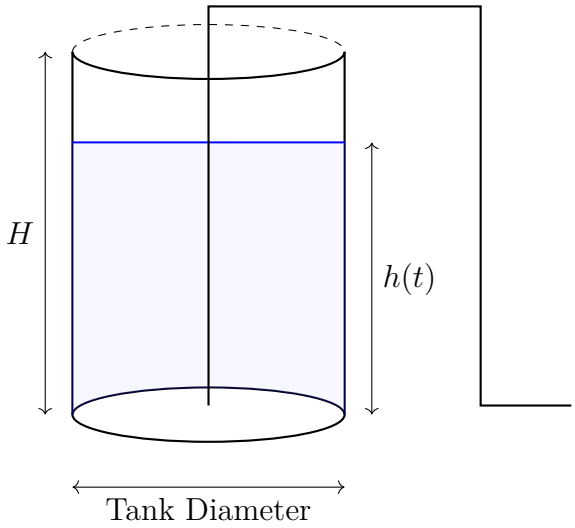
You are to design a siphon which will be used to drain a tank, as depicted below. We want to figure out what diameter pipe needs to be used for the siphon to drain the tank to the bottom in the desired time. Because the driving force (the height of fluid in the tank) varies with time, so does the velocity, the Reynolds number, and hence the friction factor. For purposes of estimation we will ignore the friction factor variation, and just use an approximate average value to be determined iteratively. Thus:

a. Set up the equation governing the change in the height of the fluid in the tank as a function of time.

b. Taking the friction factor to be constant, solve the differential equation for $h(t)$ and determine the time for the height to go from an initial value h_0 to zero. Use variables!

$$\text{Hint: recall that } \frac{1}{h^{1/2}}\frac{dh}{dt} = 2\frac{d(h^{1/2})}{dt}$$

c. For a tank 2 meters high and 2 meters in diameter, what diameter pipe should you use so that it drains in 1 hour? Take the length of the siphon to be 5 meters. You will have to solve this iteratively, however it converges pretty rapidly if you take an initial guess of a couple of inches.



You may find the following expressions useful:

$$h_L = \frac{u^2}{2g}\left(\sum K + 4f_f\frac{L}{D}\right)$$

$$\text{Laminar flow: } f_f = \frac{16}{\text{Re}}, \quad \text{Re} < 2100$$

$$\text{Blasius Equation: } f_f = 0.0791\,\text{Re}^{-0.25}, \quad 3000 < \text{Re} < 10^5$$

$$\text{Prandtl Equation: } \frac{1}{\sqrt{f_f}} = 4.0\log_{10}(\text{Re}\sqrt{f_f}) - 0.40, \quad \text{Re} > 3000$$

Fitting K values:

- Sudden contraction: $K = 0.45$
- Sudden expansion: $K = 1.0$
- 90° elbow: $K = 0.90$

Siphon Valve Design for Prescribed Flow

Subject Tags: Pipe Flow, Head Loss, Friction Factor, Minor Losses, Reynolds Number

Difficulty: Moderate

Estimated Time: 35 minutes

In this problem, you will analyze the flow of water through a siphoning pipe system and determine the appropriate resistance value K of a flow control valve to maintain a specified flow rate. The analysis must account for both major and minor losses, using expressions for friction factor in both laminar and turbulent regimes. You will also need to apply the extended Bernoulli equation and understand the use of loss coefficients for common pipe fittings.

- Application of the mechanical energy equation for pipe networks
- Use of loss coefficients for fittings and valves
- Estimation of Reynolds number and friction factor
- Combining major and minor losses to evaluate total head loss

Source: CBE 30355 Final Exam, 2004 **File:** problem0118.tex

Flow Regulation in a Siphoning Pipe Network

Consider the piping network depicted below. Water drains from the upper reservoir to the lower reservoir through a 5 cm inner diameter siphon as shown. The rate at which the water flows through the system is controlled by the valve located downstream.

Objective: Determine the required valve resistance coefficient K such that the volumetric flow rate through the system is maintained at 5 liters/sec. Friction factors for pipe segments and loss coefficients for the fittings are provided below.

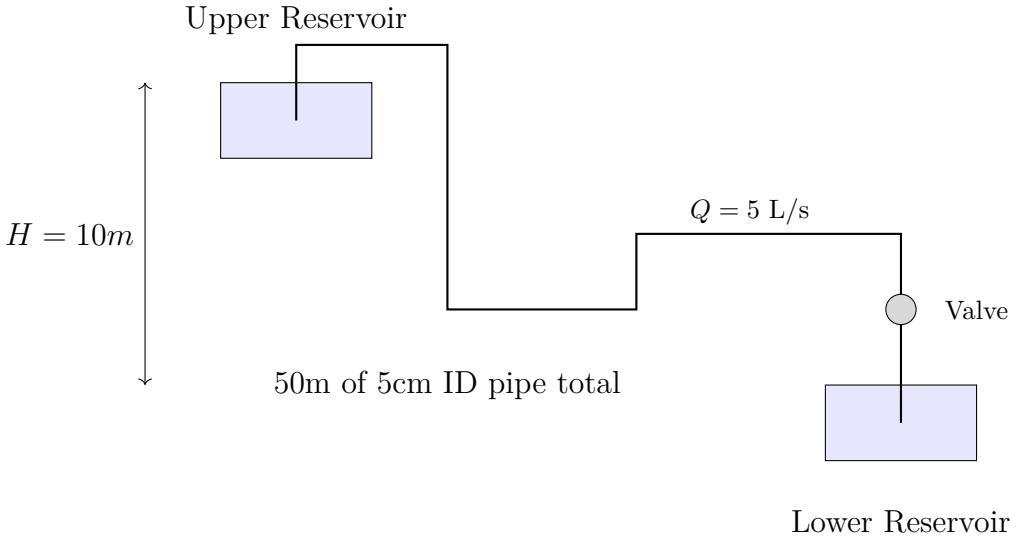
Useful relations:

$$h_L = \left(\sum K + \frac{4 f_f L}{D} \right) \frac{u^2}{2g}$$

$$f_f = \begin{cases} \dfrac{16}{Re} & Re < 2100 \text{ (laminar)} \\ \dfrac{0.0791}{Re^{1/4}} & 3000 < \mathrm{Re} < 10^5 \\ \dfrac{1}{\sqrt{f_f}} = 4.0 \log_{10}(Re\sqrt{f_f}) - 0.4 & \mathrm{Re} > 3000 \end{cases}$$

Fitting loss coefficients:

- Sudden contraction: $K = 0.45$
- Sudden expansion: $K = 1.0$
- 90° elbow: $K = 0.90$



Flow Measurement Using Orifice Meters

Tags: Orifice Flow, Bernoulli Equation, Minor Losses, Flow Rate Estimation, Manometer

Difficulty: Moderate

Estimated Time: 45 minutes

An orifice meter is a common device used to measure fluid flow rates. It typically consists of an orifice plate inserted in a pipe, and pressure taps located before and after the restriction. A U-tube manometer filled partially with air measures the pressure difference created by the constriction. This pressure drop can be related to the flow rate using Bernoulli's equation and minor loss coefficients. In this particular design, the orifice diameter is half the pipe diameter ($\beta = 0.5$). You will determine the relationship between the flow rate and manometer reading, and assess whether neglecting the pipe friction losses between taps is justified.

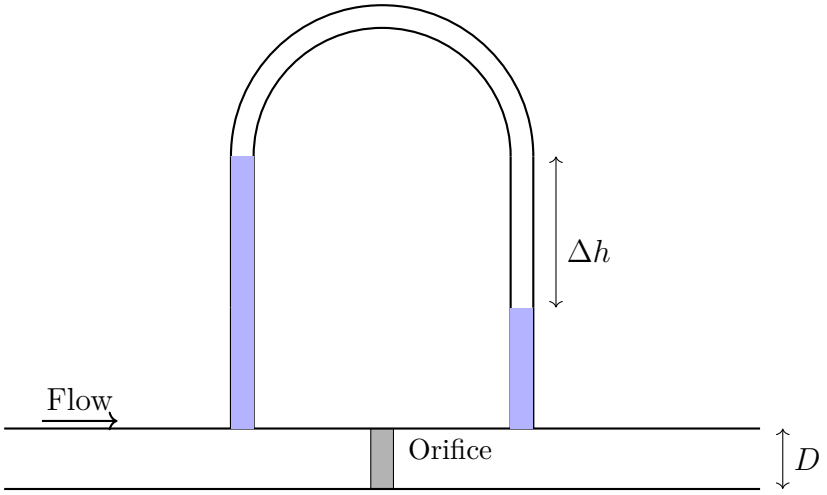
Skills Tested:

- Application of Bernoulli's equation
- Use of minor loss coefficients
- Interpretation of manometer readings
- Velocity and flow rate calculations
- Laminar and turbulent friction factor estimation

Source: CBE 30355 Final Exam, 2008 **File:** problem0133.tex

Orifice Meter Analysis

Consider the simple orifice meter depicted below. An inverted "U" tube, partially filled with air, is connected by a pair of "T" junctions separated by a short length of pipe (note that you would never normally use a pair of "T" junctions to connect to a manometer due to losses, but you could...). Between the "T"s is an orifice plate that constricts the flow and induces a pressure differential, which is related to the volumetric flow rate.



The orifice reduces the cross-sectional area by a factor of 2 (i.e., $\beta = 0.5$), and introduces a K value as given in the table of fittings below.

a. Neglecting the pipe friction factors (i.e., assuming the length of pipe between the "T"s is very short), determine the relationship between the measured Δh in the U-tube and the volumetric flow rate. The pipe diameter is D, and the fluid density is ρ. Assume the K values for the "T" fittings are distributed symmetrically upstream and downstream.

b. If $\Delta h = 20\,\text{cm}$, calculate the velocity in the pipe and the total volumetric flow rate. Use the properties of water, and take the pipe diameter to be $D = 2\,\text{cm}$.

c. A colleague argues that you need to include the frictional losses in the length of pipe between the "T" junctions. While this is in principle correct, it may not be a significant correction. Determine whether it matters in this case if the length of pipe between the "T"s is $L = 20\,\text{cm}$.

You may use the following empirical relations as needed:

$$h_L = \frac{u^2}{2g}\left(\sum K + \frac{4 f_f L}{D}\right)$$

$$\text{For Re} < 2100: \qquad f_f = \frac{16}{\text{Re}}$$

$$\text{For } 3000 < \text{Re} < 10^5: \qquad f_f \approx \frac{0.0791}{\text{Re}^{1/4}}$$

$$\text{For Re} > 3000: \qquad \frac{1}{\sqrt{f_f}} = 4.0 \log_{10}\left(\text{Re}\sqrt{f_f}\right) - 0.40$$

Fitting K-values:

- T (through side): $K = 1.5$
- T (straight through): $K = 0.4$
- Sharp orifice: $K = 2.7 \cdot (1 - \beta)\dfrac{(1-\beta)^2}{\beta^2}$

Coil Pump Design for Irrigation

Tags: Coil Pump, Head Loss, Friction Factor, Efficiency, Reynolds Number

Difficulty: Advanced

Estimated Time: 60 minutes

In this problem, you are designing a coil pump to supply water to an irrigation system. The pump must raise water 3 meters vertically and deliver it at a flow rate of 1 liter/s. You are provided with a drum of radius 20 cm and tubing of radius 2 cm. The analysis includes both ideal and non-ideal cases, accounting for head losses and determining the pump's mechanical efficiency.

Skills Tested:

- Head and Loss Analysis
- Dimensional Reasoning
- Empirical Correlations
- Pump Efficiency

Source: CBE 30355 Final Exam, 2009 **File:** problem0136.tex

Coil Pump Design for Irrigation

You are designing a coil pump to supply water to an irrigation system. The water needs to be pumped up the bank 3 meters (e.g., that is the required head output of the pump), and you need a water flow rate of 1 liter/s. You have available a rotating drum with a major radius of 20 cm and tubing (to coil around the drum) of 2 cm radius.

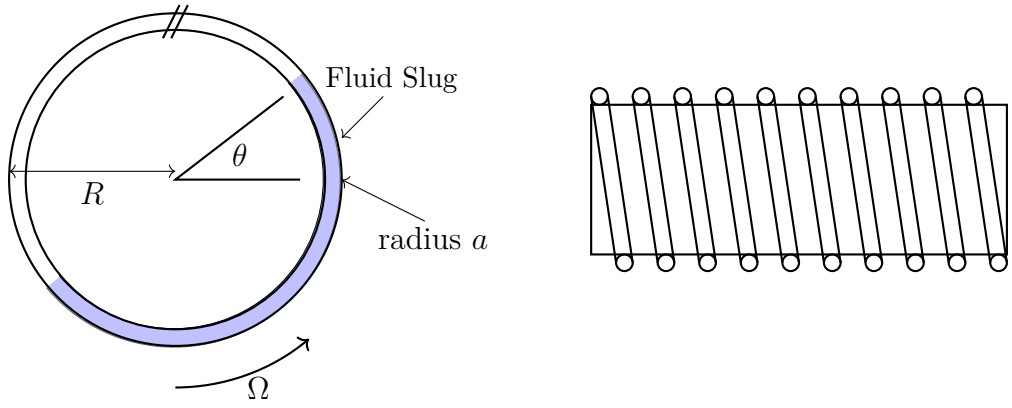


Figure 17: Schematic of a coil pump: tubing coiled around a rotating horizontal drum. Each loop is half-filled with fluid.

a. Neglecting losses, what is the minimum number N_0 of coils that are necessary to get the fluid up to the required height? Assume that the back pressure is divided uniformly among the coils, and that each of the coils is half-filled with water.

b. What is the angular velocity (e.g., radians/s) necessary to achieve the desired flow rate? What is the Reynolds number?

c. An empirical study suggests that the head losses in each coil are equivalent to a "K" factor of two long-radius 90° elbows (0.4 per elbow) plus the friction factor due to the pipe itself. Ignoring entrance and exit effects into the pump, determine the corrected minimum number of coils $N_{\min}$ (a bigger number than that in part a!).

d. Will entrance and exit effects have a significant effect on $N_{\min}$? Be quantitative!

e. In general it is a good idea to have more coils (N) than the minimum number! What is the mechanical efficiency of the pump as a function of the number of coils? You should be able to express this in terms of N, N_0, and $N_{\min}$ alone!
(Hint: this is most easily calculated by determining the mechanical work out and the total hydrodynamic losses in the system).

Use the following empirical formulas as needed:

$$h_L = \left(\sum K + 4\, f_f \frac{L}{D}\right) \frac{u^2}{2g}$$

$$f_f = \frac{16}{\mathrm{Re}} \quad \text{for } \mathrm{Re} < 2100$$

$$f_f \approx \frac{0.0791}{\mathrm{Re}^{1/4}} \quad \text{for } 3000 < \mathrm{Re} < 10^5$$

$$\frac{1}{\sqrt{f_f}} = 4.0 \log_{10}(\mathrm{Re}\sqrt{f_f}) - 0.40 \quad \mathrm{Re} > 3000$$

Overflow Drain Design for Tank

Subject Tags: Pipe Flow, Friction Factor, Minor Losses, Reynolds Number

Difficulty Level: Moderate

Estimated Time: 40 minutes

Summary: You are tasked with designing an overflow drain for a tank system that must handle a flow rate of 5 liters per second. Given the geometry and elevation of the tank and pipe, you will estimate the minimum required pipe diameter, using empirical loss correlations and appropriate assumptions about flow regime.

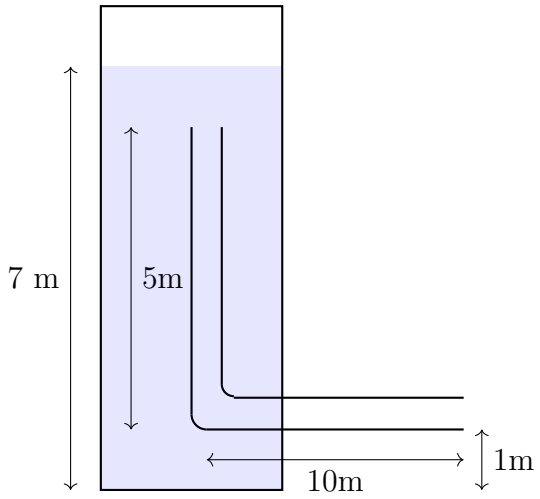
Skills Tested:

- Application of energy balance with head losses
- Use of empirical correlations for minor and major losses
- Calculation of Reynolds number and iterative friction factor determination
- Sizing of pipe diameter for a required flow rate

Source: CBE 30355 Final Exam, 2010 **File:** problem0141.tex

Tank Drainage Flow Rate and Pipe Sizing

You are designing an overflow drain for an open tank as depicted below. It is required that the pipe must handle a flow rate of 5 liters/s. What is the minimum diameter of the drain pipe?



You may find the following expressions useful:

$$h_L = \frac{u^2}{2g}\left(\sum K + 4f_f\frac{L}{D}\right)$$

$$f_f = \begin{cases} \dfrac{16}{\mathrm{Re}} & \text{for } \mathrm{Re} < 2100 \\ \dfrac{1}{\left(4.0\log_{10}(\mathrm{Re}f_f^{1/2}) - 0.40\right)^2} & \text{for } \mathrm{Re} > 3000 \\ \dfrac{0.0791}{\mathrm{Re}^{1/4}} & \text{for } 3000 < \mathrm{Re} < 10^5 \end{cases}$$

Fitting K values:

- Sudden contraction: 0.45
- Sudden expansion: 1.0
- 90° elbow: 0.90

High Re Flow Through Syringe

Subject Tags: High Reynolds Number, Syringe Flow, Frictional Losses, Dimensional Analysis

Difficulty Level: Advanced

Estimated Time: 45 minutes

Summary: This problem models the high-speed discharge of fluid from a syringe, emphasizing the effects of pressure-driven flow, needle geometry, and entrance and wall friction losses. Students must compute exit velocity and Reynolds number first without, and then with, frictional loss corrections. The problem concludes by considering how the method changes for a highly viscous fluid.

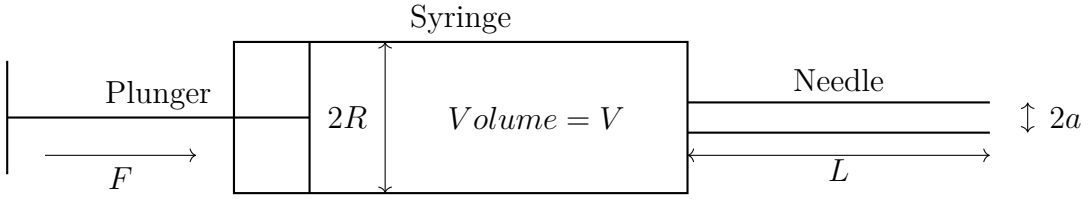
Skills Tested:

- Force Balance
- Bernoulli Equation with Head Loss
- Friction Factor Estimation
- Iterative Calculation

Source: CBE 30355 Final Exam, 2017 **File:** problem0157.tex

High Re Flow Through Syringe

Consider the syringe depicted below. Even though needle diameters are small, velocities (and hence Reynolds numbers) are often very high. It is desired to empty the syringe of volume V by applying a force F to the plunger of radius R. The fluid empties through a needle of radius a and length L.



a. Ignoring all frictional losses, calculate the velocity and Reynolds number in the needle and the emptying time. Take the following values:
$F = 10^5$ dynes, $R = 0.5$ cm, $V = 10$ mL, $a = 0.08$ cm, $L = 4$ cm, and use the properties of water.

b. Now improve your result for the velocity, Reynolds number, and emptying time by including frictional losses in the needle and a K value of 0.45 for the reentrant entrance to the needle (i.e., sudden rather than smooth contraction). The formulas below will be useful. Remember that iteration will be required, but it should converge very quickly!

$$h_L = \left(K + 4f_f \frac{L}{D}\right) \frac{u^2}{2g}$$

$$f_f = \begin{cases} \dfrac{16}{\mathrm{Re}} & \mathrm{Re} < 2100 \\ \dfrac{0.0791}{\mathrm{Re}^{0.25}} & 3000 < \mathrm{Re} < 10^5 \\ \frac{1}{\sqrt{f_f}} = 4.0 \log_{10}(\mathrm{Re}\sqrt{f_f}) - 0.40 & 3000 < \mathrm{Re} \end{cases}$$

c. How would you change your approach to this problem if you were trying to squirt out glycerin ($\rho = 1.26$ g/cm^3, $\mu = 14$ poise) instead of water? Be brief, but quantitatively justify your approach.

Pumps and Pipes

In this section are two types of problems: simple problems reading a pump curve to determine operating parameters, efficiency, and NPSHR under a variety of conditions, and more complicated piping flow problems which also involve analysis of the pump curve. Most of the pump curves are for Sykes pumps. The collection begins with the more involved pumps and pipes problems, followed by the straight pump curve reading problems. In addition to these pump curve problems, on eight of the exams the pump curve reading problems were paired with short answer questions. These are quite similar, but are not included here. If desired they may be found in the companion document `ashortanswercollection.pdf` or (to be compiled in TeXShop) `ashortanswercollection.tex`, both in this .zip file directory.

Pumping System and Head Loss

Subject Tags: Head Loss, Pump Curves, Energy Balances, Friction Factor, Pipe Network

Difficulty Level: Moderate

Estimated Time: 45 minutes (10 analysis, 15 calculations, 10 curve reading, 10 economics)

Summary: Water is pumped from a pond into an elevated tank through a piping network. This problem involves calculating the required pump head, matching the system curve with a commercial pump curve, and determining power consumption, efficiency, and potential energy savings from using a larger pipe diameter. Assumptions for fittings and flow resistance must be used carefully to evaluate the energy loss and cost tradeoffs.

Skills Tested:

- Application of Bernoulli's equation with head losses
- Use of friction factor correlations and fitting losses
- Reading and interpreting pump performance curves
- Energy efficiency and operating cost analysis

Source: CBE 30355 Final Exam, 2016, solved **File:** problem0062.tex

Pumping System and Head Loss

It is desired to pump water from a pond into a storage tank using the piping network below (pipe lengths not to scale). The requirements are a flow of 20 liters/s, and the proposed pipe diameter is 7.8 cm ID (e.g., a 3" schedule 40 PVC pipe). The total pipe length is 50 meters.

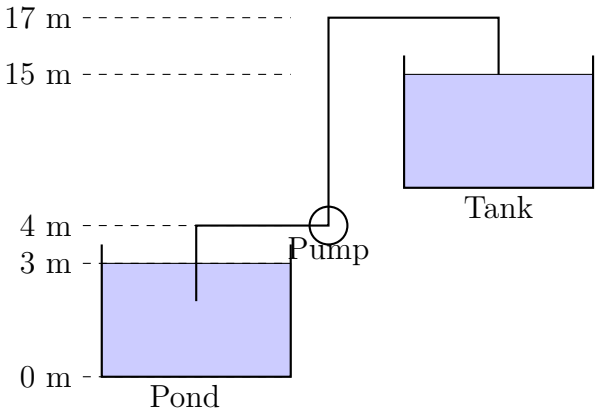
a. Calculate the total head required of the pump. Correlations for friction factors in pipes and fittings are given on the next page.

b. It is proposed to use the pump CP-80i (pump curve is on the next page). Identify the operating point on the curve, and use this to determine the required RPM and power requirement.

c. What is the efficiency of the pump under these conditions?

d. How close to the surface of the pond (vertical elevation) is it necessary locate the pump? (You can ignore f_f for this part because the pipe is short, but include other losses!)

e. It is suggested that operating costs could be significantly reduced if a larger pipe diameter is employed. If the cost of electricity is \$0.10/kWh (current Indiana rates), what would be the maximum monthly savings that going to a (much) larger pipe could possibly result in? Note that you wouldn't actually have to make it all that much bigger — the losses go roughly as D^{-4}!



Note: Drawing not to scale!

Head loss expression:

$$h_L = \left(\sum K + \frac{4 f_f L}{D} \right) \frac{u^2}{2g}$$

Friction factor correlations:

$$\text{For Re} < 2100: \quad f_f = \frac{16}{\text{Re}}$$

$$\text{For } 3000 < \text{Re} < 10^5: \quad f_f = \frac{0.0791}{\text{Re}^{1/4}}$$

$$\text{For Re} > 3000: \quad \frac{1}{\sqrt{f_f}} = 4.0 \log_{10}(\text{Re}\sqrt{f_f}) - 0.40$$

Fitting K values:

- Sudden contraction: $K = 0.45$
- Sudden expansion: $K = 1.0$
- 90° elbow: $K = 0.75$

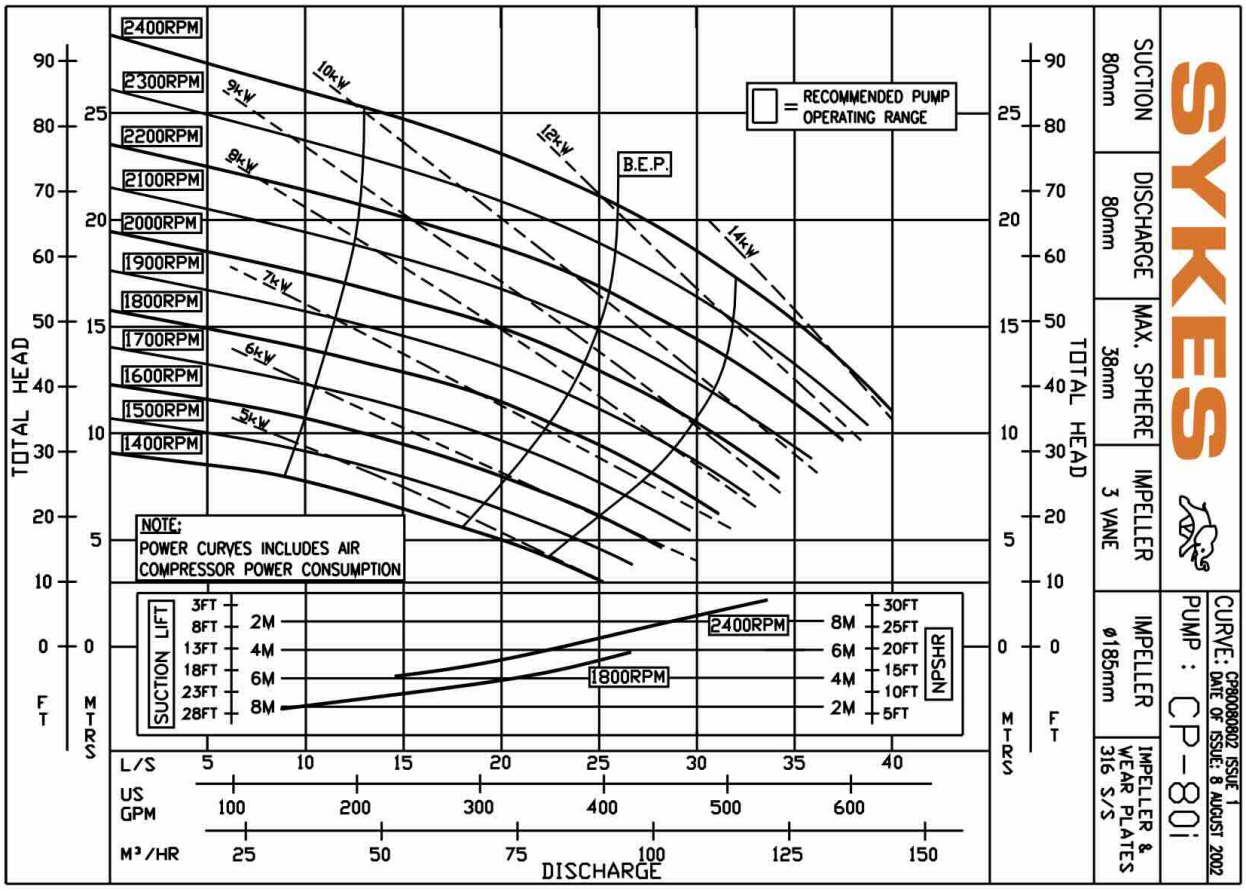


Figure 18: Source: allightsykes.com

Pumps & Pipes!

Subject Tags: Pipe Flow, Pumping Power, Head Loss, Energy Cost, Economics

Difficulty Level: Moderate

Estimated Time: 45 minutes

This problem involves pump selection and pipe flow calculations in a system moving water uphill. You will evaluate total head, energy consumption, NPSH compliance, and assess the cost savings of a proposed modification.

Skills Tested:

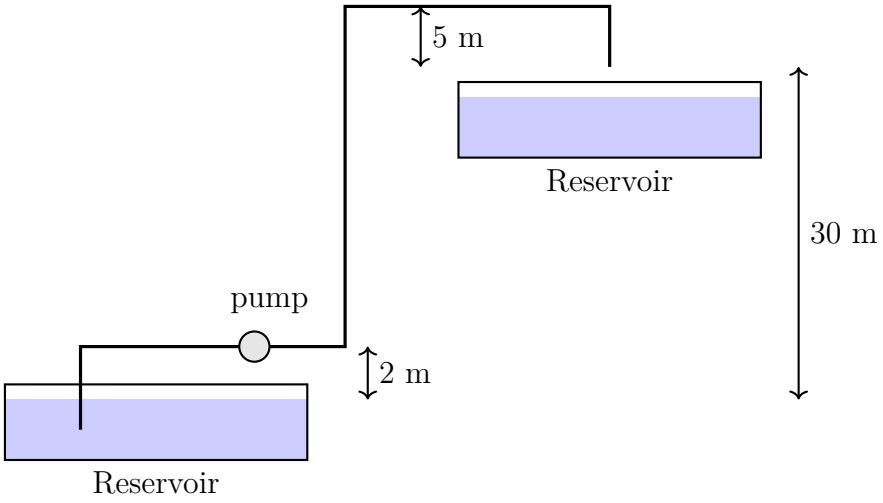
- Bernoulli equation application in real piping systems
- Calculation of major and minor head losses
- Understanding and applying pump curves and NPSHR constraints
- Economic payback analysis from energy usage

Source: CBE 30355 Final Exam, 2014 **File:** problem0152.tex

Pumps & Pipes!

Consider the piping system depicted below.

You are pumping water from one reservoir to another 30 m up the hill at a rate of 10 liters/s through 5 cm ID pipe. There is 5 m of pipe leading to the pump and another 95 m leading to the second reservoir, with a total of four 90º elbows in the system. The pump you are given to use is the HH80, whose pump curve is on the next page.



a. What is the total head the pump is required to supply (in meters of H_2O)?

b. What fraction of the energy cost of operating the pump goes into the change in elevation?

c. Will this violate NPSHR? Be quantitative!

d. Now for the main problem: Your boss isn't happy with the system, and a contractor comes by and proposes to replace the 5 cm pipe with 10 cm pipe, and to operate the pump half the time at twice the flow rate (e.g., 20 liters/s but 12/24 rather than continuous operation). He wants to charge \$5k for the fix. If the cost of electricity is \$0.15/kWhr, how long would it take for the reduction in operating costs to pay back the contractor's fee?

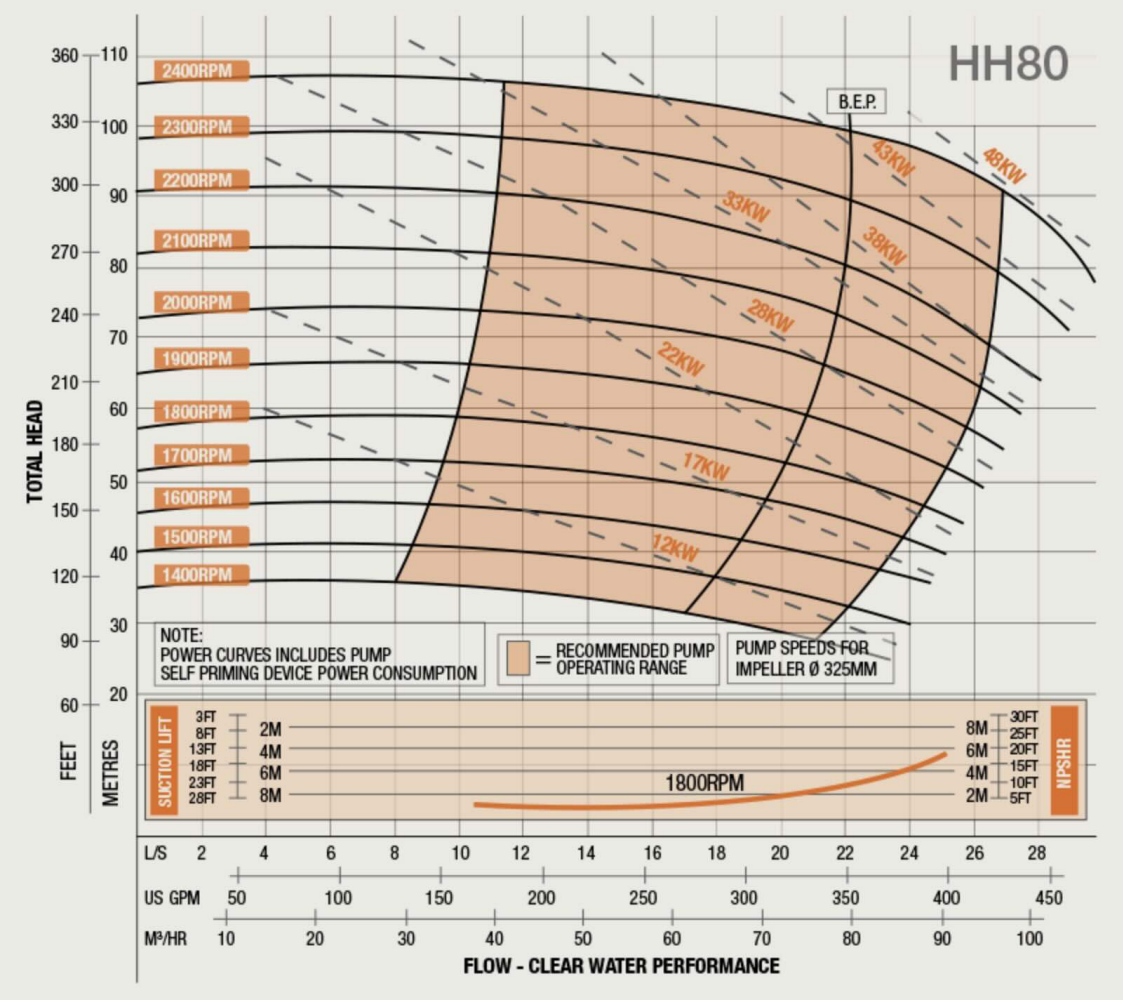
Useful relations:

$$h_L = \left(\sum K + \frac{4 f_f L}{D} \right) \frac{u^2}{2g}$$

$$f_f = \begin{cases} \dfrac{16}{Re} & Re < 2100 \text{ (laminar)} \\ \dfrac{0.0791}{Re^{1/4}} & 3000 < \text{Re} < 10^5 \\ \dfrac{1}{\sqrt{f_f}} = 4.0 \log_{10}(Re\sqrt{f_f}) - 0.4 & \text{Re} > 3000 \end{cases}$$

Fitting loss coefficients:

- Sudden contraction: $K = 0.45$
- Sudden expansion: $K = 1.0$
- 90° elbow: $K = 0.70$



High Re Flow/Pump Curves

Subject Tags: High Reynolds Number Flow, Pump Sizing, Pipe Flow, Energy Cost Estimation

Difficulty Level: Moderate

Estimated Time: 45 minutes

Summary: This problem examines the design and operation of a garden water feature pump system. Students analyze energy requirements, pump placement constraints, and cost trade-offs for pipe sizing using fluid mechanics principles including head loss, energy efficiency, and Reynolds number scaling.

Skills Tested:

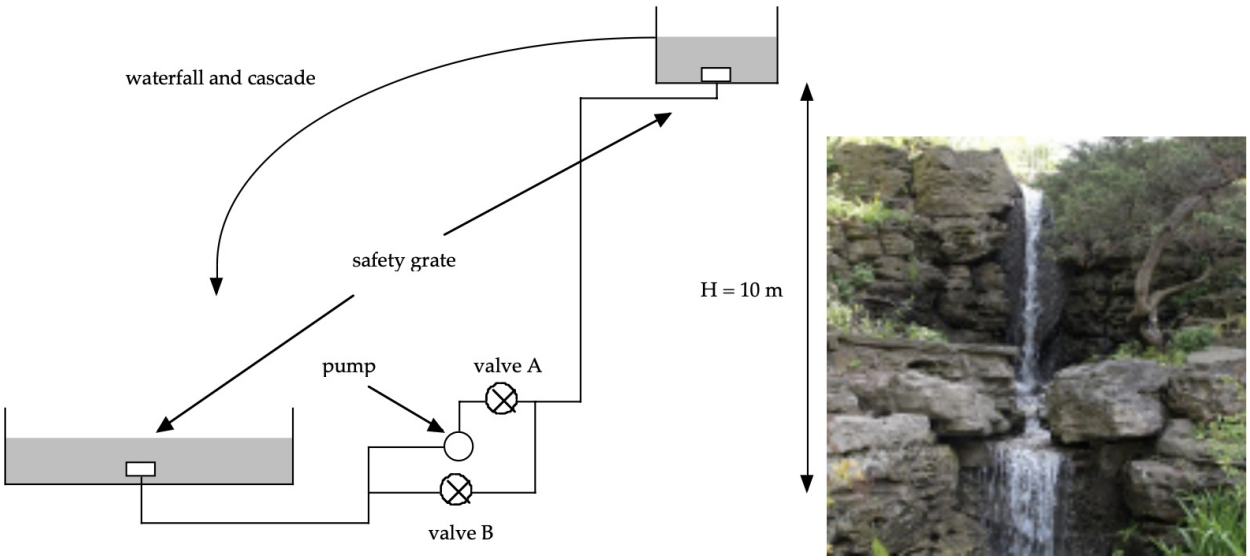
- Bernoulli Equation and head loss evaluation
- Sizing and evaluating pump performance
- Friction factor and minor losses in pipe networks
- Economic evaluation of engineering decisions

Source: CBE 30355 Final Exam, 2018 **File:** problem0162.tex

High Re Flow/Pump Curves

A very attractive part of many public gardens is the water feature (it's my favorite part, anyway). Because most of these gardens aren't blessed with a naturally occurring source of flowing water, the features are usually driven by a pump. Here we size such a system.

Consider the garden feature depicted below (the picture is from the Royal Botanical Garden near Toronto which we've visited many times). Water is pumped from a lower reservoir into an upper reservoir which then cascades in a waterfall and a creek back into the lower reservoir. The piping network consists of 50 m of 3" ID pipe, and begins and ends with a couple of safety grates (K factor of 1.5 each). The valves are gate valves which can be set either for pump operation (valve A open, valve B closed), or for draining the upper reservoir (valve A closed, valve B open). Note that a centrifugal pump usually redirects the flow, so you don't have an extra elbow for it in the diagram below.



a. It is desired to run the waterfall at 15 liters/s. What is the energy requirement for the pump CP-80i to provide this flow? Operating costs are mostly energy, which costs \$0.15/kwhr. What is the daily cost of operating the waterfall?

b. How far above the lower pond can we site the pump before "bad things happen"? Assume you've got about 10 m of pipe leading to the pump.

c. It is proposed to reduce operating costs by using 4" ID pipe rather than 3" ID pipe. The larger pipe currently costs \$2/ft more. Remember how velocity and losses scale with pipe diameter! Using this, estimate (e.g., use the same friction factor, not a terrible approximation) how long reduced operating costs would take to pay for the larger diameter pipe.

d. The upper reservoir has a volume of 5,000 liters. If we close valve A and open valve B, estimate the length of time for the reservoir to drain (with the 3" pipe).

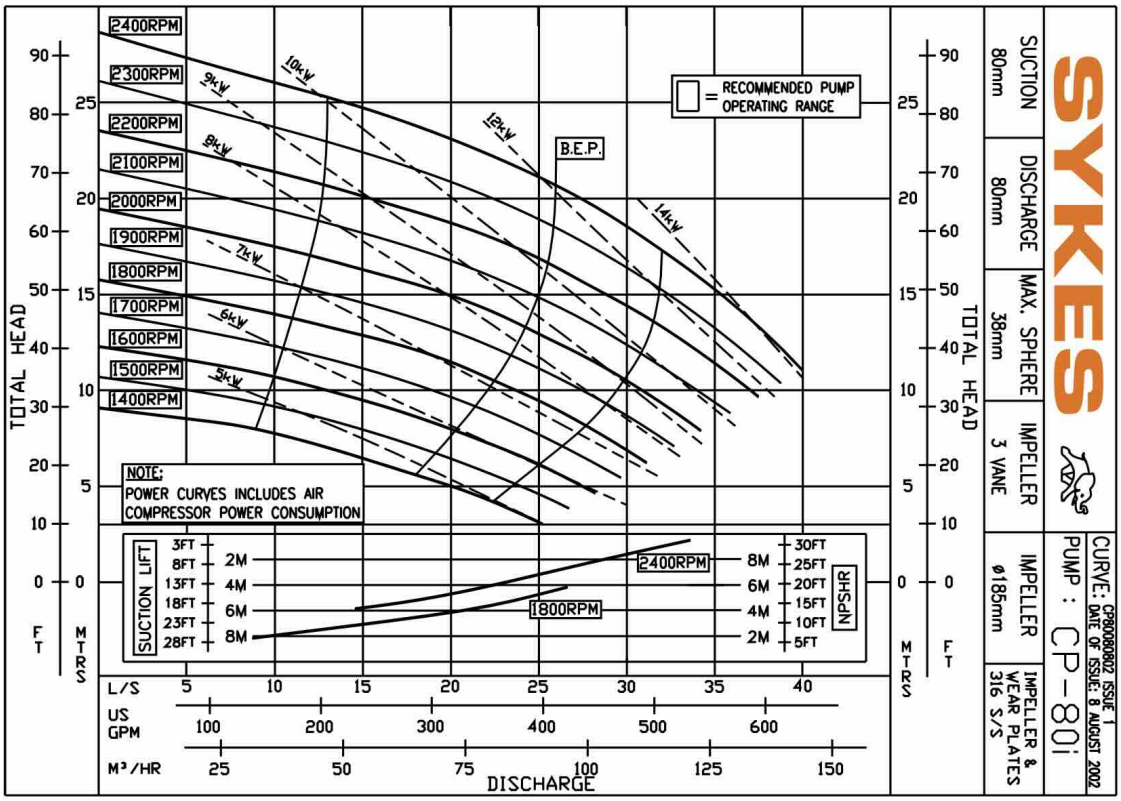
Useful formulas:

$$h_L = \left(\sum K + 4 f_f \frac{L}{D}\right) \frac{u^2}{2g}$$

$$f_f = \begin{cases} \frac{16}{\mathrm{Re}} & \mathrm{Re} < 2100 \\ \text{Blasius: } 0.0791 \cdot \mathrm{Re}^{-0.25} & 3000 < \mathrm{Re} < 10^5 \\ \text{Prandtl: } \frac{1}{\sqrt{f_f}} = 4.0 \log_{10}(\mathrm{Re}\sqrt{f_f}) - 0.40 & \mathrm{Re} > 3000 \end{cases}$$

Fitting K values:

- Safety grate: 1.5
- Gate valve (open): 0.15
- 90° elbow: 0.70
- Tee (straight through): 0.4
- Tee (through side): 1.5



Pumped Hydroelectric Energy Storage in a Residential Setting

Subject Tags: Energy Storage, Hydraulics, Pump Design, Efficiency, Economics

Difficulty Level: Advanced

Estimated Time: 60 minutes

Summary: This problem analyzes a small-scale implementation of pumped hydroelectric energy storage (PHES) in a residential apartment building. It explores mechanical energy conversion, pipe system losses, efficiency and pump curve interpretation, culminating in an assessment of cost recovery over time under ideal assumptions.

Skills Tested:

- Application of energy conversion and storage principles
- Head loss calculation and pump system analysis
- Flow rate conversions and unit consistency
- Efficiency and power curve interpretation
- Economic assessment of engineering systems

Source: CBE 30355 Final Exam, sp2021 **File:** problem0169.tex

Pumped Hydroelectric Energy Storage in a Residential Setting

Pumped Hydroelectric Energy Storage (PHES) is the most widely used energy storage method in the world. The idea is that when your electricity cost is low (e.g., when the sun is shining on all your solar cells or the wind is blowing on your wind turbines) you use the excess generating capacity to pump water to a high elevation, and when it is more costly (or when the sun isn't shining) you run the system backwards to get your electricity back. These systems can be huge: currently the largest in the world is in Bath, Virginia with a generating capacity of 3GW, enough to power 750,000 homes. Large scale systems are quite cost efficient, with a round trip energy efficiency of 70% to 80% and a relatively low capital cost/GW.

While large systems are economically viable (provided you happen to have a nice mountain with a water source nearby), they have also been proposed (and actually implemented) for individual buildings – but they are not nearly as cost effective. Here we examine some of the numbers associated with a PHES installation in an apartment building in France where one was installed (the Goudemand Residence Complex).

a. The two reservoirs (an open reservoir on the roof and storage tanks in the basement) are separated by 30m in elevation and have an exchange volume of 50m^3. Based on this, what is the total amount of electrical energy that the system can provide (assume that the turbine system converting the potential energy of the water back to electricity is about 75% efficient)? Give your answer in kWhr (unit conversion: $1\,\text{kWhr} = 3.6 \times 10^6\,\text{J}$).

b. The pumping system is required to pump water from the basement to the roof at a flow rate of 4 liters/s. How many hours would it take to "fully charge" this battery?

c. The pipe system has not been disclosed, however we shall take it to be 50 meters of 2 inch schedule 40 PVC pipe (ID 5.25 cm), plus a safety grate, a sudden expansion, 8 90° elbows, two T's straight through and an open gate valve. Based on this, what is the total head that the pump has to supply?

d. Mark the operating point on the pump curve below (you will have to put your name on the exam paper and turn it in with the bluebook) and determine the required energy input to the pump. What fraction of this energy actually gets stored as potential energy? What is the return trip efficiency of the system (e.g., electricity out / electricity in, assuming the generation side is 75% efficient)?

e. What is the maximum head loss leading to the pump which could be tolerated before violating NPSHR and cavitating the pump?

f. This apartment building was really a social experiment in energy utilization rather than trying to be cost effective, however we can look at the economics. To calculate the economics we shall assume as a best case scenario that the pump energy is "free" (it is excess energy from wind turbines and solar cells in this installation) and that the reservoir is fully cycled through the system on average once per day. If the cost of electricity is \$0.15/kWhr, what is yearly value of the energy stored by the system? The PHES system was estimated to cost \$40k. How many years would it take to recover the capital cost under this idealized (no recurring maintenance or energy costs) scenario? (Hint: get it right, but don't be disturbed if you find it is a really long time...)

Useful formulas and data:

$$h_L = \left(\sum K + 4 f_f \frac{L}{D}\right) \frac{u^2}{2g}$$

$$f_f = \frac{16}{\mathrm{Re}} \quad \text{(laminar, Re} < 2100) \qquad f_f \approx 0.0791\,\mathrm{Re}^{-1/4} \quad (3000 < \mathrm{Re} < 10^5)$$

$$\frac{1}{\sqrt{f_f}} = 4.0 \log_{10}(\mathrm{Re}\, f_f) - 0.40 \quad ; \quad \mathrm{Re} > 3000$$

Fitting K values	
Safety grate	1.5
Sudden expansion	1.0
90° elbow	0.7
T (straight through)	0.4
Gate valve (open)	0.15

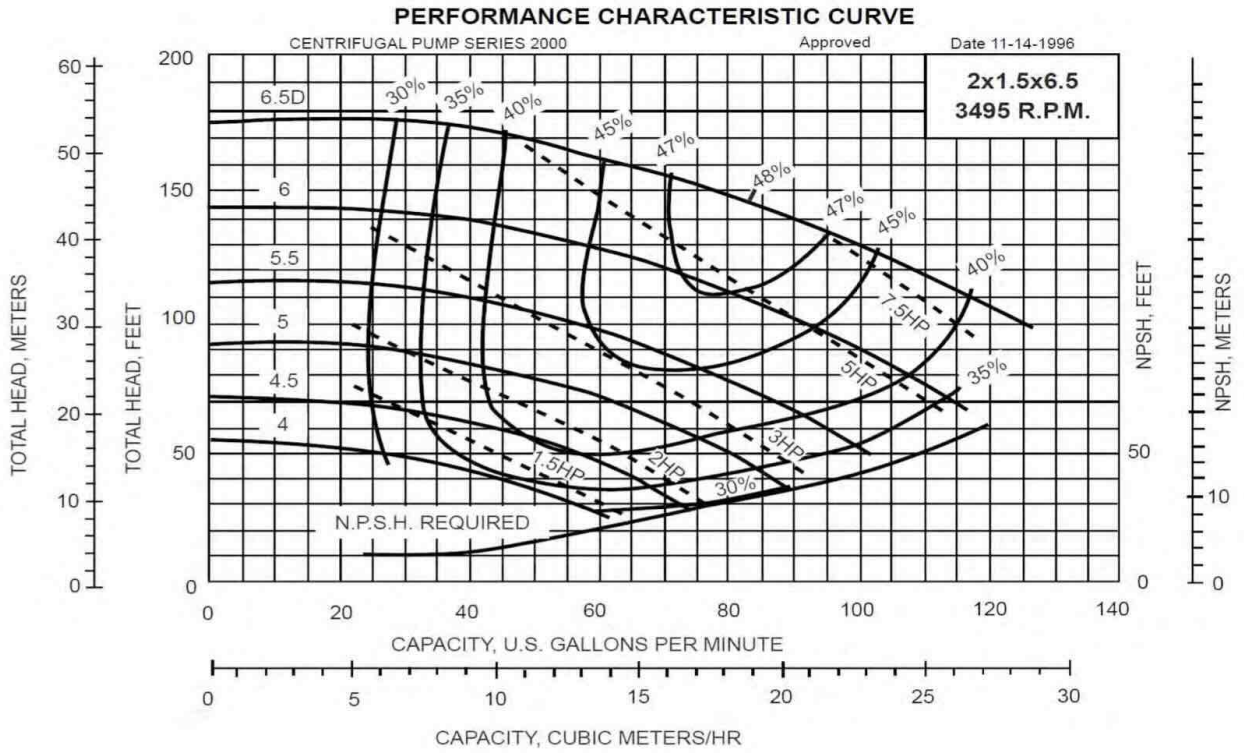


Figure 19: Source: Goulds Water Technology. Note: This pump curve gives the power requirement in HP (unit conversion: 1 HP = 0.7457 kW). The curved lines are pump efficiencies – very convenient. The flow rate is in gal/min (unit conversion: 1 liter/s = 15.85 gal/min). And, of course, 1 meter = 3.28 ft.

Irrigation Pumping with CP-80i: Flow Rate and Efficiency Estimation

Subject Tags: Pump Selection, Head Loss, Energy Balance, Pipe Flow, Pump Curves

Difficulty Level: Moderate

Estimated Time: 40 minutes

Summary: This problem explores the design and performance evaluation of a pumped irrigation system using the CP-80i pump. Students assess both ideal and real-world flow rates considering power constraints, pipe losses, and pump efficiency. The problem also includes practical system design decisions regarding pump placement and suction-side conditions.

Skills Tested:

- Head loss estimation using empirical formulas and fittings
- Reading and interpreting pump curves
- Power-to-flow rate conversion and pump sizing
- System efficiency calculation and pump placement considerations

Source: CBE 30355 Final Exam, 2021 **File:** problem0171.tex

Irrigation Pumping with CP-80i: Flow Rate and Efficiency Estimation

You are tasked with using the pump CP-80i (pump curve below) to pump water 10 m uphill to fill a reservoir for irrigation purposes. The problem is that you only have a 10 kW power supply for the pump and need to determine what flow rate you are going to get for the system. So:

a. If you have a really large diameter pipe for your system, pipe losses will be negligible. In this case, what flow rate can you expect (really the upper limit) and what would the pump efficiency be?

b. The proposed piping network consists of 40 m of 3" schedule 40 PVC tubing (ID = 7.72 cm), together with 6 90° elbows and a safety grate at the inlet with $K = 1.5$. Given this, get a better estimate of the flow rate you can expect from the pump (still running at 10 kW). Note that this would require a few iterations to get the exact value, so just show your work and try to get close in a short amount of time. Kick the iteration off by guessing that losses are about equal to the change in height (not a bad guess, actually) and using the appropriate flow rate from the pump curve. I'm looking for method here!

c. The overall efficiency of the system would be the useful work out (e.g., the potential energy gained by the water by pumping uphill) divided by the energy into the pump. Note that this is intrinsically lower than the pump efficiency. What is this overall efficiency based on the flow rate you calculate in part **b**?

d. The piping leading to the pump from the source consists of two elbows, the safety grate, and 4 meters of pipe. Based on this, what recommendations can you make about the placement of the pump relative to the water source?

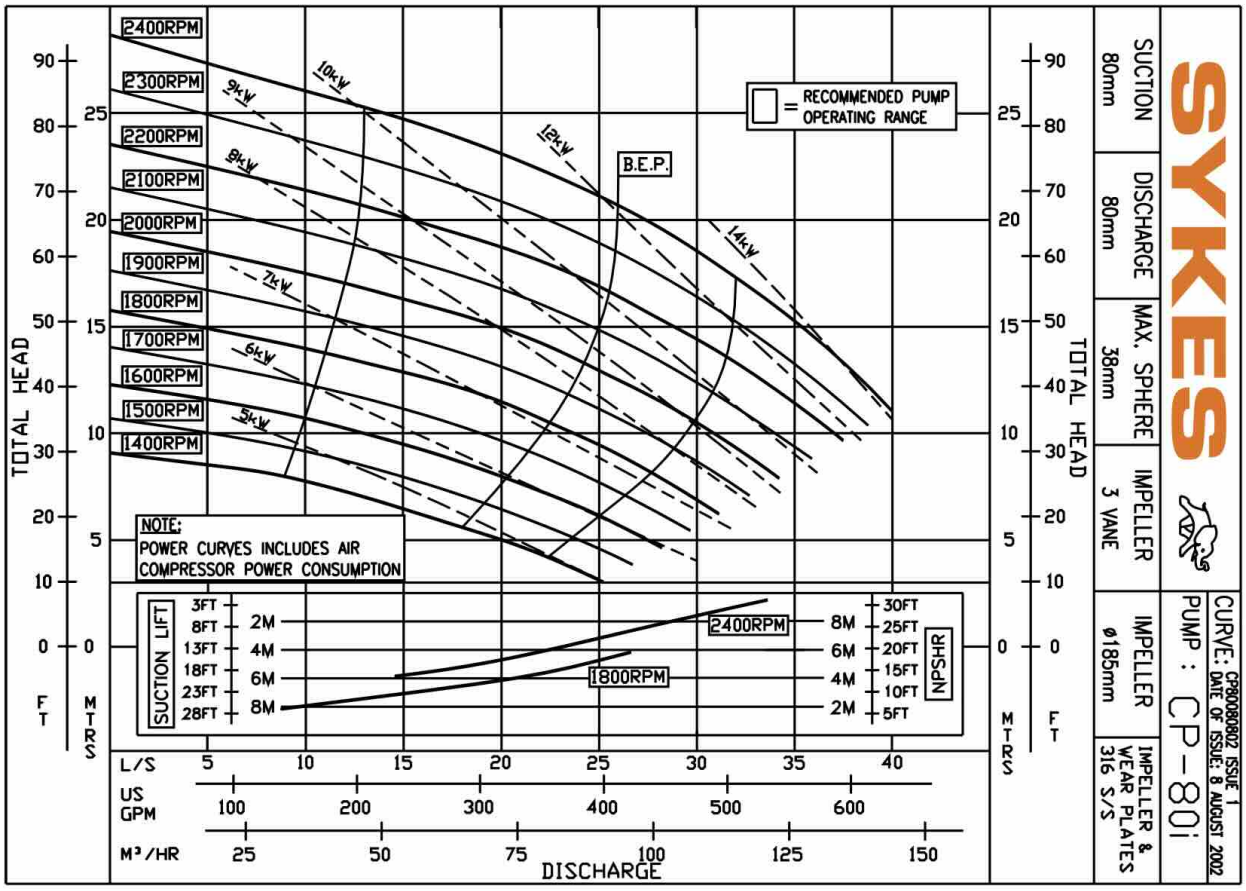
Useful relations:

$$h_L = \left(\sum K + \frac{4 f_f L}{D} \right) \frac{u^2}{2g}$$

$$f_f = \begin{cases} \dfrac{16}{Re} & Re < 2100 \text{ (laminar)} \\ \dfrac{0.0791}{Re^{1/4}} & 3000 < \text{Re} < 10^5 \\ \dfrac{1}{\sqrt{f_f}} = 4.0 \log_{10}(Re\sqrt{f_f}) - 0.4 & \text{Re} > 3000 \end{cases}$$

Fitting K values:

- Sudden expansion: 1.0
- 90° elbow: 0.90
- Safety grate: 1.5



Pump Curve Analysis and Application

Subject Tags: Pumps; Fluid Transport; Energy Balance; Efficiency; System Design

Difficulty: Moderate

Estimated Time: 35 minutes

Summary: This problem assesses your ability to interpret and apply a pump performance curve to a practical fluid transport scenario. You will evaluate the viability of a selected pump model for a given elevation and flow requirement, calculate operating speed, determine work and efficiency, and assess constraints on pump placement.

Skills Tested:

- Pump selection from performance curves
- Head and energy calculations
- Determination of efficiency from given conditions
- Evaluation of cavitation risk and safe pump placement

Source: CBE 30355 Final Exam, 2001 **File:** problem0016.tex

Pump Curves

The following questions refer to the pump curve below.

a. It is desired to pump 20 liters/sec from a pond to an elevation of 70 meters. If we neglect all frictional losses (say we use a really fat pipe!), is the pump HH80 recommended for the job?

b. What is the RPM required to do the job?

c. What is the work done by the pump on the fluid?

d. What is the efficiency of the pump at the operating conditions?

e. It may be convenient to put the pump part-way up the hill (you don't want it to flood at high pond water levels!). What is the maximum height it can be located at for safe operation? (Note: $1\,\text{atm} \approx 10.3\,\text{m}$ water)

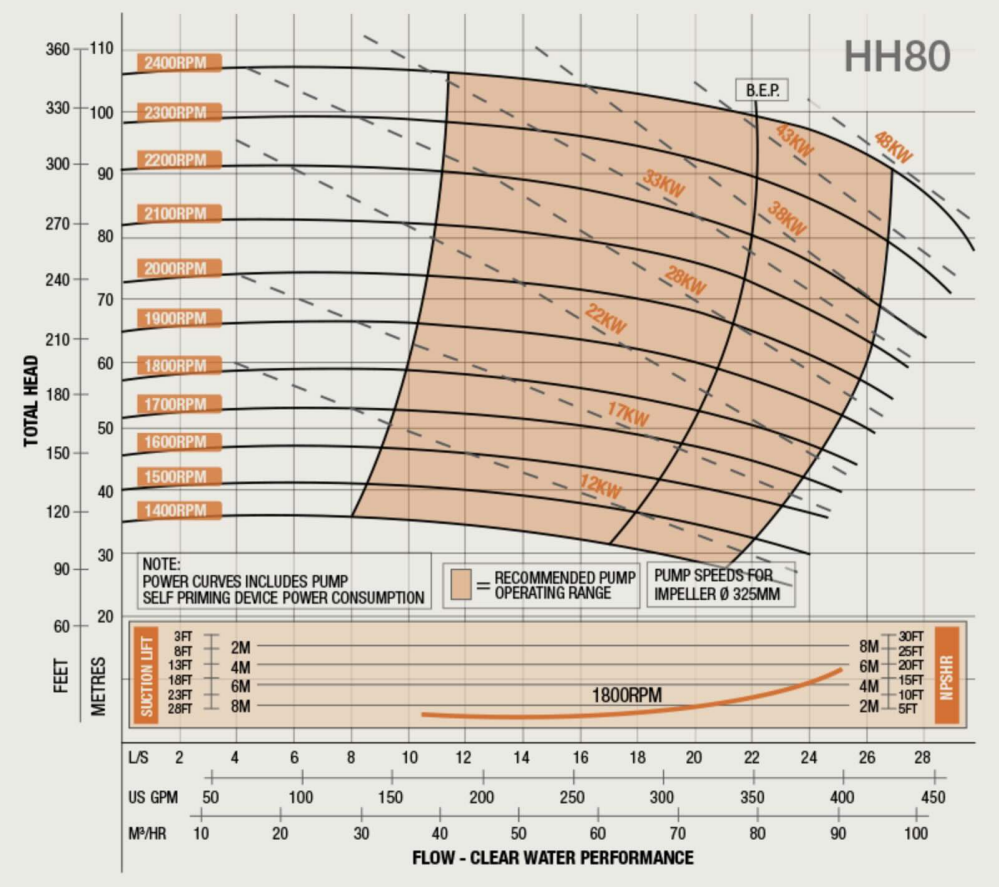


Figure 20: Sykes Pump HH80, Source: allightsykes.com

Pump Curves

Subject Tags: Pump curve, fluid mechanics, mechanical energy, head, efficiency, cavitation
Difficulty Level: Moderate
Estimated Time: 25 minutes

This problem involves interpreting pump curves and applying fluid mechanical principles to determine feasibility and limitations in pump performance, efficiency, and operation in high-altitude environments. You will also consider cavitation criteria when working at reduced atmospheric pressure.

Skills Tested:

- Reading and interpreting pump curves
- Applying Bernoulli and mechanical energy balance
- Calculating pump efficiency
- Estimating cavitation risk

Source: CBE 30355 Final Exam, 2011, solved **File:** problem0038.tex

Pump Curves

a. It is desired to pump 35 liters/sec of water from a pond to an elevation of 15 meters. If we neglect all frictional losses (say we use a really fat pipe!) is the pump CP80i recommended for the job?

b. What is the RPM required to do the job?

c. What is the useful mechanical work done by the pump on the fluid?

d. What is the efficiency of the pump at the operating conditions?

e. What vertical distance from the level of the pond can we put the pump at? (Again, neglect frictional losses)

f. If it is intended to use the pump at the ski resort in Loveland, CO (at 10,000 ft the pressure is about 2/3 of sea level), how does the answer change? If we are pumping hot water at its boiling point, what is the required height?

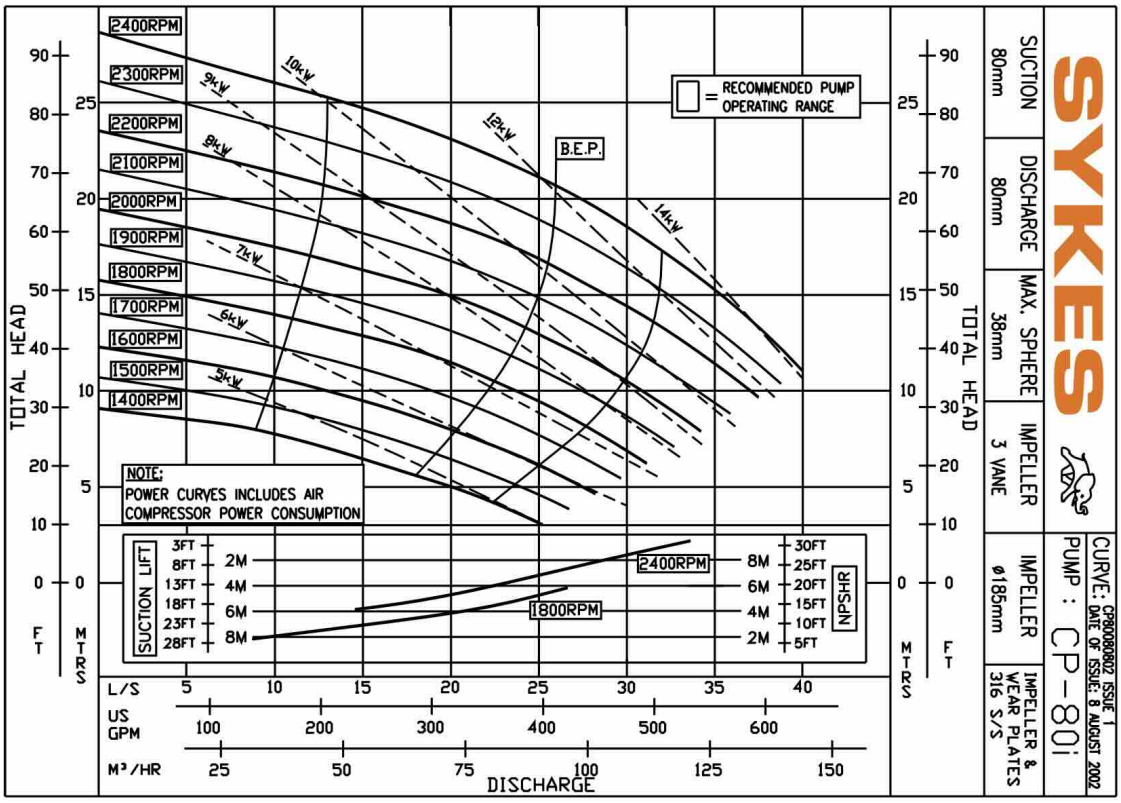


Figure 21: Source: allightsykes.com

Pump Curve Performance and Sizing

Subject Tags: Pump Curve, Fluid Elevation, Power, Efficiency, Pipe Sizing

Difficulty Level: Moderate

Estimated Time: 50 minutes

Summary: This problem focuses on the analysis of pump performance data to evaluate system feasibility, power requirements, and pipe sizing. The student must assess whether a given pump (HH150) can lift water to a specified elevation at a target flow rate, determine mechanical power output, efficiency, suction limits, and propose realistic pipe sizes based on fluid dynamics and practical considerations.

Skills Tested:

- Interpretation of pump performance curves
- Energy and power analysis in fluid systems
- Determination of mechanical work and efficiency
- Estimation of suction lift constraints
- Pipe sizing and frictional head loss estimation

Source: CBE 30355 Final Exam, 2010 **File:** problem0142.tex

Pump Curve Evaluation and Design

The following questions refer to the pump curve below:

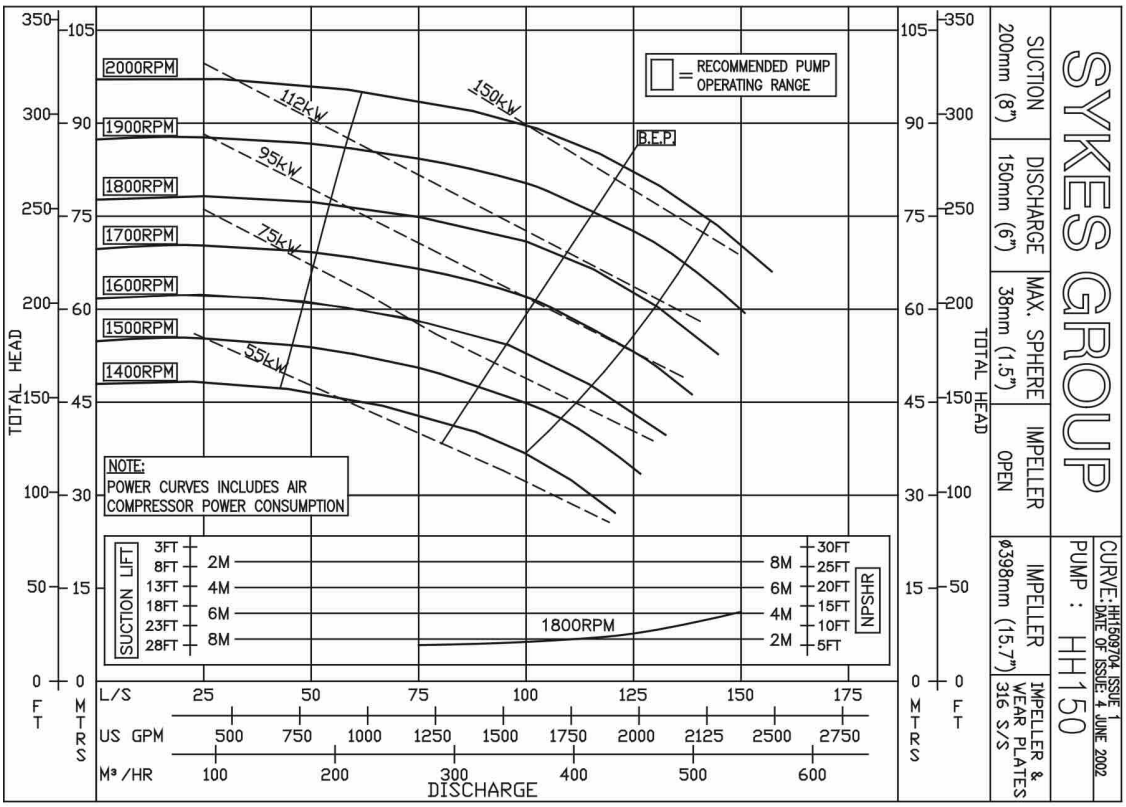


Figure 22: Source: allightsykes.com

a. It is desired to pump 75 liters/sec from a pond to an elevation of 75 meters. If we neglect all frictional losses (say we use a really fat pipe!), is the pump HH150 recommended for the job?

b. What is the useful mechanical work done by the pump on the fluid per unit time?

c. What is the efficiency of the pump at the operating conditions?

d. How far up the hill from the level of the pond can we put the pump? (Again, neglect frictional losses)
(Note: 1 atm ≈ 10.3 m water)

e. Frictional losses always add to the required head. What additional head losses can we tolerate before the pump is unable to achieve the required flow rate?

f. Your boss proposes to use a 10 cm diameter pipe for this system. Quantitatively demonstrate why this is probably a bad idea.

g. About how big should the pipe be instead? Make any approximations you think are reasonable.

Collated Short Answer Questions

This section presents the short answer questions from the CBE30355 Transport I examinations. There are over 100 short answer questions which have been used on these exams over the decades. In order to make them more useful, I have divided them into rough categories. These sorted questions are stored in the .tex files designated on each page. The questions are all part of a numbered list, and are individual "items" in that file. To extract a desired question, open the file in TeXShop and copy the item. It should be pasted into the file `ashortanswertemplate.tex` which can then be input into your exam or problem set template in the same manner as the problems.

On my transport exams the short answer questions were either in separate groups, or paired with pump curve evaluations. These exam questions were also reduced to LaTeX in the same manner as the other exam questions, however in the interests of space and clarity they are not included in this Compendium directly. Rather, these files (which are in the .zip folder) may be viewed in the companion document `ashortanswercollection.pdf` or (in TeXShop) `ashortanswercollection.tex` from which the PDF can be compiled. The groups of short answer questions in these problems can be directly inserted into an exam or homework template in the same manner as the other problems in this Compendium.

High Reynolds Number Short Answer Questions

The following table has the collected short answer questions focusing on high Reynolds number flows. For ease of extracting individual problems, the number in the typescript compendium is also added as a comment before each "item". The item, down to (or including) the vspace command should be copied into the short answer template for incorporation into an exam or homework assignment.

1. At high Re, drag principally results from:

A. Potential Flow

B. Boundary Layer Separation

C. Skin Friction

D. Turbulence

2. Why do dimpled golf balls and fuzzy tennis balls have less drag than their smooth counterparts? (One sentence, please.)

3. Why do frisbees have ridges? Briefly, please!

4. What is the kinetic energy per unit volume of a fluid (in terms of the velocity and material properties)? How does it relate to the pressure at high Re?

5. How is a drag coefficient defined at high Re?

6. What is boundary layer separation, and why does it matter?

7. Briefly describe one method for preventing stall on an aircraft wing.

8. Briefly describe two methods discussed in class for avoiding boundary layer separation on a wing.

9. What were the three ways of remediating stall discussed in class?

10. The displacement thickness is defined as:

$$\delta^* = \int_0^\infty \left(1 - \frac{u}{U}\right) dy$$

Provide a brief physical interpretation of this quantity.

11. The momentum thickness is defined as:

$$\theta = \int_0^\infty \frac{u}{U}\left(1 - \frac{u}{U}\right)\, dy$$

Provide a brief physical interpretation of this quantity.

12. Experimentally, how can you most accurately calculate the shear stress on a plate in boundary layer flow at high Reynolds numbers from the velocity profile?

13. What is the Von Karman Momentum Balance useful for, and why?

14. Write out Bernoulli's Equation. What is it a statement of (e.g., where does it come from)?

File: sahighre.tex

Turbulence Short Answer Questions

The following table has the collected short answer questions focusing on turbulent flows. For ease of extracting individual problems, the number in the typescript compendium is also added as a comment before each "item". The item, down to (or including) the vspace command should be copied into the short answer template for incorporation into an exam or homework assignment.

1. Give a physical description of the Reynolds stress (e.g., where does it come from, and how is it defined?).

2. What is the "Law of the Wall"? Where does it apply and where does it come from? Be brief.

3. For a shear stress of 16 dynes/cm^2 in the turbulent flow of water through a pipe, about how rough does the pipe wall have to be before it influences the flow?

4. For a shear stress of 25 dynes/cm^2 in the flow of water through a pipe, about how rough does the pipe wall have to be before it influences the flow?

5. For a shear stress of 100 dynes/cm^2 in the flow of water through a pipe, about how rough does the pipe wall have to be before it influences the flow?

6. The turbulent friction velocity is a characteristic velocity created from a shear stress and fluid properties. What is it in terms of these parameters?

7. The turbulent viscous length scale is a characteristic length created from a shear stress and fluid properties. What is it in terms of these parameters?

8. What is the approximate thickness of the viscous sublayer in plus units?

9. Just as in the case of turbulent momentum transfer where we define a turbulent kinematic viscosity ν_t, so in the case of turbulent energy transfer we can define a turbulent thermal diffusivity α_t. The turbulent Prandtl number is the ratio of these quantities ($Pr_t = \nu_t/\alpha_t$). What is its approximate magnitude and why?

File: saturbulence.tex

Low Reynolds Number Short Answer Questions

The following table has the collected short answer questions focusing on low Reynolds number flows. For ease of extracting individual problems, the number in the typescript compendium is also added as a comment before each "item". The item, down to (or including) the vspace command should be copied into the short answer template for incorporation into an exam or homework assignment.

1. What is the relationship between the Stokes Equations, Stokes' Law and Stokes' Paradox?

2. What does Stokes' Law describe, and where is it valid?

3. What is its mathematical expression in terms of parameters (with the constant too!)?

4. What equation governs the streamfunction for 2-D flow at zero Re?

5. What is a Stokeslet?

6. In our class demonstration, when we allowed a rod to settle in a viscous liquid (Re = 0), we observed that it didn't rotate. If inertial effects are important, it does. Why is the behavior different in this case?

7. Prove that inertial effects always increase the drag on an object relative to that caused by viscous effects alone. Be brief!

8. A tube of radius R has a surface roughness of size ϵR. At high Reynolds number the roughness is found to have a large effect on the pressure drop, while at low Reynolds number it is only of $\mathcal{O}(\epsilon)$. Briefly explain why the effect is so small at low Re.

9. What is the minimum dissipation theorem and where does it apply?

10. If two identical spheres of radius a are settling due to the force of gravity at zero Reynolds number in an infinite fluid (no boundaries), how does the vector drawn between the sphere centers change with time? Justify your answer!

11. What is Stokes Paradox? Why does it affect flow in a shower drain?

12. Order the drag (from smallest to largest) of the following objects under creeping flow (Re $\ll 1$) conditions:

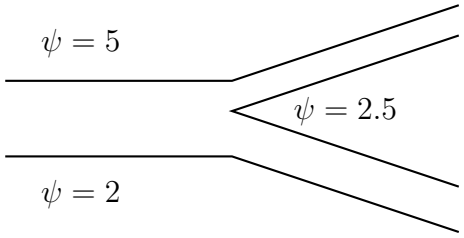
A. A cube 2 cm on a side

B. A sphere 2 cm in diameter

C. A sphere 1.75 cm in radius

13. In senior lab yesterday a student tried to get the last bit of glycerin off of the walls of a graduated cylinder by shaking it up and down (e.g., an oscillatory axial motion). Using your knowledge of creeping flow, briefly explain why this did not increase the drainage rate. Three words will suffice if they're the right ones!

14. Computer fluid simulations were carried out in a 2-D asymmetric channel bifurcation as presented below. The simulation reveals the stream function values at the channel walls. What is the flow percentage that moves through the upper branch of the channel?



15. 2-D Stokes flow has a simple analogy in solid mechanics. What is it, and how does it work? No more than two sentences!

File: salowre.tex

Term Identification Short Answer Questions

The following table has the collected short answer questions focusing on the identification of terms in transport problems. For ease of extracting individual problems, the number in the typescript compendium is also added as a comment before each "item". The item, down to (or including) the vspace command should be copied into the short answer template for incorporation into an exam or homework assignment.

1. What expression describes the volumetric flux normal to a surface, and what are its units?

2. What is the mathematical representation of the material derivative, and what does it physically represent?

3. Provide two interpretations of $\tilde{u}$.

4. Provide two interpretations of $\rho\tilde{u}$.

5. Briefly identify the physical mechanism described by each of the following terms:

a. $\mu\dfrac{\partial^2 u_x}{\partial y^2}$

b. $\rho\dfrac{D\rho}{Dt}$

c. $-\dfrac{\rho v_\theta^2}{r}$

d. $\dfrac{\rho v_r v_\theta}{r}$

e. $E^4\psi = 0$

f. $\nabla \times \mathbf{u}$

g. $\dfrac{\partial u_i}{\partial x_i}$

h. $\rho\dfrac{u_r u_\theta}{r}$

6. For each term in the full momentum equation below, identify the physical mechanism:

$$\rho\frac{\partial u_i}{\partial t} + \rho u_j\frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \mu\frac{\partial^2 u_i}{\partial x_j^2} + \rho g_i$$

7. Briefly identify the physical mechanism described by each term in the following equation:

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = \alpha \nabla^2 T$$

8. Consider the term $\rho \dfrac{u_\theta u_r}{r}$:

a. Name a physical phenomenon where it plays an important role.

b. Which component of the Navier-Stokes equations does it appear in?

9. In the Navier-Stokes equations, what term would represent viscous diffusion of x-momentum in the y-direction?

File: satermid.tex

Equation Identification Short Answer Questions

The following table has the collected short answer questions focusing on the identification of the equations used in transport problems. For ease of extracting individual problems, the number in the typescript compendium is also added as a comment before each "item". The item, down to (or including) the vspace command should be copied into the short answer template for incorporation into an exam or homework assignment.

1. What equation does a harmonic function satisfy (name or mathematical expression)?

2. Briefly identify the physical mechanism (or equation name) described by each of the following terms and a problem where it would play a role:

a. $\dfrac{\partial \sigma_{ij}}{\partial x_j} = 0$

b. $E^4\psi = 0$

c. $\epsilon_{ijk}\dfrac{\partial u_k}{\partial x_j} = 0$

3. For each of the following equations, briefly list the most important conditions under which it is valid (e.g., what flows is it good for?).

a. $\nabla^4\psi = 0$

b. $\dfrac{\partial \sigma_{ij}}{\partial x_j} = 0$

c. $p + \frac{1}{2}\rho u^2 + \rho g h = \text{Cst}$

d. $\dfrac{\partial u_i}{\partial x_i} = 0$

e. $\mathbf{u} = -\nabla\phi$

4. For the five expressions below, briefly identify the corresponding physical mechanism (or equation name) and a problem where it would play a role:

a. $\dfrac{\partial \sigma_{ij}}{\partial x_j} = 0$

b. $\rho\dfrac{D\vec{u}}{Dt} = -\vec{\nabla}P$

c. $\nabla^4\psi = 0$

d. $Q = -\dfrac{\pi R^4}{8\mu}\dfrac{\Delta P}{L}$

e. $\dfrac{D\phi}{Dt} \equiv \dfrac{\partial \phi}{\partial t} + \vec{u} \cdot \nabla \phi$

5. What equation governs the streamfunction for 2-D flow at zero Re?

6. Provide a physical interpretation for each term in Bernoulli's equation.

7. Briefly identify the following equations and a problem where each would play a role:

a. $D_0 = \dfrac{kT}{6\pi\mu a}$

b. $U_s = \dfrac{2\Delta\rho g a^2}{9\mu}$

c. $\nabla^4 \psi = 0$

d. $Q = -\dfrac{\pi R^4 \Delta P}{8\mu L}$

e. $p + \dfrac{1}{2}\rho u^2 + \rho g h = \text{const.}$

Short Answer Calculations

The following table has the collected short answer questions which involve a little calculation rather than just identification. For ease of extracting individual problems, the number in the typescript compendium is also added as a comment before each "item". The item, down to (or including) the vspace command should be copied into the short answer template for incorporation into an exam or homework assignment.

1. Estimate the Reynolds number of a 0.5 cm diameter minnow swimming at 10 cm/s.

2. Estimate the Reynolds number of a 2 cm diameter goldfish swimming at 20 cm/s.

3. My big orange goldfish, alas, passed away, and the little one left in the office is only about 1.5 cm wide. Unless it is hungry, it also usually moves with a velocity of around 0.5 cm/s. Is its motion dominated by inertial or viscous effects? Be quantitative!

4. A paramecium is a single celled animal around 200 μm in size that tends to swim around in circles with a velocity of 1 mm/s. Estimate its Reynolds number.

5. A small jellyfish is about 5 cm across and moves at a velocity of about 1 cm/s. Estimate its Reynolds number. Is its motion dominated by inertial or viscous effects?

6. Estimate the Reynolds number of a swimmer who swims a kilometer in 20 minutes.

7. A sphere of density ρ_m (greater than the fluid density ρ) and radius a is sitting on the wall of a microfluidic channel of half-width b ($b \gg a$). At high enough mean fluid velocities U, inertial lift effects will overcome gravity and cause the sphere to lift off from the wall. Estimate this critical velocity.

(Hint: In the limit $b \gg a$, the lift scales with the shear rate at the wall, not directly with the mean velocity. Use your knowledge of the scaling of inertial forces! Calculation of the lift coefficient was actually my first task as a graduate student many years ago...)

8. It is proposed to model the flow patterns in a microfluidic chip of characteristic channel depth of 25 μm and velocity of 1 cm/s with a macroscopic scale model of channel depth 2 mm. If the working fluid of the chip is water, and that of the model is a mixture of glycerin and water with kinematic viscosity 0.2 cm^2/s, what should the velocity of the fluid in the model be?

9. It is proposed to model the flow patterns in a microfluidic chip of characteristic channel depth of 100 μm and velocity of 1 cm/s with a macroscopic scale model of channel depth 2 mm. If the working fluid of the chip is water, and that of the model is a

mixture of glycerin and water with kinematic viscosity 0.1 cm^2/s, what should the velocity of the fluid in the model be?

10. In ship design, the theoretical maximum velocity of an important class of single hull designs (heavy, deep keel boats) occurs when the Froude number (based on the length of the hull at the waterline) is about 0.16. What is this maximum velocity for a 90 m long ship of this type?

11. In ship design, the theoretical maximum velocity of an important class of single hull designs (heavy, deep keel boats) occurs when the Froude number (based on the length of the hull at the waterline) is about 0.16. What is this maximum velocity for a 10 m boat of this type?

12. In ship design, the theoretical maximum velocity of an important class of single hull designs (heavy, deep keel boats) occurs when the Froude number (based on the length of the hull at the waterline) is about 0.16. Recent results from the Mars missions have shown that it once had seas much like Earth's. Given that the gravity on Mars is only 0.38 that of Earth, what would the maximum velocity of a 25 m ship have been on Mars?

13. Karo syrup has a dynamic viscosity of 40 poise. Glycerin has a dynamic viscosity of 14 poise. You prepare a mixture consisting of 75ml of Karo syrup and 25ml of glycerin (they are miscible fluids, and nothing weird happens when you mix them). About what would you expect the viscosity of this mixture to be?

14. A sphere of radius a has a surface temperature of 1. If the temperature far from the sphere is zero, what is the temperature distribution in the solid surrounding the sphere?

15. The ratio of the centerline velocity divided by the average velocity:

a. for laminar channel flow is:

b. for laminar tube flow is:

16. You put a sheet of rubber of thickness 2mm on a hot surface. The rubber has a thermal diffusivity of about 0.001 in cgs units. About how long would you have to wait before the top of the sheet warms up? Show the basis of your calculation.

17. You put a block of glass of thickness 2 mm on a hot surface. The glass has a thermal diffusivity of about 0.004 in cgs units. About how long would you have to wait before the top of the glass warms up? Show the basis of your calculation.

18. A glass sphere of radius 5 mm with thermal diffusivity $\alpha = 10^{-2}\,\mathrm{cm}^2/\mathrm{s}$ is dropped into hot oil. About how long will it take for the temperature at the center of the sphere to equilibrate?

19. A sphere of radius 1 mm with thermal diffusivity $\alpha = 10^{-3}\ \mathrm{cm}^2/\mathrm{s}$ is dropped into hot oil. About how long will it take for the temperature at the center of the sphere to equilibrate?

General Short Answer Questions

The following table has the collected short answer questions which did not fit into the other categories, often general questions regarding transport phenomena. For ease of extracting individual problems, the number in the typescript compendium is also added as a comment before each "item". The item, down to (or including) the vspace command should be copied into the short answer template for incorporation into an exam or homework assignment.

1. At the Mole Hole (a gift store on the East Race) I saw an interesting variant on a Galileo's Thermometer. Front and back pictures of it are reproduced below. It is filled with a fluid, probably water, and one end of a chain is attached to the wall, and the other to the floating disk (you can see this more clearly from the back side picture). Briefly explain how the chain makes the thermometer work.

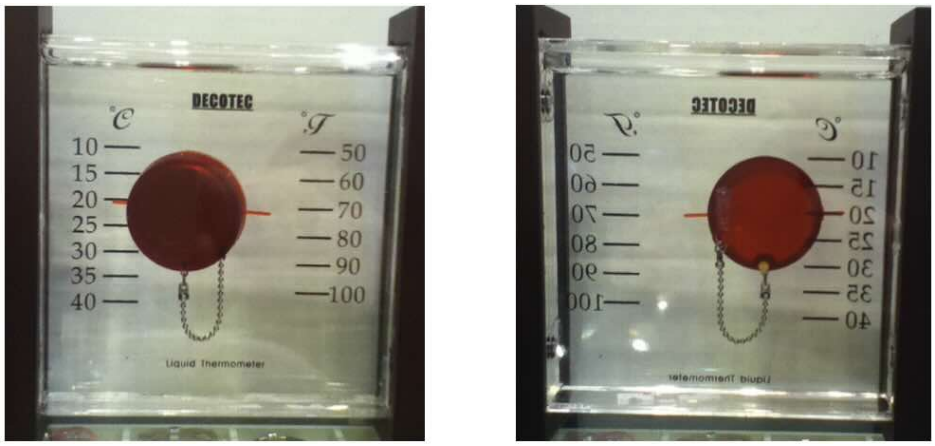


Figure 23: Galileo Thermometer

2. How is the Prandtl number defined, and what does it represent?

3. The Prandtl number is the ratio of the kinematic viscosity to the thermal diffusivity $\left(\mathrm{Pr} = \frac{\nu}{\alpha}\right)$. What is its approximate magnitude for a low density gas and why?

4. What does the Froude number represent?

5. What is the difference between strict and approximate dynamic similarity, giving one example?

6. What is a Jeffrey Orbit? (I demonstrated this in class)

7. What forms when the critical Taylor number is exceeded?

8. What are Taylor-Couette vortices, and why do they occur?

9. What is the capillary number Ca? (How is it defined, and what is it good for?)

10. What is the difference between quasi-parallel flow and uni-directional flow? Give a (brief!) example of each.

11. Under what condition is the rate of strain tensor traceless?

12. Briefly identify the following:

- **a.** Stokes Paradox
- **b.** D'Alembert's Paradox
- **c.** Osmotic Pressure
- **d.** Ideal Flow
- **e.** Electroosmosis
- **f.** Cold Gas Approximation
- **g.** Brownian Motion
- **h.** Stokeslet
- **i.** Richardson Number
- **j.** Poiseuille's Law

13. What does NPSHR stand for, and why should we care?

14. Ensuring that the NPSHR is met for a pump installation will help prevent:

- **A.** Cavitation in the pump
- **B.** Damage to pipes and fittings due to vibrations
- **C.** Reduced pump performance
- **D.** All of the above

15. What is the difference between an orifice meter and a venturi meter?

16. Sketch and briefly explain the principle behind a venturi flow meter.

17. Sketch and briefly explain the principle behind an orifice flow meter.

18. In ship modeling, we try to preserve which dimensionless number?

A. Reynolds Number

B. Prandtl Number

C. Froude Number

D. Weissenberg Number

19. Which (if any) of the following can be discontinuous at a fluid-fluid interface?

A. Shear stress

B. Heat flux

C. Mass flux

D. Velocity

20. Which of the following operate in an inertia-dominated regime? (Select all that apply.)

A. A car

B. A bicycle

C. A minnow

D. A LabChip

21. Why did the Tacoma Narrows Bridge collapse? *One sentence, please.*

22. What are the key assumptions which lead to Newton's Law (of viscosity)?

23. Which (if any) of the following must be continuous at a fluid-fluid interface?

A. Shear rate

B. Heat flux

C. Mass flux

D. Velocity

24. Which (if any) of the following can be discontinuous at a fluid-fluid interface?

A. Shear stress

B. Heat flux

C. Mass flux

D. Velocity

25. Match up the kinematic viscosities of the following materials:

1.	Water	A.	17.0 cSt
2.	Air	B.	1.0 cSt
3.	Glycerine	C.	0.118 cSt
4.	Mercury	D.	650 cSt

26. The Reynolds number is probably the key dimensionless group in fluid mechanics.

a. How is it defined for flow through a pipe?

b. Why does it govern the transition from laminar to turbulent flow?

27. Provide the mathematical representation of the Reynolds number in terms of physical parameters and explain its significance.

28. Briefly explain how a manometer works.

29. Why is the relaxation time important in the study of the rheology of a complex material such as a polymer melt?

30. What is the Weissenberg number? Name a phenomenon discussed in class where it would play a key role.

31. Does Poiseuille's Law govern pressure drop / flow rate relationships for your typical house plumbing system? Briefly (quantitatively) justify your answer.

32. What is the continuum hypothesis, and where does it break down?

33. What is electroosmosis, and where does it come from?

34. What is a yield stress, and name a common fluid that possesses one.

Index Notation Short Answer Questions

The following table has the collected short answer questions focusing on the use of index notation in fluids problems. For ease of extracting individual problems, the number in the typescript compendium is also added as a comment before each "item". The item, down to (or including) the vspace command should be copied into the short answer template for incorporation into an exam or homework assignment.

1. What is the rate of strain tensor (index notation)?

2. Write down the Navier-Stokes equations in index notation, and describe the physical mechanism giving rise to each of the five terms.

3. Write down the continuity equation for a compressible fluid using index notation.

4. Write down the continuity equation for an incompressible fluid using index notation.

5. What is the representation of the deviatoric stress τ_{ij} in terms of the velocity for an incompressible Newtonian fluid? (Index notation, please!)

6. Using index notation, write down the force exerted by the surrounding fluid on an arbitrary object in terms of the stress tensor σ_{ij}.

7. Decompose a general matrix A_{ij} into the sum of a symmetric matrix and an antisymmetric matrix, identifying each.

8. The vorticity is defined as the curl of the velocity. Write down this defining relation using index notation.

9. The vorticity is defined as the curl of the velocity. Using index notation and the symmetry of partial derivatives, show that the divergence of the vorticity is zero.

10. The symmetric part of the rate of strain tensor is usually given the symbol e_{ij}. What is its representation in terms of u_i? (Index notation, please!)

11. Match up the following:

1.	$\delta_{ij}\delta_{ij}$	A.	1
2.	$\delta_{i1}\delta_{j2}$	B.	0
3.	$\delta_{i1}\delta_{i2}$	C.	3
4.	$\delta_{i1}\delta_{i1}$	D.	None of the above

12. What is the most general isotropic second-order tensor?

13. What is the most general second order pseudo tensor characterized by a single director p_i? (Hint: this is the one we used in the notes to show that rotation of a body of revolution with fore-and-aft symmetry due to a force such as gravity was zero at zero Re.)

14. What is the most general relationship for the velocity of a body of revolution falling under some force F_j in terms of its orientation vector p_j at zero Reynolds number? Use index notation.

15. Give an example of:

- A pseudo vector
- A second-order physical tensor discussed in class

CBE 30355 Quizzes

The following quizzes are assigned before and after each lecture of CBE 30355. Note that when administered I used the "randomize answers" approach, and for many of these here they have **not** been randomized, so they would need to be adjusted before being placed in an exam without that option. The quizzes are administered out of Canvas, and the .xml zip file containing the quizzes (as well as the solutions) in Canvas format are available to faculty on request. The detailed contents of each lecture may be found in the course directory at `https://diffusivepress.org/courseware/cbe30355/` as well as the narrated lecture notes.

Lecture 1: What are Fluid Mechanics?

1. What are the units of Young's Modulus of Elasticity? Mark all that are correct!

   (A) m/s^2

   (B) $kg \cdot m/s^2$

   (C) $\dfrac{kg}{m \cdot s^2}$

   (D) m/s

   (E) erg/cm^3

   (F) psi

   (G) Pa

2. If the relaxation time of a material is λ and the shear rate is $\dot{\gamma}$ the product is called the Weissenberg number $Wi = \dot{\gamma}\lambda$. If $Wi \ll 1$ will a material behave more as an elastic solid or a Newtonian liquid?

   (A) Newtonian Liquid

   (B) Elastic Solid

3. Match the following fluids to the correct description:

(1–4) Description	(A–D) Term
1. Water	A. Shear Thickening
2. Mayonnaise	B. Shear Thinning
3. Molten Plastic	C. Bingham Plastic
4. Flexible Body Armor	D. Newtonian Fluid
5. Air	

4. Match the following to their kinematic viscosity:

(1–4) Description	(A–E) Term
1. Air	A. 0.5 cs
2. Water	B. 25 stokes
3. Mercury	C. 15 cs
4. Karo syrup	D. 0.7 cm^2/s
5. Canola oil (look it up...)	E. 0.01 stokes

Lecture 2: The Continuum Hypothesis

1. What is the speed of sound in helium (assume 300K)?

   (A) 1000 m/s

   (B) 3×10^4 cm/s

   (C) 655 mph

   (D) 3×10^{10} cm/s

2. You are studying the force of a class 4 hurricane wind (130 mph) on a building. Is this regarded as compressible or incompressible flow?

   (A) Incompressible

   (B) Compressible

3. You are measuring the viscosity of a suspension of 50 micron diameter spheres in a viscous fluid using a parallel-plate device. The plates are separated by a distance h. What is the minimum value of h for which the measured viscosity will be close to the bulk effective medium value?

   (A) 0.5 mm

   (B) 50 microns

   (C) 5 mm

   (D) 5 cm

4. Pay attention to the wording on this one: You have air in a sealed glass bottle at room temperature. You increase the temperature of the bottle by 30 degrees C. What is the change in the density of the gas in the bottle?

   (A) +10%

   (B) –10%

   (C) –20%

   (D) –0.03%

Lecture 3: Hydrostatics

1. Which of the following are examples of (or causes of) body forces?

   (A) Shear Stress

   (B) Gravity

   (C) Force Causing Electroosmosis

   (D) $\mathbf{J} \times \mathbf{B}$

   (E) Pressure

2. Consider a Newtonian fluid at rest (say, a cup of water sitting on your desk). Which of the following are true? (Mark all that are correct)

   (A) The stress tensor is zero.

   (B) The stress tensor is isotropic.

   (C) The diagonal elements of the stress tensor are negative.

   (D) The stress tensor equals a constant times the identity matrix.

   (E) The off-diagonal elements of the stress tensor are zero.

3. If gravity acts in the z-direction in a fluid at rest, what happens to the normal stress component σ_{xx} using the definition and convention defined in the lecture?

   (A) It increases in the z direction.

   (B) It decreases in the z direction.

   (C) It increases in the x direction.

   (D) It is identically zero as gravity does not act in the x direction.

   (E) It decreases in the x direction.

4. We usually consider the case where gravity and density are constant, but this is not always true. If they aren't constant, will the differential equation governing the pressure distribution for hydrostatics still be correct?

   (A) Yes, because it is valid at every point for a fluid at rest.

   (B) No, because if density and gravity vary this must be taken into account in the derivation.

Lecture 4: Lagrangian and Eulerian Descriptions

1. You are all aware from physics that power requirements (e.g., rate of work) are related to force and velocity. In fluid mechanics we generally work with pressure drop and volumetric flow rate Q. How is power (units of energy/time) related to these quantities? Mark **all** that are correct! Note: U is the fluid velocity, and A is the cross-sectional area.

   (A) Power $= \Delta P \cdot Q$

   (B) Power $= \Delta P \cdot U \cdot A$

   (C) Power $= \Delta P / Q$

   (D) Power $= \Delta P \cdot Q / A$

2. In the lecture we looked at the possibility of using hydrostatics to drive desalinization of sea water using an RO membrane. Unfortunately, the osmotic pressure was way too high to make this work—some 28 atmospheres. RO membranes work very well for this purpose, but only if the pressure is supplied by a pump! Using the results from question 1 (which I'm sure you got right!), determine the minimum power requirement (e.g., ignoring all losses and just worrying about the osmotic pressure) to produce 10 liters/s of fresh water from sea water.

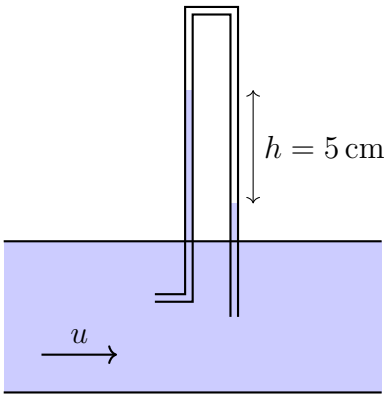
   (A) 28.3 kW

   (B) 28.3 MW

   (C) 28.3 W

   (D) 2.83 kW

3. A couple of decades ago my son built a water tunnel to look at propeller cavitation for a high school science project. He needed to measure the velocity of water in the tunnel, so I suggested he use a pitot tube and an air-water manometer as depicted. The difference in the pressure at the two tubes, one facing into the flow and one perpendicular to the flow, is the dynamic pressure of the flowing water (look up dynamic pressure in the glossary — it's under kinetic energy/volume in the first section). So if he measured an h of 5 cm, what was the fluid velocity?



(A) 100 cm/s

(B) 1 cm/s

(C) 70 cm/s

(D) 7 cm/s

4. Match the following for unsteady flows.

(1–4) Description	**(A–D) Term**
1. A curve which is everywhere tangent to the instantaneous velocity	A. streamline
2. A snapshot of particles which have been released over time at a point in the flow	B. streakline
3. A time-lapsed picture of a tracer in a flow field	C. pathline
4. A mathematical function which is constant along a streamline	D. streamfunction

Lecture 5: Fluids in Motion: The Continuity Equation

1. Water is flowing through a pipe of radius 2 cm with an average velocity of 1 m/s. It goes through a splitter into two parallel pipes with smaller radius of 1 cm each. What is the average velocity of the water in the smaller pipes?

   (A) 50 cm/s

   (B) 1 m/s

   (C) 2 m/s

   (D) 4 m/s

2. **Match the following:**

(1–4) Description	(A–B) Term
1. momentum/volume	A. $\vec{u}$
2. velocity	B. $\rho\vec{u}$
3. volumetric flux	
4. mass flux	

3. A temperature field is measured to have the following conditions:

$$\frac{\partial T}{\partial t} = 1°\text{C/s}, \quad \vec{\nabla} T = (0,\ -0.5,\ 0)\ °\text{C/m}, \quad \vec{u} = (1,\ 2,\ 3)\ \text{m/s}$$

   Given these values, what is the time rate of change of the temperature of a fluid element?

   (A) 0°C/s

   (B) 1°C/s

   (C) 2°C/s

   (D) 3°C/s

4. In Senior Lab you will measure a two-dimensional velocity profile in the x-y plane, such that u_x and u_y are non-zero, but $u_z = 0$ (nothing happens in that direction). If we observe that the derivative $\dfrac{\partial u_y}{\partial y} < 0$, what does this tell us about how u_x varies with x?

   (A) There is no information on u_x.

   (B) u_x must grow with x.

   (C) u_x decreases with increasing x.

   (D) u_x is independent of x.

Lecture 6: Conservation of Mass in Multicomponent Systems

1. First order differential equations occur in many cases beyond engineering, notably in finance! Suppose you start saving for your future kids' college expenses by saving in a 529 plan (e.g., tax free, which simplifies the calculation a lot!) at a rate of \$10/day (e.g., \$3.65k/yr). The annual yield on the plan is 7% (pretty optimistic in the current climate, I'm afraid). How much would be in the plan after 25 years?

   (A) \$248,000
   (B) \$91,250
   (C) \$300,000
   (D) \$24,800

2. You are flushing a tank with volume of 500 liters with fresh water at a rate of 0.5 liters/s. The tank is full at all times. What is the residence time?

   (A) 1000 seconds
   (B) 500 seconds
   (C) 1 ms
   (D) 1 hour

3. If the tank in problem 2 was initially filled with salt water, how long would it take for the concentration to decrease by 95%? (e.g., the time when the concentration is equal to 0.05 of the initial concentration)

   (A) 1000 s
   (B) 3000 s
   (C) 690 s
   (D) 10,000 s

4. You are tasked with measuring the density of a "gold" crown. Using a pan balance scale (e.g., the "justice is blind" type), you balance a container with water in it (this is the ancient equivalent to pushing the "tare" button...). You then suspend the crown in the water on a fine thread, and find that it requires an additional 50 g to balance the scale. You then let the crown settle to the bottom and find that you require a total of 750 g to balance the scale. If the gold crown is actually a mixture of gold (density 19.3 g/cm^3) and silver (density 10.5 g/cm^3), approximately what is the mass of gold in the crown?

   (A) 500 g
   (B) 750 g
   (C) 375 g
   (D) 50 g

Lecture 7: Conservation of Momentum

1. Water is flowing through a horizontal pipe of 5 cm diameter at a flow rate of 1 liter/s. Assuming a flat velocity profile, what is the rate momentum is convected out of the exit of the pipe?

   (A) 0.51 kg m/s^2

   (B) 1 kg/s

   (C) 0.51 kg/s

   (D) 0.25 kg m/s^2

2. You have a 5 cm diameter pipe again, but this time it has a U shaped bend. Draw a control volume around the bend. If the fluid enters the bend with a velocity of 100 cm/s in the x-direction, what is the *net* rate of x momentum convected out of the control volume?

   (A) -3.9 kg m/s^2

   (B) 3.9 kg m/s^2

   (C) -2.0 kg m/s^2

   (D) 2.0 kg m/s^2

   (E) 0 kg m/s^2

3. You have a time-varying velocity with some mean value $\langle u \rangle$ such that $u = \langle u \rangle + u'$. Which of the following is (are) always true? Mark all that are correct.

   (A) $\langle u' \rangle = 0$

   (B) $\langle u^2 \rangle \geq \langle u \rangle^2$

   (C) $\langle u^2 \rangle \leq \langle u \rangle^2$

   (D) $\langle u' \rangle < 0$

   (E) $\langle u' \rangle > 0$

4. Index Notation: Which of the following violate the rules of index notation?

   (A) $\delta_{ii} = 1$

   (B) $A_i = x_i y_i z_i$

   (C) $\delta_{ij} \epsilon_{ijk} = 0$

   (D) $B_i = A_{ij} x_j$

Lecture 8: Cauchy Stress Equation

1. A pitcher is throwing a ball at a target. The ball is either wet clay or bouncy rubber (each with the same mass). Which exerts the greater force on the target?

   (A) The rubber ball.

   (B) The clay ball.

   (C) Neither, since they both travel with the same velocity and have the same mass.

   (D) There is insufficient information to answer the question.

2. You are squirting water at a car with a vane on top, such as was described in the lecture. In one case the vane is flat, while in the other the vane is curved so that the water is reflected exactly backwards in the moving reference frame. Which of the following statements are true?

   (A) The two cars have the same velocity at large times.

   (B) At short times the car with the curved vane travels twice as fast as the one with a flat vane.

   (C) The asymptotic speed of the car with the curved vane is twice as fast as the one with a flat vane.

   (D) It takes twice as long for the car with a flat vane to reach a particular velocity as the one with a curved vane.

3. Power production for a Pelton wheel depends on the rotation rate. If the jet velocity is U_j, and the radius of the wheel is R, what is the optimum angular velocity Ω of the wheel?

   (A) Power is optimum when $\Omega = 0$ because the torque is at a maximum.

   (B) Power is optimum when $\Omega R = U_j$.

   (C) Power is optimum when $\Omega R = \frac{U_j}{2}$.

   (D) Power is optimum when $\Omega = U_j$.

   (E) Power is optimum when $\Omega = \frac{U_j}{2}$.

4. Index Notation: A rod settles through a viscous fluid at zero Re. Which of the following is true?

   (A) The rod will rotate until it is pointing downwards.

   (B) The rod will rotate until it is settling broadside on.

   (C) The rod will not rotate.

   (D) Insufficient information is provided: it depends on the aspect ratio of the rod.

Lecture 9: The Navier-Stokes Equations

1. It is critical to understand the physical origin of each term in the Navier-Stokes equations in order to apply them correctly! Match the numbered terms with the lettered descriptions.

(1–7) Equation Term	(A–G) Description
1. $\rho\dfrac{\partial u_y}{\partial t}$	A. Accumulation of y-momentum
2. $\rho u_x\dfrac{\partial u_y}{\partial x}$	B. Convection of y-momentum in the x-direction
3. $\rho u_y\dfrac{\partial u_x}{\partial y}$	C. Convection of x-momentum in the y-direction
4. $\mu\dfrac{\partial^2 u_x}{\partial y^2}$	D. Diffusion of x-momentum in the y-direction
5. $\mu\dfrac{\partial^2 u_y}{\partial x^2}$	E. Diffusion of y-momentum in the x-direction
6. $-\dfrac{\partial p}{\partial x}$	F. Pressure gradient in the x-direction
7. ρg_y	G. Source of y-momentum due to gravity

2. There are many analogies between momentum and heat transfer, including the basic governing equations! The equation governing heat transfer in a fluid is given by:

$$\rho\widehat{C}_p\left(\frac{\partial T}{\partial t}+u_j\frac{\partial T}{\partial x_j}\right)=k\frac{\partial^2 T}{\partial x_j^2}+\dot{S}$$

Using this, identify the physical meaning of each of the following terms:

(1–6) Equation Term	(A–F) Description
1. $\rho\widehat{C}_p\dfrac{\partial T}{\partial t}$	A. Accumulation of energy
2. $\rho\widehat{C}_p u_x\dfrac{\partial T}{\partial x}$	B. Convection of energy in the x-direction
3. $\rho\widehat{C}_p u_y\dfrac{\partial T}{\partial x}$	C. Term is not in the equation
4. $\rho\widehat{C}_p u_y\dfrac{\partial T}{\partial y}$	D. Convection of energy in the y-direction
5. $k\dfrac{\partial^2 T}{\partial x^2}$	E. Diffusion of energy in the x-direction
6. $\dot{S}$	F. Source of energy per unit volume

3. Which of the following was **not** used in deriving Newton's law of viscosity?

(A) Stress is proportional to the rate of strain.

(B) Viscosity is independent of position.

(C) The fluid is isotropic.

(D) The trace of the deviatoric stress is zero.

4. The Navier–Stokes equations are written in index notation as:

$$\rho\left(\frac{\partial u_i}{\partial t} + u_j\frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} + \mu\frac{\partial^2 u_i}{\partial x_j^2} + \rho g_i$$

What assumption was **not** made in deriving this equation?

(A) The fluid is isotropic.

(B) The viscosity is independent of position.

(C) The viscosity is independent of time.

(D) The Mach number is small.

Lecture 10: Plane Couette Flow, Flow Down an Inclined Plane

1. For gravity driven flow down an incline as was discussed in the lecture, how do the following vary with film depth δ?

(1–3) Description	(A–C) Term
1. $u_{\max}$	A. δ
2. Q/W	B. δ^2
3. $\tau_{yx}\Big\vert_{y=0}$	C. δ^3

2. In the lecture we discussed plane-Couette flow driven by a plate at $y = h$ moving with velocity U_0 in the x-direction. Suppose instead we apply a shear stress F/A to the plate instead (e.g., a different boundary condition). Which of the following are true?

   (A) At long times the velocity profile will still be a linear function of y.

   (B) The shear stress on the upper plate will cause a pressure gradient in the x-direction.

   (C) The velocity of the upper plate will be inversely proportional to the fluid viscosity.

   (D) The upper plate will continuously accelerate, never reaching a steady-state.

   (E) The shear stress condition is applied as a boundary condition on the velocity derivative at the upper plate.

3. A fluid is flowing in the x-direction through a conduit of triangular cross section. Under what circumstances can the flow profile evolve in x (e.g., non-zero x derivative of the velocity)?

   (A) If the fluid is shear thickening.

   (B) If the fluid is shear thinning.

   (C) If the fluid is viscoelastic.

   (D) If it is flowing at finite Mach number (e.g., in a shock tube).

   (E) If there is at least some component of the velocity in the y or z direction.

4. An incompressible fluid is undergoing unidirectional flow in the x-direction. A colleague asserts that because of this, the physical mechanism of diffusion of x-momentum in the x-direction must be identically zero. Is this assertion correct?

   (A) true

   (B) false

Lecture 11: Poiseuille Flow, Couette Flow

1. Mark all of the statements below that are true:

   (A) In the northern hemisphere hurricanes always rotate counter-clockwise.

   (B) In the northern hemisphere bathtub and sink vortices always rotate counter-clockwise.

   (C) Rotation in a hurricane is the result of centrifugal force.

   (D) Centrifugal force appears as a source of momentum in the radial component of the Navier-Stokes equations.

   (E) The coriolis force acts in the theta direction, but requires velocity in the radial direction.

2. A pitcher on the asteroid Vesta is trying to put his fastball into orbit. If the mass of Vesta is 4.47×10^{-5} that of Earth, and the radius is 265,000 meters (0.041 that of Earth), how fast would he have to throw the ball? (Hint: this is easy if you use both the universal gravitational law and the radial component of the Navier-Stokes equations...)

   (A) 260 m/s
   (B) 55 m/s
   (C) 26 m/s
   (D) 68 m/s
   (E) 4260 m/s

3. A water is flowing into a tube of diameter 1 cm and velocity 100 cm/s. You are looking at entrance effects, and want to figure out how far down the tube you need to be before fully developed laminar flow is achieved. About how far down the tube should you make your measurements? (Be careful — this is a little tricky...)

   (A) 350 cm
   (B) 3.5 cm
   (C) Laminar flow is never achieved.
   (D) 25 m
   (E) 25 cm

4. A fluid is undergoing fully developed Poiseuille flow through a tube of radius R. Which of the following statements are true?

   (A) The velocity is proportional to the pressure gradient.

   (B) If I double the radius, I would increase the volumetric flow rate by a factor of four (for the same pressure gradient).

   (C) If I decrease the radius by 2 and keep the flow rate the same, I would increase the pressure drop by a factor of 16.

   (D) The centerline velocity is twice the average velocity.

Lecture 12: Couette Flow, Dynamic Similarity

1. The following deals with Couette flow discussed in the lecture. Mark all of the statements below that are true:

   (A) Taylor vortices will not form if the inner cylinder is stationary and the outer cylinder is rotating.

   (B) The torque on each cylindrical surface in the fluid increases as the radius decreases.

   (C) The shear stress varies inversely with the square of the radius.

   (D) The torque on the inner cylinder is proportional to the fluid viscosity.

   (E) The torque exerted by the outer cylinder is proportional to the angular velocity.

2. What is the inertial scaling for the pressure? Mark all that are correct!

   (A) ρU^2

   (B) $\mu U/L$

   (C) ρU

   (D) $(\mu U/L) \times Re$

3. What is the viscous scaling for the pressure? Mark all that are correct!

   (A) ρU^2

   (B) $\frac{\rho U^2}{Re}$

   (C) $\mu U/L$

   (D) μU

4. In dimensional analysis, which of the following statements are true?

   (A) If correctly scaled, all dimensionless variables in the equation are of $O(1)$.

   (B) If correctly scaled, all dimensionless variables are exactly equal to 1.

   (C) Viscous effects are negligible at large length scales for high Re flow past an object.

   (D) The Froude number is important for inertial flows involving a free surface where gravity plays a role, such as describing bow waves.

Lecture 13: Dynamic Similarity

1. You are trying to model the flow pattern that results when water flows past a complex shape with a length scale of 1 cm and a velocity of 10 cm/s. Because the shape is so small, it is proposed to study it by visualizing the flow past the same shape but with a length scale of 10 cm (10 times larger). Which of the following would work?

   (A) Use a fluid with kinematic viscosity of 1 cs and a velocity of 1 cm/s.

   (B) Use a fluid with kinematic viscosity of 1.0 stokes and a velocity of 1 m/s.

   (C) Use a fluid with kinematic viscosity of 10 cs and a velocity of 10 cm/s.

   (D) Use a fluid with kinematic viscosity of 1 cs and a velocity of 10 cm/s.

   (E) You can't model a system with low viscosity.

2. You are modeling the drainage vortex of a tank (characteristic length scale of 5 meters) of a viscous liquid (kinematic viscosity of 10 stokes) with water using a scale model. What should be the length scale of the model to preserve strict dynamic similarity?

   (A) 5 cm

   (B) 5 mm

   (C) 1 m

   (D) 10 cm

3. When you do the experiment in the model system in question 2, you observe the tank to drain in 100 s. Assuming that strict dynamic similarity is preserved, how long would it take for the full scale tank to drain?

   (A) 1000 s

   (B) 100 s

   (C) 10 s

   (D) 10,000 s

   (E) There is not enough information provided to determine this.

4. You are evaluating ship hull designs using scale models and a towing tank. Which is the most important dimensionless group to preserve?

   (A) The Reynolds Number

   (B) The Froude Number

   (C) Neither Re or Fr are necessary to preserve to model a ship hull.

   (D) Both Re and Fr must be preserved to get a good result.

Lecture 14: Dimensional Analysis

1. You are studying the degree to which a sphere sinks into a thick elastic material. This distance h is a function of the applied force F, the radius of the sphere a, and the Young's modulus of elasticity of the material E. How many dimensionless groups are involved in the problem?

   (A) 0 (C) 2 (E) 4

   (B) 1 (D) 3

2. One of the more complicated dimensional analysis problems my students have studied over the years was the motion of a sphere in a tube undergoing oscillatory flow. The tidal displacement of the sphere (what we were measuring) is given by Δx, the tidal volume of the fluid flow is ΔV, the radius of the sphere is a, the surface roughness of the spheres was $\varepsilon_s a$, the density of the sphere is ρ_s, and the radius of the tube is R. The displacement is a further function of gravity (the spheres weren't neutrally buoyant), the frequency of oscillation ω, and the properties of the fluid (incompressible Newtonian liquid). Based on this, how many dimensionless groups are involved in the problem?

   (A) 10 (C) 8 (E) 6

   (B) 9 (D) 7 (F) 5

3. The horrific explosion in Beirut on 8/4/20 actually rivaled a nuclear explosion in its intensity. An examination of the video footage reveals that the shock wave reached a radius of 400 m after about 1 second. If 1 kT of high explosives has a yield of 4.18×10^{12} J, what was the approximate yield of the Beirut blast?

   (A) 0.1 kT (D) 20 kT

   (B) 3 kT (E) There is not enough information provided to determine this.

   (C) 15 kT

4. Stokes Law (drag on a sphere) is a fundamental result for low Re hydrodynamics (one might argue that it is *the* fundamental result!). The dependence on the parameters of the problem can be obtained via dimensional analysis, and the usual result is that the remaining "unknown constant" should be $\mathcal{O}(1)$. If we normalize the force F with the shear stress (e.g., the viscosity μ times the characteristic shear rate U/a) and the surface area of the sphere, what does the constant in Stokes Law become?

   (A) 6π (C) 1

   (B) 4π (D) 1.5

Lecture 15: Lubrication Flows

1. In 2-D lubrication flows (x-y plane where y is the thin direction), which of the following are true?

   (A) The flow is unidirectional in the x-direction.

   (B) The flow in the y-direction is small, but non-zero.

   (C) The pressure gradient in the x-direction is negligible.

   (D) The pressure gradient in the y-direction is negligible.

2. In a defined flow rate lubrication problem such as the squeeze flow examined in the lecture, how many conditions are necessary to determine the velocity profile and pressure gradient?

   (A) 0

   (B) 1

   (C) 2

   (D) 3

   (E) 4

3. You have solved for the force necessary to pull a disk away from a plane with constant velocity, but are instead interested in the force resisting its approach. What do you do?

   (A) Resolve the problem from the beginning using the new boundary condition.

   (B) Use the existing solution, but just change the sign of the force.

   (C) Give up.

4. The time necessary for a disk to detach from a plane under the action of a constant force in the lubrication limit depends on its initial separation. If we double this initial separation, what happens to the detachment time?

   (A) It doubles.

   (B) It is reduced by 2x.

   (C) It is reduced by 4x.

   (D) It is reduced by 8x.

Lecture 16: Reynolds Lubrication Equation, Streamfunction

1. In 2-D lubrication flows (x-y plane where y is the thin direction), which of the following are true?

   (A) The pressure gradient in the y-direction is negligible.

   (B) The pressure gradient in the x-direction is negligible.

   (C) The lubrication force can be estimated from scaling analysis.

   (D) The velocity in the x-direction is much larger than the velocity in the y-direction.

   (E) The velocity in the y-direction is determined from continuity rather than the y-momentum equation.

2. In the Reynolds Lubrication Equation, what causes there to be a non-zero pressure gradient in the x-direction?

   (A) Variation of the gap width h with x.

   (B) Variation of the tangential (x-direction) velocity of the boundary with x.

   (C) Relative velocity of the two surfaces in the y-direction (e.g., h as a function of t).

   (D) Pressure variation outside the lubrication gap.

3. You are trying to lubricate a 100 kg load sitting on a plate which is 100 cm long (in the direction of motion) and 20 cm wide (e.g., "into the board"). You want to use the sliding block approach. You can force the plate to be at any angle you like, and the lubricating fluid has a viscosity of 1 poise. The problem is that the surface you are dragging it across has a roughness of typical magnitude of 200 μm. Based on scaling analysis, what is the order of magnitude of the velocity at which you can expect lubrication to occur?

   (A) 0.0001 m/s

   (B) 0.01 m/s

   (C) 1 m/s

   (D) 10 m/s

4. You are examining the sliding block example in question 3, and (using the same geometry), move it in the negative x-direction. What happens?

   (A) The sign of the lubrication force reverses and the plate comes into contact with the surface.

   (B) The lubrication force remains the same and the plate remains lubricated.

   (C) There is insufficient information to answer the question.

Lecture 17: Flow Near a Wiper

1. Which of the following are true about the streamfunction ψ for 2-D flow at low Re?

   (A) The streamfunction is constant along a streamline.

   (B) The pressure gradient in the y-direction is negligible.

   (C) For a two-dimensional flow, the velocity in the x-direction equals the derivative of ψ with respect to x.

   (D) The velocity in the x-direction is much larger than the velocity in the y-direction.

   (E) The magnitude of ψ equals the deflection of a thin elastic plate driven by deformation at the boundaries.

2. Which of the following are true about the Taylor wiper problem?

   (A) The value of the streamfunction is the same on both the bottom surface and the knife edge.

   (B) The pressure in the corner as $r \to 0$ is an integrable singularity.

   (C) The streamfunction is proportional to r everywhere.

   (D) The characteristic equation for determining the angular dependence of the streamfunction has repeated roots.

3. The general separable solution to the Biharmonic equation in the cylindrical geometry is $\psi \sim r^\lambda f(\theta)$ where λ is usually 0, 1, or 2. If, instead of a wiper blade, we have a closing wedge such that u_θ at $\theta = \theta_0$ is $-\Omega r$, what should λ be?

   (A) 0

   (B) 1

   (C) 2

   (D) There is insufficient information to answer the question.

4. A classic streamfunction problem is the drainage of a viscous fluid through a thin slit at the bottom of the vessel. We can solve this using the same approach as was used in the Taylor wiper problem (e.g., separable in cylindrical coordinates). What should λ be for this problem?

   (A) 0

   (B) 1

   (C) 2

   (D) There is insufficient information to answer the question.

Lecture 18: Flow Past a Sphere

1. Which of the following are true about flow past a sphere at zero Re?

   (A) The streamfunction satisfies the biharmonic equation.

   (B) The contribution of the pressure integral to the drag on a sphere is negligible relative to the integral of the deviatoric stress.

   (C) The drag is proportional to the fluid velocity.

   (D) The disturbance velocity produced by a sphere decays as $O(1/r)$ far from a sphere.

   (E) The drag on a sphere is proportional to the sphere radius.

2. The minimum dissipation theorem is very useful for understanding the behavior of flows at zero Re. Which of the following are true?

   (A) The modification of the velocity distribution caused by inertia always increases the drag on an object.

   (B) If you double the characteristic length scale of an object, you quadruple the surface area. Thus, you should quadruple the drag at zero Re.

   (C) If you have a fuzzy sphere and shave off the fuzz, you reduce the drag.

   (D) If you have a fuzzy sphere and streamline it by encasing it in a smooth shell, you reduce the drag.

3. A 1 μm diameter gold microsphere (density of 19.3 g/cm^3) has a monolayer of 100 nm polymer brushes attached to its surface. What is a good estimate of its sedimentation velocity in water?

   (A) 8.3 μm/s

   (B) 11 μm/s

   (C) 0.6 cm/s

   (D) There is insufficient information to answer the question.

4. It is possible to construct the solution to any problem at zero Re from a linear combination of harmonic functions.

   (A) true

   (B) false

Lecture 20: Bernoulli's Equation

1. Which of the following were used in deriving Bernoulli's equation?

   (A) The curl of the velocity is zero.

   (B) The flow is inviscid.

   (C) The Reynolds number is very low.

   (D) The flow is steady.

   (E) Gravity is negligible.

2. Water is flowing at high Re through a pipe of diameter D with velocity U. It passes through a smooth contraction which reduces the diameter by a factor of 2. What happens to the velocity?

   (A) The velocity doubles.

   (B) The velocity increases by a factor of 4.

   (C) The velocity decreases by a factor of 2.

   (D) There is insufficient information to answer the question.

3. Water is flowing at high Re through a pipe with diameter D and velocity U, and passes through a smooth contraction with a diameter of $D/2$. What happens to the pressure?

   (A) The pressure decreases by $\frac{7}{2}\rho U^2$.

   (B) The pressure increases by $\frac{7}{2}\rho U^2$.

   (C) The pressure decreases by $\frac{15}{2}\rho U^2$.

   (D) The pressure decreases by $8\rho U^2$.

   (E) There is insufficient information to answer the question.

4. It is possible to use the measurement of the pressure decrease in a smooth contraction to calculate the fluid velocity (called a venturi meter). If the pressure in a contraction where the diameter decreases by a factor of 2 is 1000 Pa and the fluid is water, what is the fluid velocity? (Assume that there are no losses, and so Bernoulli's equation applies)

   (A) 37 cm/s

   (B) 54 cm/s

   (C) 11.5 m/s

   (D) 1.6 m/s

Lecture 21: Potential Flow Past a Cylinder

1. You are solving the Euler flow equations for uniform flow past an object. Which of the following are true?

   (A) The velocity equals the gradient of a scalar.

   (B) The vorticity is zero.

   (C) The no-slip condition yields the velocity at the surface.

   (D) The normal velocity at the surface of the object is zero.

   (E) Viscosity is unimportant at length scales comparable to the size of the object.

2. A friend's family has a small beach house, the kind which is up on stilts to handle tidal surges that sticks up into the wind. A tropical storm is bearing down on the house and your friend wants to know whether it is necessary to board their big picture window. If the window pane measures 4' $\times$ 6', about what is the expected force on the window? Assume a wind speed of 50 mph and take the pressure inside the house to be atmospheric.

   (A) 688 Newtons

   (B) 1400 Newtons

   (C) 38 Newtons

   (D) 243 Newtons

3. Your friend opens the door on the back side of the house (e.g., away from the wind and the ocean). By what factor should the force on the window facing the wind and ocean be increased?

   (A) It should increase by 1.5x.

   (B) It should more than double.

   (C) It should decrease by 2x.

   (D) It should be unchanged, as the window on the back side doesn't affect the flow on the front.

4. The drag on a cylinder is predicted to be zero at high Re. Why is this?

   (A) Because that's what D'Alembert found (paradoxically).

   (B) Because of Stokes' Paradox.

   (C) Because the Euler flow solution says that pressure should recover on the back side of the cylinder.

   (D) Because the integral of an odd function over an even domain is zero.

   (E) Because the no-slip condition and boundary layer separation has been neglected.

Lecture 21: Prandtl Boundary Layer Equations

1. Ethylene glycol (kinematic viscosity of 10 cs) is flowing at a velocity of 100 cm/s past a flat plate of length 10 cm. What is the approximate thickness of the boundary layer at the end of the plate?

   (A) 1 mm
   (B) 1 cm
   (C) 10 cm
   (D) 100 µm
   (E) 0

2. One method that has been used to control boundary layers is suction: literally "sucking out" the region of slow flow next to a plate (or wing) by imposing a vertical velocity at the surface (e.g., sucking fluid out through a porous plate). In order for this to work, the suction velocity should be of the same order as the vertical velocity in the boundary layer (e.g., what you get from scaling). For the conditions described in problem 1, about how large should this suction velocity be?

   (A) 1 cm/s
   (B) 1 mm/s
   (C) 100 µm/s
   (D) 10 cm/s

3. In flow over a cylinder, the characteristic pressure gradient in the tangential direction along the surface should scale as $\rho U^2/a$ and is a significant source (or sink) of momentum. Suppose we have water flowing past a cylinder of radius 1 cm at 100 cm/s. In the Prandtl boundary layer equations we neglect the pressure variation across the thin boundary layer even though we know from the inviscid flow solutions that it does vary in the radial direction. If we wanted to include this effect, about how large should it be relative to the tangential pressure gradient?

   (A) About 0.01%
   (B) About 0.1%
   (C) About 1%
   (D) About 10%

4. Students have been known to confuse reductions arising from the defining of a stream function for 2-D flows with those achieved via application of Morgan's Theorem. Which of the following statements are true?

   (A) Recasting a problem in terms of a streamfunction always increases the order of the resulting differential equation.

   (B) Developing a self-similar solution using Morgan's Theorem always increases the order of the resulting differential equation.

   (C) Recasting a problem in terms of a streamfunction reduces the number of independent variables the problem depends on.

(D) Developing a self-similar solution using Morgan's Theorem reduces the number of independent variables the problem depends on.

(E) If you can successfully apply Morgan's Theorem to a problem governed by a PDE in x and y, you can obtain a transformed problem governed only by an ODE.

Lecture 22: Flow Past a Flat Plate: The Blasius Equation

All four questions today deal with the classic problem of mass transfer from a membrane in the presence of a shear flow. In this problem we are looking at the concentration distribution in a boundary layer near the membrane, where the concentration at the surface is at an equilibrium concentration c_0. Mass transfer boundary layers are really thin, so this applies equally well to a flat membrane or to the inside of a hollow fiber membrane (e.g., it looks like a plane on the length scale of the boundary layer). The fluid coming in is assumed to have a concentration $c = 0$. The velocity (in the x-direction) is the simple shear flow $u_x = \lambda y$ where λ is the shear rate and y is the distance away from the wall. The concentration distribution is governed by the equation (ignoring diffusion in the x direction) by:

$$\lambda y \frac{\partial c}{\partial x} = D \frac{\partial^2 c}{\partial y^2}$$

where D is the mass diffusivity (analogous to the kinematic viscosity, but for mass rather than momentum).

The boundary conditions are:

$$c|_{y=0,\, x>0} = c_0$$

and

$$c|_{y\to\infty} = c|_{x<0} = 0$$

We are interested in calculating the total rate of mass transfer from the membrane (per unit width). This is calculated from the integral of the mass flux at the surface, which in turn depends on the concentration gradient you obtain from solving the mass transport equation. The mass flux is:

$$N_y|_{y=0} = -D \left.\frac{\partial c}{\partial y}\right|_{y=0}$$

And the total rate of mass transfer Q/W is:

$$Q/W = \int_0^L N_y|_{y=0}\, dx$$

We render the variables dimensionless as:

$$x^* = \frac{x}{L}; \quad y^* = \frac{y}{\delta}; \quad c^* = \frac{c}{c_0}$$

1. By scaling the equation and boundary conditions, how does the characteristic scale of the boundary layer thickness δ vary with L?

   (A) $\delta \sim L$

   (B) $\delta \sim L^{1/2}$

   (C) $\delta \sim L^{1/3}$

   (D) $\delta \sim L^{1/4}$

2. For the problem described above, how does the scale of the mass flux at the wall $N_y|_{y=0}$ depend on L? Hint: render it dimensionless too!

   (A) $N_y \sim L$

   (B) $N_y \sim L^{-1/4}$

   (C) $N_y \sim L^{-1/3}$

   (D) $N_y \sim L^{1/2}$

3. Now we look at the dimensionless problem and show it admits a self-similar solution via simple affine stretching. If we let:

$$c^* = A\bar{c}; \quad x^* = B\bar{x}; \quad y^* = C\bar{y}$$

   What are the restrictions on A, B, and C in canonical form?

   (A) $A = 1; \quad \dfrac{C}{B^{1/3}} = 1$

   (B) $A = 1; \quad \dfrac{C^3}{B} = 1$

   (C) $\dfrac{AC}{B} = 1; \quad \dfrac{A}{C^2} = 1$

   (D) $\dfrac{A}{B^{1/2}} = 1; \quad \dfrac{C}{B^{1/2}} = 1$

4. Scaling is an extremely useful way of estimating the magnitude of quantities! For the problem above water is flowing over the membrane with shear rate $\lambda = 10\,s^{-1}$. If the concentration at the surface of the membrane is $c_0 = 0.001\,\mathrm{g/cm^3}$ and the diffusion coefficient is $D = 10^{-6}\,\mathrm{cm^2/s}$, what is the approximate mass transfer Q/W for a membrane 10 cm long?

   (A) 10^{-8} g/(cm s)

   (B) 10^{-7} g/(cm s)

   (C) 10^{-6} g/(cm s)

   (D) 10^{-5} g/(cm s)

   (E) There is insufficient information to make an estimate.

Lecture 23: Complex Boundary Layer Flows

1. Consider flow past a flat plate where the inner limit of the Euler flow is given by $u_{EF} = \lambda x^{\alpha}$. What is the resulting pressure gradient in the x-direction in the boundary layer?

   (A) $-\rho\lambda x^{\alpha}$

   (B) $\rho\lambda x^{\alpha}$

   (C) $-\rho\alpha\lambda^2 x^{\alpha}$

   (D) $-\rho\alpha\lambda^2 x^{2\alpha-1}$

   (E) $\frac{1}{2}\rho\lambda^2 x^{2\alpha}$

   (F) There is no pressure gradient in the boundary layer.

2. Depending on the geometry the pressure gradient in the boundary layer can be positive, negative, or zero. If the inner limit of the Euler flow velocity is increasing along the surface, which of the following are true?

   (A) The pressure gradient is negative.

   (B) The pressure gradient is zero.

   (C) The pressure gradient is positive.

   (D) The pressure gradient is favorable.

   (E) The pressure gradient is unfavorable.

   (F) The pressure gradient retards the growth of the boundary layer.

   (G) The pressure gradient promotes the growth of the boundary layer.

   (H) The pressure gradient has no affect on the growth of the boundary layer.

3. The drag coefficient for flow past a sphere experiences an abrupt drop at a Reynolds number of 3.4×10^5. Why does this occur?

   (A) The boundary layer becomes turbulent, injecting momentum into the boundary layer and delaying separation.

   (B) The boundary layer becomes turbulent, promoting boundary layer separation and preventing pressure recovery.

   (C) Stokes flow becomes valid, and by the minimum dissipation theorem yields a lower drag.

   (D) The Euler flow solution is no longer valid.

4. Why are there dimples on a golf ball? Mark all that are correct.

   (A) Dimples promote turbulence in the boundary layer on a golf ball, promoting pressure recovery and reducing drag.

(B) Dimples promote interaction between rotation and the boundary layer, leading to boundary layer deflection and Magnus lift.

(C) Dimples prevent the boundary layer from becoming turbulent.

(D) It is the result of Stokes Law drag.

(E) Dimples have no effect on the flight of a golf ball, but are a good advertising gimmick.

Lecture 24: Integral Boundary Layer Equations

1. Which of the following statements are correct about the von Karman Integral Momentum Balance?

   (A) The momentum balance applies only to flat plates.

   (B) The momentum balance provides a convenient and accurate way to measure the shear stress at a surface.

   (C) The momentum balance would have to be modified if there is suction on the plate ($v \neq 0$ at $y = 0$).

   (D) The momentum balance works only if there is uniform flow outside the boundary layer.

2. Water is flowing past a flat plate at a velocity of 1 m/s. How far down the plate is the laminar boundary layer solution (e.g., the Blasius equation) valid?

   (A) 30 cm

   (B) 30 m

   (C) 3 cm

   (D) 30 microns

   (E) The Blasius solution has no validity for this problem.

3. Which of the following are true about the displacement thickness δ^*?

   (A) It is dimensionless.

   (B) It is the distance streamlines outside the boundary layer are displaced by the wedge of slow fluid inside the boundary layer.

   (C) It is used in calculating the shear stress at the surface, but only plays a role for accelerating or decelerating flows.

   (D) It is a lot easier to measure experimentally than a velocity derivative.

4. Which of the following are true about the momentum thickness Θ?

   (A) It is dimensionless.

   (B) It is a measure of the amount of momentum sucked out of the boundary layer by shear stress at the wall.

   (C) The drag on a flat plate (the Blasius problem) is proportional to Θ.

   (D) It is a lot easier to measure than the velocity derivative at the surface.

Lecture 25: Friction Factors

1. Which of the following statements are correct about the Fanning Friction Factor f_f?

   (A) It equals the wall shear stress in a pipe rendered dimensionless with the kinetic energy per unit volume of the fluid.

   (B) It is inversely proportional to Re for laminar flow.

   (C) It is proportional to L/D for long pipes.

   (D) It is independent of Re for sufficiently rough pipes.

2. Water is flowing through a pipe with a wall shear stress of 25 dynes/cm^2. About how large do roughness elements have to be to promote turbulence under these conditions?

   (A) 0.014 cm

   (B) 1.4 cm

   (C) 0.002 cm

   (D) 1 m

   (E) There is insufficient information to answer the question.

3. Water is flowing through a 5 cm ID pipe at a velocity of 1 m/s. The pipe is 20 m long, has 5 90º standard elbows, a wide open globe valve, and includes a sudden expansion (or just the acceleration of the fluid). Order these contributions to the pressure drop.

(1–4) Description	(A–D) Term
1. pipe friction	A. largest
2. acceleration of fluid	B. second largest
3. 5 elbows	C. second smallest
4. globe valve wide open	D. smallest

4. You are pumping water to a total head of 70 m at a flow rate of 15 liters/s. You propose to use the pump HH-80. What would be the efficiency of the pump at the operating condition?

   (A) 47%

   (B) 50%

   (C) 45%

   (D) 24%

Lecture 26: Piping Networks & Review

1. Match the following analogies between piping networks and electrical networks.

(1–4) Description	(A–D) Term
1. total head	A. voltage drop
2. flow rate	B. battery
3. pump	C. current
4. head losses	D. voltage
5. friction factors	E. resistance
6. tee fitting	F. Ohm's Law
7. Bernoulli's Equation	G. electrical junction

2. Water is flowing through a 10cm diameter pipe at a flow rate of 70 liters/s. What is the value of f_f for this flow rate?

   (A) 0.0030
   (C) 0.0081
   (B) 0.0026
   (D) 0.0046

3. You are siting a pump, trying to avoid cavitation. The pump is being used to pump water from a pond at a flow rate of 20 liters/s through a 10cm ID pipe. There is 10 meters of pipe leading to the pump as well as a sudden contraction (K value of 0.45) and the acceleration of the fluid. If the NPSHR is 4 meters, where can the pump be sited?

   (A) No more than 5m above the pond.
   (B) No more than 6m above the pond.
   (C) At least 2m below the level of the pond.
   (D) At least 5m below the level of the pond.

4. You are designing a gravity fed overflow drain for a tank containing water, which must accommodate a flow rate of 2 liters/s. The total head driving the flow is 2 meters. If the fitting losses consist of a sudden contraction and an elbow (as well as the acceleration of the fluid!) and the pipe length is 10 meters, what is the minimum diameter pipe you can get away with?

   (A) 3.4 cm
   (C) 10.6 cm
   (B) 5.2 cm
   (D) 1.4 cm

Lecture 27: Turbulence

1. What is the Reynolds Stress? Mark all that are correct!

   (A) The momentum flux arising from turbulent velocity fluctuations.

   (B) $-\langle \rho u' u' \rangle$

   (C) Proportional to the time average of the square of velocity fluctuations, multiplied by the density.

   (D) Orders of magnitude greater than laminar stresses for turbulent flow.

2. For turbulent flow through a pipe, what is true about the eddy viscosity? Mark all that are correct!

   (A) It is proportional to the density.

   (B) It is proportional to the shear rate.

   (C) It is proportional to the square of the distance from the wall.

   (D) In the turbulent core it is orders of magnitude greater than the dynamic viscosity.

3. A key result of the Prandtl mixing length theory model for turbulent flow is that the time-averaged velocity in the turbulent core varies logarithmically with distance from the wall (e.g., the "Law of the Wall").

   (A) true

   (B) false

4. Water is flowing with a velocity of 200 cm/s through a 5 cm ID pipe. What fraction of the cross-sectional area of the pipe is occupied by the turbulent core?

   (A) 99%

   (B) 95%

   (C) 50%

   (D) None — the flow is laminar.

CBE 30355 Problem Sets

The following problem sets are used in CBE 30355. Each problem set has a summary of "what students should learn from the problem set" which is part of what is distributed in the assignment (it lets the students know why I'm asking them to solve these particular problems). The individual problem files have Chat generated summaries which are not part of the problem sets. The implementation of the problem sets may be seen in the LaTeX script `ahomeworktemplate.tex`. Some of these problems are identical to exam questions, and some also have solutions. The solutions, when they exist, may be found in the course directory at `https://diffusivepress.org/courseware/cbe30355/`.

What You Should Learn from Problem Set 1

Problem 1: This problem is asked to get you to do several things.

- **a.** To remember how to sequentially integrate an expression (including the tricky bits associated with the cylindrical geometry).
- **b.** To apply boundary conditions to determine the unknown constants of integration.
- **c.** To get you to open your Calc II text if you've forgotten how to do this stuff...

Problem 2: Again, several things should be learned:

- **a.** Practice with inner products and cross products.
- **b.** Recognizing what happens when vectors are perpendicular or parallel.

Problem 3: This problem gives you practice with operators we will use extensively in this course:

- **a.** To discriminate between gradient, divergence, Laplacian, and curl.
- **b.** To think about how all this works with vectors and tensors.

Problem 4: This problem deals with combinations of operators:

- **a.** Practice with divergence and curl of a vector field.
- **b.** Ultimately, this is most easily approached using index notation and the concepts of symmetric and anti-symmetric tensors...

Problem 5: This problem deals with transients:

- **a.** Figure out how to estimate viscosities from the time to approach steady-state.
- **b.** Determine how the characteristic time depends on the geometry and boundary conditions.
- **c.** Start building your intuition on fluid mechanics problems.

Problem 6: A problem you really only want to do if you read ahead in the notes on index notation...

Velocity Distribution in a Tube under Pressure Gradient

Subject Tags: fluid mechanics; pressure-driven flow; pipe flow

Difficulty: Moderate

Estimated Time: 20 minutes

Summary: This problem asks the student to determine the velocity profile $u(r)$ of a viscous fluid flowing through a cylindrical tube, given a second-order differential equation and boundary conditions. The solution involves direct integration and application of boundary conditions. The physical meaning of the pressure gradient and viscosity is emphasized.

Skills Tested:

- Solving second-order differential equations
- Applying boundary conditions
- Interpreting physical parameters in fluid mechanics

Velocity Distribution in a Tube under Pressure Gradient

The axial velocity u of fluid in a tube of radius a is governed by the second-order differential equation:

$$\frac{1}{r}\frac{d}{dr}\left(r\frac{du}{dr}\right) = \frac{1}{\mu}\frac{\Delta P}{L}$$

with boundary conditions:

$$u|_{r=0} = \text{finite}, \quad u|_{r=a} = 0$$

The expression $\Delta P/L$ is the pressure drop per unit length of the tube (and is negative for positive flow), and is a constant. μ is the fluid viscosity and is also constant.

What is the velocity distribution $u(r)$ in the tube as a function of r?

(Hint: It's directly integrable!)

Vector Dot and Cross Products

Subject Tags: vector calculus; dot product; cross product

Difficulty: Simple

Estimated Time: 10 minutes

Summary: This problem involves straightforward evaluation of dot and cross products between 3D vectors. Students must apply standard vector algebra operations to compute the results.

Skills Tested:

- Evaluating dot products
- Evaluating cross products
- Using vector notation

Vector Dot and Cross Products

Calculate the following quantities. (Note: $\times$ denotes the cross-product and $\cdot$ the dot product.)

a. $(1, 0, 1) \cdot (1, 0, -1)$

b. $(1, 1, 1) \cdot (0, 1, 0)$

c. $(1, 1, 1) \times (2, 2, 2)$

Vector Operators on Scalar and Vector Fields

Subject Tags: vector calculus; gradient; divergence; curl; Laplacian

Difficulty: Moderate

Estimated Time: 20 minutes

Summary: This problem involves evaluating several fundamental vector calculus operations, including gradient, divergence, curl, and Laplacian, for given scalar and vector fields. It reinforces understanding of differential operators in three-dimensional space.

Skills Tested:

- Applying the gradient operator to scalar fields
- Computing divergence and curl of vector fields
- Evaluating scalar and vector Laplacians

Vector Operators on Scalar and Vector Fields

For the scalar potential function $\phi = (x+y+z)^2$ and the velocity vector field $\mathbf{u} = (z,\, y^2,\, x)$, calculate the following vector quantities:

a. $\nabla\phi$; $\quad \nabla \cdot \mathbf{u}$

b. $\nabla^2\phi = (\nabla \cdot \nabla)\phi$; $\quad \nabla^2\mathbf{u}$

c. $\nabla \times \mathbf{u}$

where the boldface operator $\nabla = \left(\dfrac{\partial}{\partial x},\, \dfrac{\partial}{\partial y},\, \dfrac{\partial}{\partial z}\right)$.

Divergence of the Curl of a Vector Field

Subject Tags: vector calculus; curl; divergence; identity

Difficulty: Moderate

Estimated Time: 15 minutes

Summary: This problem asks the student to prove a fundamental identity in vector calculus: that the divergence of the curl of any vector field is zero. This identity has important implications in fluid mechanics, where it implies that vorticity is a solenoidal (divergence-free) field.

Skills Tested:

- Applying vector calculus identities
- Understanding the properties of the curl and divergence
- Interpreting physical meaning of vorticity in fluid mechanics

Divergence of the Curl of a Vector Field

Prove that for an arbitrary vector $\mathbf{u}$:

$$\nabla \cdot (\nabla \times \mathbf{u}) = 0$$

In fluid mechanics, where $\mathbf{u}$ is the velocity vector, this is equivalent to saying that the vorticity—the curl of the velocity—is a solenoidal vector field (divergence free). It is very useful in manipulating the equations of motion, particularly at high Reynolds numbers.

Transient Simple Shear Flow

Subject Tags: unsteady flow; plate motion; viscosity estimation

Difficulty: Moderate

Estimated Time: 20 minutes

Summary: This problem involves analyzing the transient behavior of a fluid between two plates when the upper plate is suddenly set into motion. Students are asked to estimate the fluid's kinematic viscosity from the time to steady-state, and assess how changes in force and geometry affect this timescale.

Skills Tested:

- Estimating kinematic viscosity from physical data
- Applying unsteady flow concepts
- Using dimensional reasoning and scaling arguments

Transient Simple Shear Flow

Two plates are separated by a distance of 0.5 mm. A tangential force is applied to the upper plate (in the same manner as was described in class) and it begins to move, eventually reaching a steady velocity. Answer the following:

a. If the steady-state is achieved in about 1 second, estimate the kinematic viscosity of the fluid.

b. If the force is doubled, how does the time to steady-state change? (Trick question...)

c. If the gap between the plates is doubled, how does the time to steady-state change?

Vector Identity Involving Double Curl

Subject Tags: vector calculus; vector identity; curl; Laplacian; index notation

Difficulty: Advanced

Estimated Time: 25 minutes

Summary: This optional (ungraded) problem challenges students to prove a fundamental identity in vector calculus involving the double curl of a vector field. The problem introduces index notation as a useful tool for general tensor calculus in transport phenomena.

Skills Tested:

- Applying vector calculus identities
- Manipulating expressions using index notation
- Interpreting physical meaning of differential operators

Vector Identity Involving Double Curl

This is completely optional (and not for credit—solve only if you like puzzles):

Prove the following vector identity for an arbitrary vector $\mathbf{u}$:

$$\nabla \times (\nabla \times \mathbf{u}) = \nabla (\nabla \cdot \mathbf{u}) - \nabla^2 \mathbf{u}$$

Hint: I usually solve this problem using index notation, which is very useful for describing advanced transport problems. Detailed notes on index notation are available through the class website. We will go over index notation in our first few in-class sessions.

What You Should Learn from Problem Set 2

Problem 1: This problem is asked to get you to do several things.

- **a.** To open the book and get practice looking up and using correlations.
- **b.** To recognize that you can calculate the viscosity of air (or any low density gas) from simple collision integral arguments.
- **c.** To get practice cutting and pasting data from charts and graphically comparing data to models/correlations.

Problem 2: Again, several things should be learned:

- **a.** To recognize (and memorize) that for simple fluids such as oils the log of the viscosity can be well correlated with a polynomial in the inverse of the absolute temperature.
- **b.** To refresh your memory on how you do such a linear regression curve fit.
- **c.** To get practice using such a correlation to calculate quantities such as the temperature coefficient.
- **d.** To recognize that, for a viscous liquid, the viscosity changes by about 4% to 7% per degree C — this is a pretty large variation, and is why you can't clean viscous liquids out of a vessel with cold water!
- **e.** To get practice with the mixing rule for simple fluids (doesn't work for more complicated mixtures).

Problem 3: Lots of very important things:

- **a.** To refresh your memory on how to use first-year calculus to change/eliminate variables.
- **b.** To combine several equations describing different physical phenomena (mass conservation, hydrostatics, gravitation, adiabatic gas law) together to get a mathematical description of a pretty off-the-wall problem.
- **c.** Your first practice scaling equations to "figure out the answer before you start." This will be a recurring theme in the class, and something you will find useful all your careers.
- **d.** Refresh your memory on how to solve systems of first-order ODEs numerically.

Problem 4: More hydrostatics:

- **a.** Connect the mathematics of hydrostatics and buoyancy to a simple problem.
- **b.** Practice making the "flat Earth limit" approximation that is often used to (drastically) simplify calculations.
- **c.** Get a more physical understanding of how pressure distributions are connected to forces.

Estimating Viscosity of Air Using Chapman-Enskog Theory

Subject Tags: gas viscosity; Chapman-Enskog; temperature dependence; air; physical properties

Difficulty: Advanced

Estimated Time: 25 minutes

Summary: Use the Chapman-Enskog theory to compute the dynamic viscosity of air over a wide temperature range. Results are compared with reliable tabulated data for validation and insight into the accuracy of the kinetic theory model.

Skills Tested:

- Applying Chapman-Enskog theory for gases
- Using tabulated molecular data and correlations
- Comparing theoretical predictions with empirical data
- Plotting and interpreting physical property trends

Estimating Viscosity of Air Using Chapman-Enskog Theory

Compute the viscosity of air from 200 K to 500 K at 1 atm pressure. The Chapman-Enskog equation described in Chapter 1 of BS&L (Eq. 1.4-14) and the data in Tables E.1 and E.2 are useful here (note the correlation at the bottom of Table E.2)!

Compare your result graphically with the data from the following website (the relevant table is located near the bottom of the page):

`https://www.engineeringtoolbox.com/air-absolute-kinematic-viscosity-d_601.html`

Modeling Liquid Viscosity Using Inverse Temperature Fits

Subject Tags: liquid viscosity; empirical models; inverse temperature; motor oil; curve fitting

Difficulty: Moderate

Estimated Time: 30 minutes

Summary: This problem investigates how the viscosity of various motor oils depends on temperature. Using an empirical model quadratic in inverse temperature, you will extract fitting constants from tabulated data, convert units, plot results on a semilog scale, and determine temperature sensitivity at 50°C.

Skills Tested:

- Curve fitting using empirical relationships
- Working with inverse temperature models
- Converting SI to cgs units (poise)
- Graphical analysis on a semilog scale
- Calculating temperature coefficients

Modeling Liquid Viscosity Using Inverse Temperature Fits

The natural log of the viscosity of many liquids is approximately quadratic in the inverse of the temperature in ^{o}K (e.g., the equation in Chapter 1 of BS&L, with an extra term).

a. Using this model, and data from the web page:

`https://www.engineeringtoolbox.com/dynamic-viscosity-motor-oils-d_1759.html`

determine constants for such a model for the different grades of motor oil. Plot the correlations and the data using a semilog scale. Convert the SI units for viscosity into poise (cgs system). Be sure to label your graph!

b. What are the temperature coefficients for the different grades at 50°C? This is the fractional change in viscosity per degree Centigrade, and can be calculated from your fitting parameters.

Hydrostatics of a Self-Gravitating Gas Sphere

Subject Tags: hydrostatics; gas laws; adiabatic expansion; spherical symmetry; numerical ODEs

Difficulty: Advanced

Estimated Time: 90 minutes

Summary: This problem considers a ball of air floating in space, where hydrostatics, the adiabatic gas law, and mass conservation determine the structure of the sphere. Students will formulate governing equations with variable gravity, nondimensionalize them, and then solve the resulting coupled ODEs numerically to find the total mass of the gas ball and compare it to the mass of Earth.

Skills Tested:

- Application of hydrostatic balance with position-dependent gravity
- Use of the adiabatic gas law in spherical geometry
- Scaling and nondimensionalization of differential equations
- Numerical integration of nonlinear ODEs

Hydrostatics of a Self-Gravitating Gas Sphere

Here's a weird application of hydrostatics: Consider a ball of gas (air) floating in space far from any other source of gravity (tidal orbital dynamics would really mess this up!). At the center of the ball, we take the pressure to be 1 atm and the temperature to be a balmy 20°C. As we move outward from the center, the pressure decreases by hydrostatics and the temperature drops by adiabatic expansion (e.g., it obeys both the ideal gas law and PV^γ adiabatic expansion). This is the result for a "well mixed" atmosphere, and applies to the earth's atmosphere (at least below the stratosphere, anyway) as well. Our goal is to determine the mass of the ball of gas.

a. Set up the problem as a pair of equations for total mass inside a particular radius and density as a function of position using hydrostatics, the adiabatic gas law, and mass conservation. Don't forget that gravity is a function of position, and that the gravitational attraction inside a spherical shell is identically zero!

b. Scale the mass and radius by some unknown values, and then use the equations to determine the characteristic mass and radius (e.g., the magnitude of the scaling parameters so that the resulting dimensionless equations will be of $\mathcal{O}(1)$).

c. Solve the dimensionless problem numerically to determine the final value of the mass of gas, and compare it to the mass of the earth. It's pretty easy to set the dimensionless problem up as a pair of coupled first order non-linear ODEs. We will review how to do this in class. Note that the integrand for the change in density is mathematically well-behaved at the origin, however you need to kick it off at a value just above zero (e.g., machine precision) or otherwise fix things to avoid division by zero numerical errors.

Hydrostatic Force on a Ball Plugging a Pool Drain

Subject Tags: hydrostatics; fluid mechanics; force balance; safety; dimensional analysis

Difficulty: Moderate

Estimated Time: 45 minutes

Summary: This problem examines the hydrostatic force conditions on a spherical ball blocking a pool drain, focusing on the balance of forces due to hydrostatic pressure distribution and atmospheric pressure. The goal is to estimate the pool depth at which the net force on the ball is zero for small drain-to-ball diameter ratios, with application to a specific ball and drain size.

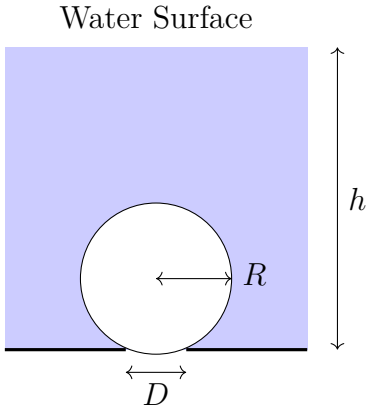
Skills Tested:

- Hydrostatic pressure calculation
- Force balance and estimation
- Dimensional analysis and unit conversions

Hydrostatic Force on a Ball Plugging a Pool Drain

Pool drains can be dangerous things - there was a tragic case a number of years ago in this area where a child was stuck in a drain on the bottom, plugging it, and drowning as a result. Here we look at a somewhat simpler problem. Suppose a ball of radius R is plugging a drain of diameter D at the bottom of a pool of depth h as depicted above. Obviously, $2R > D$ or the ball goes down the drain! Estimate the conditions under which the net force on the ball is zero for very small ratios of D/R (you can do the precise calculation for arbitrary D/R, but the math gets a little messy!). Assume that the pressure distribution in the drain is just atmospheric pressure, and that in the water is governed by the hydrostatic pressure distribution. If R is 0.5 ft and D is 3 inches, what is the corresponding depth?

You may neglect the weight of the ball (e.g., its density is very small compared to water).



What You Should Learn from Problem Set 3

Problem 1: This problem is asked to get you to do several things.

- **a.** To get a bit more practice with hydrostatics, this time for a more terrestrial problem.
- **b.** To think about controllability of a system and how things can go wrong.
- **c.** Because it was a cool story and fun to analyze...

Problem 2: Again, several things should be learned:

- **a.** To set up a time-dependent two-component mass balance.
- **b.** To solve a first-order linear ODE (e.g., to memorize the general solution, or to memorize where the solution is written down).
- **c.** To think about the concept of residence time.

Problem 3: This problem is a set-up for later material in the class and has lots of conceptual aspects:

- **a.** To think about how conservation of mass (or volume, for an incompressible liquid) connects velocities in two directions.
- **b.** To think about how average velocity, flow rate, and velocity profiles are connected together in a cylindrical geometry.

Problem 4: Index Notation:

- **a.** To get you to fish out the notes on index notation and study them...
- **b.** To demonstrate that it is far easier to solve these sorts of problems using index notation, at least after you are proficient in it!
- **c.** Practice using the concepts of symmetry and isotropy.
- **d.** Simplifying calculations by multiplication with the Kronecker delta function (e.g., turning nasty stuff into simple algebraic equations).

Lawn Chair Larry and Hydrostatic Lift

Subject Tags: Hydrostatics, Buoyancy, Atmospheric Physics, Adiabatic Expansion

Difficulty Level: Moderate

Estimated Time: 60 minutes

Summary: This problem explores the physics behind the infamous real-world story of "Lawn Chair Larry," who ascended to 16,000 ft using helium balloons. Students are asked to examine the feasibility of such a flight, determine the forces and buoyant lift involved, assess the role of an additional balloon, and account for balloon elasticity at high altitudes.

Skills Tested:

- Application of hydrostatics to buoyant lift
- Estimating forces from real-world constraints
- Modeling atmospheric variation with altitude
- Accounting for elasticity and expansion

Lawn Chair Larry and Hydrostatic Lift

A classic "Honorable Mention" on the Darwin Awards website is the saga of "Lawn Chair Larry," who decided to go flying by attaching 42 helium-filled 113 ft^3 weather balloons to his lawn chair. Instead of leveling off at around 30 ft of altitude, he wound up at 16,000 ft and was actually cited for violating LAX airspace.

a. Analyze this problem in hydrostatics and determine if it is possible to control elevation with any precision. Note that in addition to the weight of Larry and his chair (and the balloons, and the helium — you can estimate the weight of the balloons by assuming a thickness of 0.051 mm), Larry was also carrying about a dozen gallon milk jugs full of water. Assuming that he had just one more balloon than necessary to take off, estimate the total weight of his system and how much Larry and his chair weighed.

b. Now, assuming that the volume of the balloons didn't change with altitude, determine the altitude he would ascend to with the "extra balloon." Use the "well-mixed atmosphere" assumption (adiabatic expansion) coupled with hydrostatics to get the variation of density and lift with altitude. Assume gravity is constant.

c. The key error Larry made was ignoring the effect of the elasticity of the balloons. At his final altitude of 16,000 ft, what were the diameters of the balloons at that altitude?

You can find more about the story at:

`http://www.legacy.com/news/celebrity-deaths/article/larry-walters-the-flying-man`

PS: A movie *Danny Deckchair* came out a while back, which is (very) loosely based on Larry's adventure. I've got the DVD, and it's pretty amusing. If you want to borrow it (and have something that can play a DVD), let me know.

Salt Accumulation in an Evaporating Fish Tank

Subject Tags: evaporation, mass balance, exponential growth, dilution

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: This problem explores the accumulation of salt in a fish tank due to evaporation, using two different scenarios: one with simple replenishment and one with partial weekly water changes. The first part requires modeling the increasing concentration over time due to pure evaporation, while the second part uses steady-state and exponential approaches to calculate required dilution levels and time to approach steady state.

Skills Tested:

- Mass balances
- Exponential growth and decay
- Steady-state analysis

Salt Accumulation in an Evaporating Fish Tank

The fish tank in my old office had about 50 gallons of water in it. In theory, the water was fresh – but ND tap water actually has quite a bit of salt in it. The water slowly evaporated at a rate of 1 gallon/week, and all the water evaporating was fresh (the salt stays in the tank). We want to see how the salt concentration changed with time under two different scenarios:

a. If I just topped off the water every week with more tap water, how long would it be before the concentration increases by 50% (e.g., 1.5x the original value)? (this is really easy, and pretty much what happened – fortunately goldfish don't mind salty water).

b. You are *supposed* to do weekly partial water changes (I tended to forget, alas). The idea is that you remove some water from the tank, and then replace it with more fresh tap water (after removing the chlorine, of course!). How much water should I have removed/exchanged each week so that the steady-state concentration is no more than twice the tap water concentration? What is the time for it to reach about 90% of this steady-state value (estimate only)? Note that the amount extracted each week is pretty small in comparison to the total volume of the tank, so you can approximate the process as continuous rather than discrete – this makes it much easier!

Lubrication Flow Between Squeezing Plates

Subject Tags: lubrication, radial flow, conservation of mass, cylindrical coordinates

Difficulty Level: Moderate

Estimated Time: 45 minutes

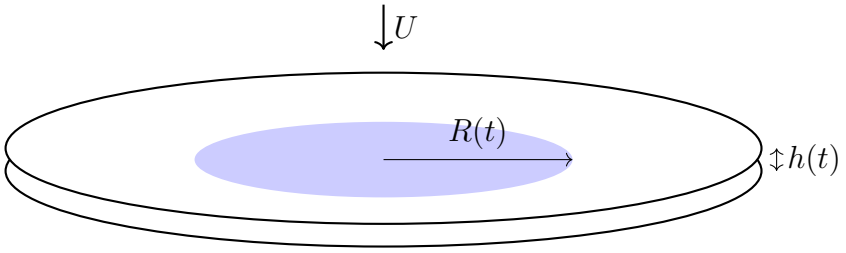
Summary: This problem analyzes the radial flow of a fluid squeezed between two parallel plates. Students apply conservation of mass and cylindrical coordinate continuity to derive the flow profile. This setup introduces the quasi-parallel approximation and sets the stage for later use of momentum balances to determine pressure and force.

Skills Tested:

- Mass conservation in cylindrical coordinates
- Velocity profile derivation using boundary conditions
- Understanding of lubrication flow and quasi-parallel approximation

Lubrication Flow Between Squeezing Plates

A classic problem in lubrication flow occurs when a fluid is squeezed between two parallel plates as is depicted below:



In this case the fluid of volume V is inserted between the plates, and the upper plate falls with a velocity U in the $-z$ direction. The lower plate is fixed, so the gap width h is governed by the simple equation:

$$\frac{dh}{dt} = -U$$

a. As the plates move together, the fluid is squeezed out radially. If the initial separation is h_0, use conservation of mass to determine the radius R of the fluid between the plates as a function of time.

b. The radial velocity will be a function of radial position (it is zero in the center, for example). Using the continuity equation in cylindrical coordinates, determine the average radial velocity (averaged over h) as a function of r and time.

c. In the limit of small h/R, the velocity is dominated by the radial flow. This is the quasi-parallel approximation that always occurs in lubrication problems, such as we shall examine in detail later this term. For this geometry (and for a Newtonian fluid) the radial velocity is a parabola in z and proportional to a function of r and t (what you got in part b, actually). From the no-slip condition, it is also zero at both $z = 0$ and $z = h$. Using this information, determine the radial velocity profile as a function of r, z, and t. Later on we'll use the momentum equations to determine pressure inside the fluid and the force required for this motion!

Surface Integrals Over Spheres in Index Notation

Subject Tags: Index Notation; Tensor Integrals; Isotropy; Symmetry; Vector Calculus

Difficulty Level: Moderate

Estimated Time: 45 minutes

Summary: This problem involves evaluating a variety of integrals over the surface of a sphere using tensor index notation, with the help of symmetry and isotropy arguments. The integrals include vector and tensor forms and test understanding of how such quantities behave over spherical domains.

Skills Tested:

- Tensor index notation
- Isotropy and symmetry arguments
- Surface integration over spherical domains

Source: CBE 30355 Problem Set 3 **File:** hw0034.tex

Surface Integrals Over Spheres in Index Notation

Using the concept of symmetry, isotropy, and index notation, evaluate the following integrals over a spherical (e.g., isotropic) surface of radius a:

a.

$$\int_{r=a} x_1^2 x_2^2 \, dA$$

b.

$$\int_{r=a} x_1^4 \, dA$$

c.

$$\int_{r=a} x_2^2 \left(x_1^2 - x_2^2 \right) \, dA$$

d.

$$\int_{r=a} \left(x_1^2 + x_2^2 + x_3^2 \right) \, dA$$

e.

$$\int_{r=a} x_1 x_3^2 \, dA$$

Hint: The integral

$$\int_{r=a} x_i x_j x_k x_\ell \, dA$$

is a symmetric, isotropic fourth-order tensor...

What You Should Learn from Problem Set 4

Problem 1: There are several things you should get from this problem.

- **a.** Application of conservation of mass (volume for incompressible fluids) to figure out velocity/flow rate relationships with change in cross-sectional area.
- **b.** Application of Bernoulli's equation (an equation that every educated person should know...) to relate pressure to velocity along a streamline.
- **c.** Relationship between changes in elevation and velocity, again through Bernoulli's equation.

Problem 2: This one has a couple of things in it:

- **a.** Recognition that force (and change in momentum) is a vector. Thus, you can apply a force without changing speed by changing direction.
- **b.** Practice in drawing appropriate control volumes which simplify calculations.

Problem 3: This problem is a tricky combination of hydrostatics and conservation of momentum:

- **a.** You need to draw the correct control volume to keep track of velocities, forces, and changes in momentum.
- **b.** You often get more momentum out of a control volume than you put in — which means there is a force being applied somewhere.

Problem 4: Index Notation:

- **a.** This is just the first example of how you can use index notation to reveal hidden physics (we'll see more later).
- **b.** Recognition of what it means to have a linear relationship and how it can dramatically simplify problems.
- **c.** Practice with the procedure of (1) reducing the number of unknowns in a problem by constructing a "most general form" of a tensor and (2) choosing simple problems to get the remaining unknowns.
- **d.** Figuring out what a director is...

Force on a Firehose Nozzle

Subject Tags: control volume; Bernoulli's equation; fluid force; pipe contraction

Difficulty Level: Moderate

Estimated Time: 25 minutes

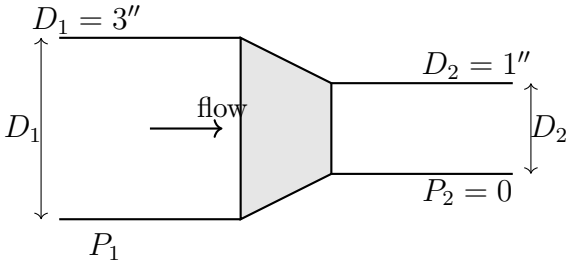
Summary: This problem asks you to determine the force acting on a firehose nozzle using a control volume approach and Bernoulli's equation, and to estimate the maximum height the nozzle can project water vertically. Design implications are discussed briefly.

Skills Tested:

- Control volume analysis
- Bernoulli's equation applications
- Unit conversion (gallons/sec to m/s, etc.)

Force on a Firehose Nozzle

A firehose nozzle is basically just a contraction in a pipe. Here we want to calculate the force on the nozzle, so we can design it so it doesn't come off! We take the firehose to have an ID of 3", the corresponding nozzle diameter is 1", the upstream pressure is P_1, the downstream pressure is 0 psig (e.g., it just comes out at atmospheric pressure, which can be ignored as it pushes equally on everything), and the flow rate is 1 gal/sec of water.



a. What's the force? Assume uniform flow (the assumptions made in class). Note that to solve this problem you have to figure out what P_1 is. If you ignore all losses, you can use Bernoulli's equation, which relates the pressure to the velocity.

b. Neglecting all losses, what is the highest fire you could use this system to put out? (Hint: Bernoulli's equation again). This height is pretty wimpy – name two ways you could increase it!

Force Exerted by a Jet on a Plate

Subject Tags: jet impingement; momentum balance; fluid force

Difficulty Level: Moderate

Estimated Time: 20 minutes

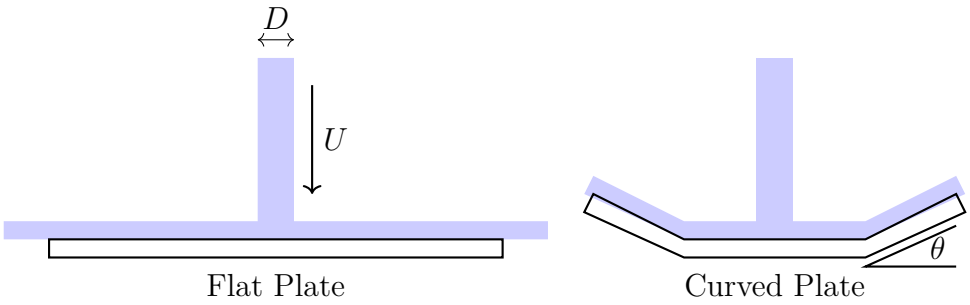
Summary: This problem examines the force generated when a fluid jet strikes a flat or curved surface. By applying conservation of momentum to a control volume, the student calculates how the force on the plate depends on the deflection of the jet and the angle at which it exits.

Skills Tested:

- Control volume momentum balance
- Application of fluid dynamics to surface interactions
- Vector resolution of force components

Force Exerted by a Jet on a Plate

A jet of fluid with density ρ, velocity U, and diameter D impinges on a flat plate as depicted below.



a. Ignoring the effects of gravity, what is the force on the plate?

b. How does this answer change if the plate is curved so the fluid sprays back out at an angle θ, and keeps the same speed U?

Force on a Sluice Gate

Subject Tags: sluice gate; hydrostatics; control volume; momentum balance

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: In this problem, you derive the force per unit width on a sluice gate by combining conservation of mass and momentum within a control volume. You will relate upstream and downstream velocities and analyze how fluid motion reduces the expected hydrostatic force on the gate.

Skills Tested:

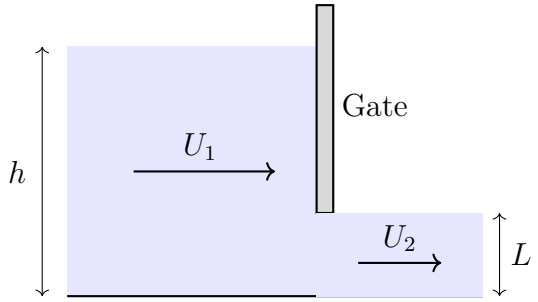
- Control volume momentum analysis
- Application of hydrostatic pressure assumptions
- Derivation of force from velocity fields and fluid properties

Force on a Sluice Gate

A liquid of density ρ flows through a sluice gate as shown. If the upstream and downstream flows are parallel, we can take the pressure distribution far upstream and downstream to be hydrostatic. If the upstream velocity is U_1, the upstream height is h, and the opening is L, derive an expression for the force per unit width necessary to keep the sluice gate in place.

Hint: Draw a control volume around the water upstream and downstream of the gate, and determine the velocity U_2 from conservation of mass. Then do a similar momentum balance. You can again ignore atmospheric pressure, but don't forget gravity! The fluid flowing out of the gate actually reduces the force on the gate from the purely hydrostatic result—the maximum possible.

Note that while we have specified the upstream velocity and height together with the gate opening, these are actually all related via Bernoulli's equation as well: higher upstream flow rates would make the fluid pile up higher on the gate. Specifying these parameters makes your calculations easier, though!



Determining λ_1 and λ_2 from Settling Cylinder Mobilities

Subject Tags: index notation; linear mobility; force-velocity relation; tensor algebra

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: This problem asks you to use measurements of the motion of a settling cylinder in two orientations to determine coefficients in a general linear tensorial mobility relation. It involves applying index notation and vector contraction for symbolic expressions.

Skills Tested:

- Application of index notation
- Symbolic analysis of tensor expressions
- Use of orthonormality and symmetry in simplification

Determining λ_1 and λ_2 from Settling Cylinder Mobilities

Index Notation: Consider a cylinder settling in a viscous fluid as depicted below. In the notes on index notation, it was shown that if an object's orientation was specified by a single director p_i, and if the relation between force and velocity was linear, then the general expression for the velocity was given by:

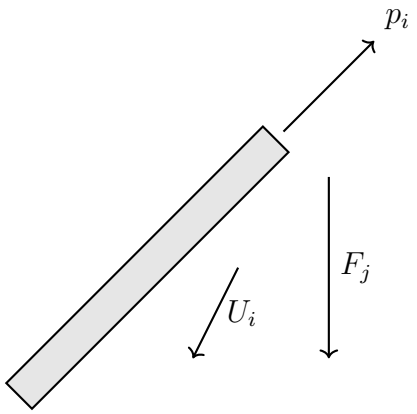
$$U_i = (\lambda_1 \delta_{ij} + \lambda_2 p_i p_j) F_j$$

where λ_1 and λ_2 are constants independent of orientation.

To determine these two constants, we measure the velocity of the cylinder when it is aligned with gravity, and when it is perpendicular to gravity. If these two mobilities are given by:

- Parallel: $\dfrac{|U|}{|F|} = A$
- Perpendicular: $\dfrac{|U|}{|F|} = B$

Determine λ_1 and λ_2.



What You Should Learn from Problem Set 5

Problem 1: There are several things you should get from this problem.

- **a.** Practice using the Navier–Stokes equations to solve unidirectional flows.
- **b.** Recognize the connection between the velocity profile, the centerline velocity, and the flow rate in channel flow, a very important geometry.
- **c.** Connecting the velocity profile to the shear stress.
- **d.** Recognizing the equivalence between gravity-driven and pressure-gradient-driven flows (both are uniform sources of momentum/volume).

Problem 2: Some things to get from this problem:

- **a.** Application of the shear stress boundary condition in addition to the no-slip condition.
- **b.** Identifying the relationship between velocity profiles and flow rates.
- **c.** More practice with parabolas...

Problem 3: This problem deals with flow in cylindrical coordinates:

- **a.** Practice using the Navier–Stokes equations in cylindrical coordinates, this time using the θ component. It's not too bad if you cross out all the terms except the one that matters!
- **b.** Showing that the "flat Earth limit" provides a useful and accurate approximation if the separation of length scales is sufficiently large.

Problem 4: Index Notation:

- **a.** This is an extension of the homework problem from last week, reinforcing your skills with manipulating linear problems using index notation.
- **b.** Showing how you can use simple measurements to get a large amount of information about a system (e.g., calculating angles of inclination and expected lateral displacements from measurements of three fall times).
- **c.** Refreshing your skills with doing relatively complex error propagation calculations. Note that if you write a little program that takes in the two fall times and spits out the lateral displacement, you can use a MC simulation approach to get the confidence interval...

Flow Between Parallel Plates (Plane-Poiseuille Flow)

Subject Tags: Viscous Flow; Navier-Stokes; Pressure Gradient; Shear Stress; Gravity-Driven Flow

Difficulty: Moderate

Estimated Time: 40 minutes

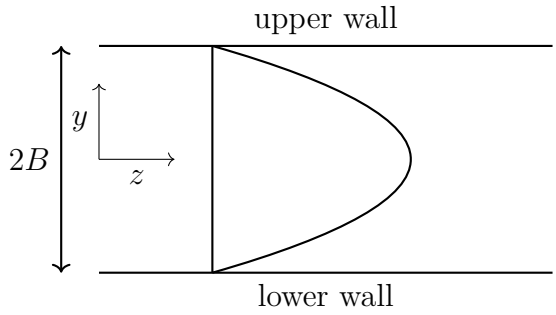
Summary: This problem explores fully developed laminar flow between two parallel plates (a slit) under a pressure gradient and under gravity. It requires applying the Navier-Stokes equations to determine velocity profiles, flow rates, shear stress, and comparing centerline and average velocities.

Skills Tested:

- Applying the Navier-Stokes equations in one-dimensional steady flow
- Computing flow rate and velocity profiles
- Determining shear stress at solid boundaries
- Analyzing gravity-driven flow in the absence of pressure gradients

Flow Between Parallel Plates (Plane-Poiseuille Flow)

Consider a viscous fluid flowing in a laminar manner through a slit formed by two parallel walls a distance $2B$ apart as is depicted below.



a. Given that we have a uniform pressure gradient in the negative z direction (i.e., $\frac{dp}{dz}$ equals a negative constant) and the fluid is massless with viscosity μ, calculate the flow rate per unit extension in the third direction (e.g., per unit width) using the Navier-Stokes equations.

b. What is the ratio of the centerline velocity to the average velocity?

c. What is the wall shear stress (force/area exerted by the fluid on the lower wall)?

d. Now suppose we reorient the system so that gravity is in the positive z direction, the fluid has density ρ, and there is no pressure gradient (e.g., open to the atmosphere at both ends). What is the flow rate in this case?

Recall that in order to preserve the parallel between mass, momentum, and energy transport, BS&L defined the viscous stress tensor τ_{ij} to have a sign opposite of that defined in class and used in most other textbooks. If you keep this in mind, and recall that the sign of the stress tensor is completely arbitrary (as long as you are consistent!), it should not cause you any confusion.

Shear Stress Needed to Arrest Flow Down an Incline

Tags: Viscous Flow; Inclined Plane; Shear Stress; Marangoni Effect

Difficulty: Moderate

Estimated Time: 25 minutes

Summary: This problem models the unidirectional flow of a viscous fluid film down an inclined plane under gravity, while a breeze imposes an opposing shear stress on the upper surface. The task is to determine how large this surface shear stress must be in order to arrest the flow entirely — that is, for the volumetric flow rate to become zero.

Skills Tested:

- Momentum balance for viscous flow
- Applying boundary conditions to differential equations
- Integration of velocity profiles to compute net flow

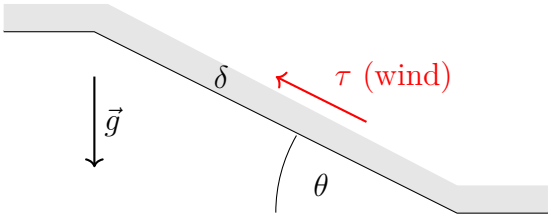
Source: CBE 30355 Problem Set 5 **File:** hw0052.tex

Shear Stress Needed to Arrest Flow Down an Incline

Consider the velocity distribution acquired by the slow flow of a very viscous fluid down an incline, as is depicted below. In contrast to the problem considered in the lecture notes, here we have a breeze trying to blow the fluid back up the incline, in addition to the gravity trying to draw the fluid down. We model the effect of the breeze with a constant shear stress τ at the upper surface of the fluid (note: τ will be negative for the coordinate system as drawn, and is equivalent to the Marangoni stress in the tears of wine demonstration!).

For some film depth δ, angle θ, density ρ, and viscosity μ, how large does this shear stress have to be for the net flow rate (the integral of the velocity over the film depth) to be zero?

Solve the problem assuming unidirectional flow.



Curvature Effects in a Couette Viscometer

Tags: Couette Flow; Viscosity Measurement; Cylindrical Coordinates; Torque-Rotation Relationship

Difficulty: Advanced

Estimated Time: 45 minutes

Summary: This problem analyzes how curvature affects the relationship between torque and rotation rate in a Couette viscometer. The student is first asked to derive the exact torque relationship in cylindrical coordinates, then to examine and compare various approximations of the viscosity formula for increasing gap width. The error associated with using approximations in a real instrument is also evaluated.

Skills Tested:

- Cylindrical momentum balance
- Integration of shear stress to compute torque
- Comparison of exact vs. approximate analytical models
- Quantitative error analysis and visualization

Curvature Effects in a Couette Viscometer

In class we derived a way to estimate the viscosity of a fluid using a Couette viscometer by neglecting the curvature effects (e.g., if the gap width is ΔR and the inner radius is R_1, we required $\Delta R/R_1 \ll 1$).

a. Using cylindrical coordinates, derive the exact relationship between torque and rotation rate where the inner radius is R_0 and the outer radius is R_1 such that $\Delta R = R_1 - R_0$. Note that if you get desperate, this problem is worked out completely in BS&L...

b. The approximate formula results from ignoring curvature (e.g., $\Delta R/R_0 \ll 1$) and uses some characteristic radius R (e.g., a shear rate of $\Omega R/\Delta R$, circumference of $2\pi R$ and a lever arm of R). It is interesting to see how well this approximate formula works. Plot up the ratio of the approximate formula to the exact value for four choices for the radius R in the approximation:

(a) where we take R to be the inner radius R_0

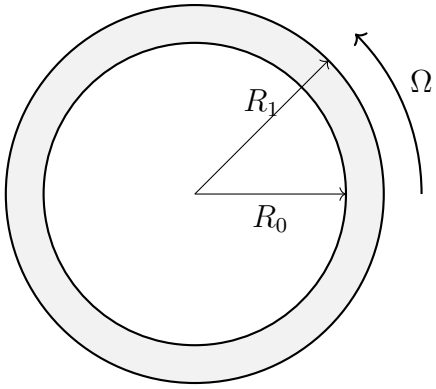
(b) where we take R to be the outer radius R_1

(c) where we use a mixed relation where the velocity of the outer cylinder is based on R_1 and the lever arm and area for calculating the torque is based on R_0

(d) where we take R to be the average of the inner and outer radii

Do this for values of $0 < \Delta R/R_0 < 0.5$.

c. A Couette device in my old lab had a bob (inner cylinder) radius of 2.375 cm and a gap width of 1.261 mm. How large is the error in the calculated viscosity if we use the four different approximate formulae for this instrument?



Mobility Tensor and Trajectory of a Falling Washer

Tags: Mobility Tensor; Viscous Drag; Falling Body; Index Notation; Uncertainty

Difficulty: Advanced

Estimated Time: 60 minutes

Summary: This problem uses experimentally measured fall times of a washer dropped flat-side-on and edge-on to determine the complete mobility tensor for a body of revolution in a viscous fluid. From this, velocity projections as a function of angle are derived, and the predicted trajectory and confidence intervals for lateral displacement are calculated using uncertainty analysis.

Skills Tested:

- Use of index notation for projection of mobility tensors
- Interpretation of experimental velocity data
- Geometric decomposition of motion in a viscous fluid
- Propagation of uncertainty and confidence interval estimation

Source: CBE 30355 Problem Set 5 **File:** hw0054.tex

https://diffusivepress.org/courseware/cbe30355/

Mobility Tensor and Trajectory of a Falling Washer

In the last homework you determined the complete mobility tensor for a falling body of revolution in a viscous fluid from two simple experiments: the measured velocity where the director was perpendicular to gravity, and where it was parallel to gravity.

In class the time measured for a washer to fall a set distance through Karo syrup was 11.2 s if flat side on and 7.5 s if edge on (well, it would have been if I had used enough corn syrup to get the correct depth!). Using this information and the magic of index notation, we can calculate all sorts of stuff.

a. If the director p_i of the axis of revolution is given by:

$$p_i = \delta_{i1}\cos(\theta) + \delta_{i2}\sin(\theta)$$

where θ is the angle relative to vertical (e.g., gravity), derive the formula for the velocities in the 1 and 2 directions as a function of angle. Note that an angle of zero corresponds to dropping it flat side on.

b. Suppose we drop the washer at an angle and observe a fall time of 9.3 s. Based on this, what would be the angle θ?

c. The ratio of the two velocities determines the trajectory. It was about 11 cm from where I let it go to the bottom. Using the information above, calculate the expected lateral displacement.

d. If the 1-sigma random error in each of the time measurements was 0.1 s (pretty optimistic, I'm afraid) and the standard deviation in the height was 1 cm, what is the 95% confidence interval of the predicted lateral displacement for part **c**? Assume that all the measurement errors are independent. You can do this any way you like, but a MC simulation (e.g., throwing the computer at it) is probably the easiest...

What You Should Learn from Problem Set 6

Problem 1: This one has three key features:

- **a.** Solving the velocity profile in a geometry that is very important to rheology.
- **b.** Allowing the boundary conditions to suggest the form of the solution. Most fluid mechanics problems are rendered much easier using this concept.
- **c.** Recognizing that inertial effects can produce secondary flows – which are quite large and significant in this geometry.

Problem 2: This one has several things in it:

- **a.** Solving unidirectional flow in cylindrical coordinates.
- **b.** Showing that small obstructions can have large effects on pressure drop/flow rate relationships.
- **c.** Showing that the forces calculated from detailed velocity distributions must and do match what you would get from integral force balances.
- **d.** Practice in rendering a problem dimensionless to simplify calculations.

Problem 3: This problem focuses on two-fluid flows:

- **a.** Solving an industrially relevant problem – at least for a simple model system.
- **b.** Practice rendering a problem dimensionless (unless you want to do lots of extra work!).
- **c.** Recognizing that velocity and shear stress are continuous at an interface!
- **d.** Showing that the "flat earth limit" provides a useful and accurate approximation if the separation of length scales is sufficiently large.

Problem 4: The viscosity pump:

- **a.** Demonstrating that, once again, the "flat earth limit" makes a problem a lot simpler to solve!
- **b.** Application of the principle that (for uni-directional flows) you can regard a flow as the linear superposition of two simpler problems. This works for any linear problem, actually, which is one reason they're much easier to solve.
- **c.** Connection between pressure drop, flow rate, and rate of work.

Parallel Plate Viscometer Analysis

Subject Tags: torque; Navier-Stokes; secondary flows; cylindrical coordinates; viscometer

Difficulty: Moderate

Estimated Time: 40 minutes

Summary: This problem asks you to analyze the velocity profile and torque in a parallel plate viscometer, assuming low Reynolds number conditions. You will determine the functional form of the azimuthal velocity and calculate the torque, then consider how neglected terms in the Navier-Stokes equations produce secondary flows.

Skills Tested:

- unidirectional flow solution
- torque calculation
- physical interpretation of NS terms

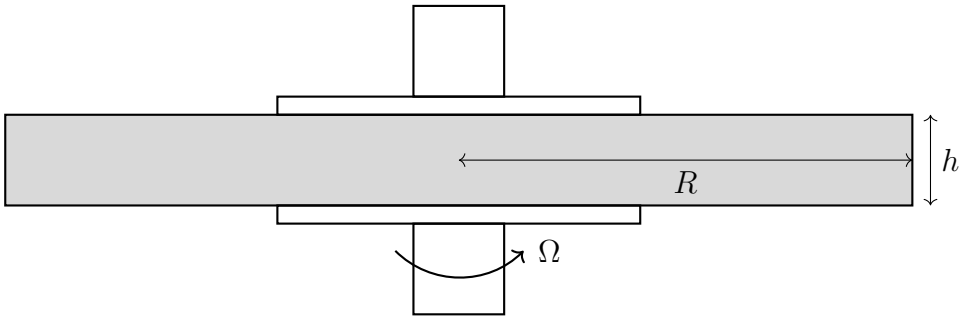
Source: CBE 30355 Problem Set 6 **File:** hw0061.tex

Parallel Plate Viscometer Analysis

Consider the parallel plate viscometer depicted below. The viscometer consists of two parallel disks of radius R separated by a gap h. In operation, the gap between the disks is filled with a viscous fluid and the lower plate is rotated with some angular velocity Ω, resulting in some torque on the upper plate. The gap width is quite small ($h/R \ll 1$), so the fluid is confined to the space between the plates by surface tension. The ratio of the torque to the angular velocity is proportional to the fluid viscosity (at least for Newtonian fluids).

a. If we may neglect the non-linear inertia terms (i.e., low Reynolds number flow), show that the equations of motion are satisfied by a velocity $u_\theta = f(r, z)$ with $u_r = 0$, $u_z = 0$. Determine the velocity profile and calculate the torque on the upper plate as a function of the experimental parameters.

b. By examining the equations of motion in the r and z directions, demonstrate that the above solution will not satisfy the full Navier-Stokes equations. Identify which terms give rise to difficulties and provide a short physical explanation for what is occurring. Sketch the velocity profile you expect to see in the r-z plane (Don't try to solve for this secondary current velocity profile unless you like a lot of extra work). In our lab we have used this secondary current for all sorts of experiments.



Flow in a Tube with an Axial Wire

Subject Tags: gravity-driven flow; cylindrical coordinates; viscous drag; pressure-free flow

Difficulty Level: Moderate

Estimated Time: 45 minutes

Summary: This problem extends the classical problem of gravity-driven flow in a vertical pipe by introducing a central obstruction: a thin axial wire. The aim is to quantify how much the presence of the wire affects the flow rate, and to analyze the distribution of viscous forces on both the wire and the outer tube wall. The calculation is straightforward if performed in nondimensional form.

Skills Tested:

- Navier-Stokes application in cylindrical geometry
- Use of boundary conditions for velocity profile derivation
- Dimensionless parameterization of fluid dynamics problems
- Analytical and numerical evaluation of flow rate integrals
- Calculation of viscous force distributions

Flow in a Tube with an Axial Wire

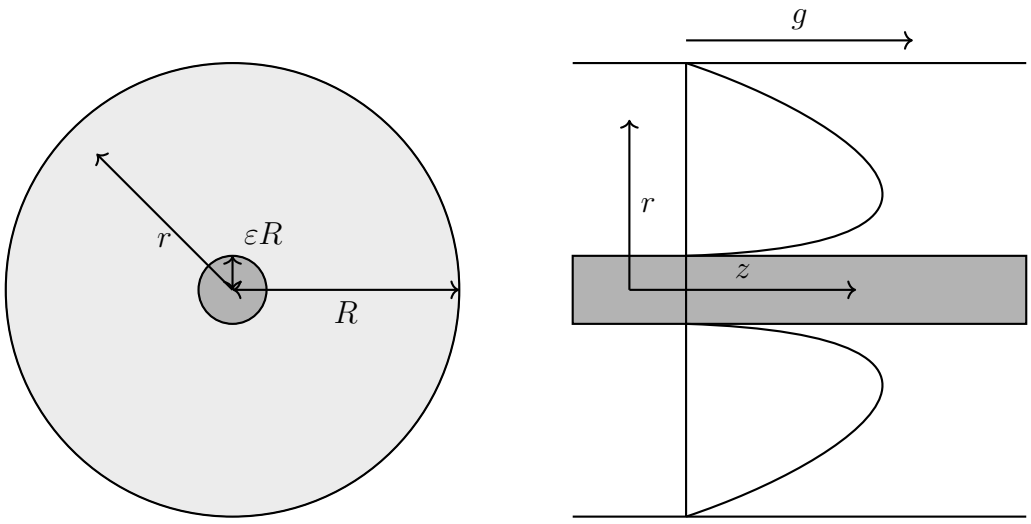
In class last Tuesday we discussed the flow of a fluid through a pipe driven by a pressure gradient. In this problem, consider a pipe of radius R with a small cylindrical wire of radius εR running axially down the center. If the remaining space in the tube is filled with a viscous fluid of density ρ and viscosity μ and there is no pressure gradient (the tube is open at both ends), calculate the resulting flow rate due to gravity as a function of ε and compare its magnitude to that when the wire is absent. The velocity profile needs to be obtained analytically, and isn't too bad, but the flow rate gets a bit messy. It really really helps to do this in dimensionless form! You may do the flow rate numerically (plotting it up as a function of ε) if you wish, or get it using Wolfram Alpha.

a. What is the flow rate as a function of ε? By how much is the flow rate reduced if $\varepsilon = 0.05$? Is this surprising? (Note: this is actually relevant to the reduction in flow rate/increase in pressure drop when you thread a catheter through an artery or vein.)

b. Determine the force per unit length exerted by the fluid on the wire for this value of ε.

c. What is the ratio of the force on the wire to the force on the outer wall at $r = R$ (again per unit length)?

d. Show that the sum of the force per unit length on the wire and wall calculated above is equal to the weight per unit length of the fluid in the tube.



Core-Annular Flow of a Viscous Fluid

Subject Tags: core-annular flow; pressure drop; viscosity ratio; lubrication theory

Difficulty Level: Moderate

Estimated Time: 45 minutes

Summary: This problem analyzes the reduced pressure drop achieved in transporting a highly viscous fluid by surrounding it with a thin sheath of low-viscosity fluid. Students calculate the flow rate as a function of the geometry and viscosity ratio, then compare exact and approximate approaches.

Skills Tested:

- Cylindrical coordinate flow analysis
- Using viscosity ratios and nondimensionalization
- Applying asymptotic (lubrication) limits

Source: CBE 30355 Problem Set 6 **File:** hw0063.tex

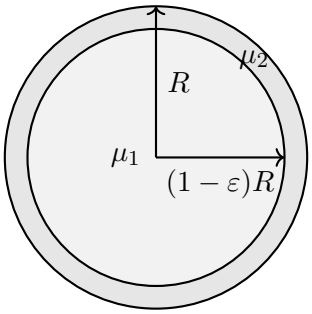
Core-Annular Flow of a Viscous Fluid

Core-Annular flow is an important technique for reducing the pressure drop for pumping a viscous fluid such as a heavy oil. The idea is that the viscous oil is surrounded by a narrow sheath of fluid with much lower viscosity (such as water). Because most of the shear would occur in this narrow region next to the wall, the stress and pressure drop is much lower for a given flow rate. Here we solve this problem.

a. Consider the geometry depicted below. A tube of radius R is filled by two fluids. The inner fluid has a viscosity μ_1, and occupies the center of the tube up to a radius $(1-\varepsilon)R$, while the outer fluid has a viscosity μ_2 and occupies the annulus between this radial position and the wall. Calculate the pressure-drop flow rate relationship for the first fluid (we don't care about the sheath fluid!) as a function of the parameters of the problem.

b. The ratio of the flow rate/pressure drop relationship in core-annular flow to that of the case $\varepsilon = 0$ (e.g., no sheath fluid) is just a dimensionless function of ε and the viscosity ratio $\lambda = \mu_1/\mu_2$. What is the value of this ratio (e.g., $Q/Q|_{\varepsilon=0}$) for $\varepsilon = 0.01$ and $\lambda = 1000$?

c. Since ε is usually very small, it is actually much easier to solve the problem in the "flat earth limit". Essentially this reduces to the flow of the viscous inner fluid filling the entire pipe, but with the usual no-slip boundary condition at the wall modified to include a wall slip velocity that is proportional to the shear stress at the wall. Determine what this boundary condition looks like and re-solve the problem in the small ε limit. Compare the solution obtained this way to your answer for part **b**.



Viscosity Pump Performance and Efficiency

Subject Tags: low Reynolds number flow; Couette-Poiseuille flow; energy balance; mechanical efficiency

Difficulty: Moderate

Estimated Time: 40 minutes

Summary: This problem models a viscosity pump composed of a rotating drum and narrow fluid gap. Students will analyze flow rate generation via superposed Couette and pressure-driven Poiseuille flow, identify the pressure limit, and perform a mechanical energy balance to determine pump efficiency and energy loss mechanisms.

Skills Tested:

- lubrication theory
- superposition of Couette and Poiseuille flows
- torque and mechanical power calculations
- interpreting energy efficiency

Source: CBE 30355 Problem Set 6 **File:** hw0064.tex

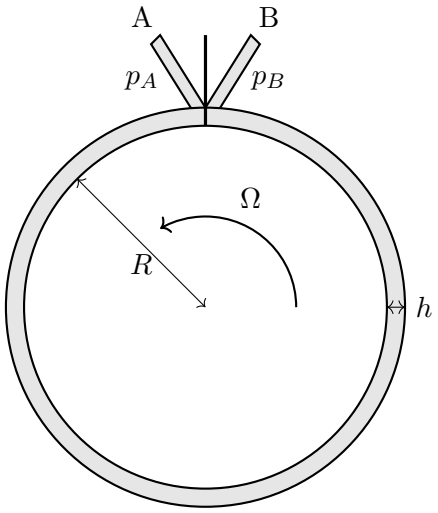
Viscosity Pump Performance and Efficiency

A viscosity pump is depicted below. Fluid of viscosity μ is pumped from inlet A to outlet B by the rotating drum of radius R. Note that $p_A < p_B$, and that this pressure gradient will induce some backflow from B to A. The gap width h is considered to be much less than R, so that the flow in the gap may be modeled as flow between parallel planes (e.g., the flat earth limit!).

a. Neglecting all inertial effects, calculate the flow rate per unit width of this pump Q/W (it is assumed to extend out of the paper a distance W) as a function of Ω, $p_B - p_A$, R, and h for a fluid with viscosity μ.

b. What is the maximum Δp it can pump against?

c. Calculate the resulting torque on the shaft that drives the drum and the mechanical energy input to the system. If the useful work done on the fluid is given by $Q \cdot \Delta p$, what is the energy efficiency of the pump? Where does the extra energy go?



It is interesting to note that the behavior of a Newtonian liquid in this geometry is very straightforward, but really weird things happen if you use a concentrated suspension or an emulsion of droplets instead. What occurs then is that particles (or drops) will migrate to different locations across the gap and, due to the backflow, wind up with a highly non-uniform concentration variation in the θ direction. In our laboratory we've looked into this as a way of concentrating particles or mechanically breaking an emulsion.

What You Should Learn From Problem Set 7

Problem 1: There are a couple of things you should get from this problem.

- Practice using the Buckingham Π theorem and straight-up dimensional analysis.
- A bit of practice using graphical comparison, linear regression, error calculation, etc.

Problem 2: This is a scaling problem, and you should:

- Learn how to render a differential equation and boundary condition dimensionless.
- Use scalings to "estimate" quantities without actually having to solve a differential equation. Such estimates are often useful for "back of the envelope" calculations that let you quickly recognize what is going on in an experiment or problem you are studying – hopefully in time to determine if things are occurring which are contrary to expectations, which lets you examine it more closely while "things are going on". This can be a very important time saver in the long run as experiments are expensive!
- Practice solving a second order ODE.

Problem 3: There are several things you should get from this problem.

- Practice using the relationships for strict dynamic similarity.
- Recognition that strict similarity is (usually) not practical.
- Coming up with a "best choice" scale model for approximate dynamic similarity.
- Figuring out how to scale things (like mixing time) based on the other scaling relationships.

Problem 4: The Stokes Flow Equations:

- Practice manipulating PDE's in index notation (pretty much straight math / multivariable calculus).
- This is a set-up for solving for flow past a sphere using spherical harmonics. We'll play with this a bit, but I don't really regard it as "core". It is the fastest way to get the "6π" factor for the drag on a sphere, though, as well as the starting point for all really complicated Stokes flow problems.

Dimensional Analysis in Cooking a Turkey

Subject Tags: dimensional analysis; heat transfer; scaling laws; data fitting

Difficulty Level: Moderate

Estimated Time: 45 minutes

This problem uses dimensional analysis to understand the physics behind cooking time for turkeys of different sizes. It challenges students to apply the Buckingham Pi method and test their theoretical results using regression analysis and confidence intervals. A practical, real-world application of scaling laws and heat transfer concepts.

Skills Tested:

- Applying dimensional analysis (Buckingham Pi)
- Understanding thermal diffusion and scaling with mass
- Performing regression analysis and interpreting fits
- Using t-statistics for confidence level hypothesis testing

Dimensional Analysis in Cooking a Turkey

a. If you've ever looked at the cooking timetables for large things like turkeys or roasts, you may have noticed that they are pretty complex: so much per pound if the weight is in one range, and a different ratio in another range, etc. In general, the cooking time per pound goes down the larger the bird.

If we assume that:

- All turkeys are geometrically similar,
- The physical properties such as thermal diffusivity, density, etc., are all constant, and
- The cooking time depends only on the mass, the density, and the thermal diffusivity (for idealized turkeys, anyway),

use dimensional analysis to show how cooking time should vary with the mass of the bird.

b. We have below values for cooking time obtained from the website:

`http://www.butterball.com/how-tos/roast-a-turkey`

The convection oven times are given by weights:

Weight Range (lbs)	**Cook Time (Unstuffed)**
6–10	1.5–2 hrs
10–18	2–2.5 hrs
18–22	2.5–3 hrs
22–24	3–3.5 hrs

Since the weights are in a range, you can get a total of 5 "exact" weights and times. Using regression analysis, fit the data to a power law and determine if the exponent matches that predicted from part **a** to within the 90% confidence level. Since it is a limited data set, don't forget to use t-statistics!

Graphically present your results showing:

- The data,
- The best fit power law, and
- The "theoretical prediction" based on assuming the exponent obtained in part **a** and the constant determined numerically.

Damped Pendulum in a Viscous Fluid

Subject Tags: pendulum; dimensional analysis; critical damping; ODEs

Difficulty Level: Moderate

Estimated Time: 60 minutes

This problem explores the motion of a pendulum subject to viscous drag from a surrounding fluid. It combines physical modeling, nondimensionalization, and analysis of second-order ODEs to identify conditions for critical damping and estimate time scales. Students will compare results from dimensional estimates with exact analytical solutions in the linearized limit.

Skills Tested:

- Nondimensionalization using physical parameters
- Identification of dimensionless groups and their interpretation
- Analysis of critically damped systems
- Solving and interpreting second-order linear ODEs

Source: CBE 30355 Problem Set 7 **File:** hw0072.tex

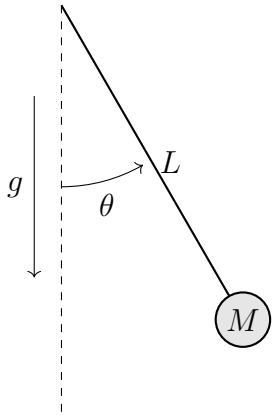
Damped Pendulum in a Viscous Fluid

Consider the damped pendulum depicted below. The pendulum consists of a ball of mass M and radius a suspended by a fine wire of length L below a pivot. The ball is moving through a fluid of viscosity μ (the fluid is assumed massless for convenience), which retards its motion. It is assumed that the ball moves sufficiently slowly that Stokes Law applies. The resistance of the wire is neglected as well (that one's really questionable!). Under these conditions, the equation describing the motion of the sphere is given by:

$$ML\frac{d^2\theta}{dt^2} = -Mg\sin(\theta) - 6\pi\mu aL\frac{d\theta}{dt}$$

with initial conditions:

$$\theta(t=0) = \theta_0, \quad \frac{d\theta}{dt}(t=0) = 0$$



a. By choosing an appropriate reference time scale, render the equation and initial condition dimensionless so that the dependent and independent variables are of order one. Show that the dimensionless solution depends on only two dimensionless groups, one of which is just the initial angular displacement. What is the physical meaning of the other dimensionless group?

b. You are assigned the problem of adjusting the fluid viscosity so as to bring the sphere to rest as rapidly as possible. Recognizing that if $\mu = 0$ the pendulum will oscillate forever, and if μ is very large the sphere will take a long time to get to the equilibrium ($\theta = 0$) position, estimate the correct value of the viscosity for critical damping. Estimate from dimensional analysis (e.g., assume all unknowns are "O(1)") how long it will take the sphere to approach to within, say, 25% of the equilibrium position.

c. Given that θ_0 is very small, we may make the approximation $\sin(\theta) \approx \theta$. Solve the resulting constant coefficient differential equation, and compare the exact solution to that estimated by dimensional analysis above.

Scaling of Mixer for Density-Stratified Fluids

Subject Tags: dimensional analysis; dynamic similarity; scale modeling; fluid mixing

Difficulty Level: Moderate

Estimated Time: 45 minutes

Summary: This problem explores how to scale a fluid mixing process involving stratified fluids of different densities using dynamic and approximate dynamic similarity. Students are asked to evaluate physical properties in model systems and determine how time and velocity should scale, based on fluid mechanics principles and scaling laws.

Skills Tested:

- dimensional analysis
- modeling with dimensionless groups
- practical scaling of physical systems

Scaling of Mixer for Density-Stratified Fluids

Mixing two fluids of different density in a very large tank can be quite challenging: the denser fluid tends to stay on the bottom, and it is hard to get it to mix in. You are in charge of developing a scale model to simulate a mixer for the Hanford cleanup problem. The full-scale system has a tank that is 10 m high, and the two fluids to be mixed have a density of 1.4 g/cm^3 and 1.05 g/cm^3, and a viscosity of 4 cp and 2 cp, respectively. You have constructed a 1:8 scale model to test mixing strategies.

a. If you were to preserve strict dynamic similarity, what should be the densities and viscosities of the fluids in the model system? Based on this, what should be the scaling factor for the velocities between the model and full-scale systems?

b. You discover that there simply isn't any reasonable way to satisfy the conditions in (a), thus you go to the concept of approximate dynamic similarity. What fluids and scaling factors should you use in this case? Note that there isn't a unique solution to this part, but it is convenient to have one of the fluids in the model system be water.

c. For the conditions determined in (b), how would the mixing times scale between the two systems?

Stokes Flow and Index Notation

Subject Tags: creeping flow; index notation; pressure field; incompressible flow

Difficulty Level: Moderate

Estimated Time: 50 minutes

Summary: This problem demonstrates how to derive and manipulate the Stokes flow equations using index notation. Students are led to understand how the pressure field in creeping flow satisfies Laplace's equation, and how this leads to a general structure for solutions involving both particular and homogeneous components.

Skills Tested:

- vector and index calculus
- manipulation of Navier-Stokes in limiting cases
- partial differential equations

Stokes Flow and Index Notation

The Stokes Flow equations are:

$$\mu \nabla^2 \vec{u} = \nabla p$$

a. By taking the divergence of the equations of motion and applying the equation of continuity, prove that

$$\nabla^2 p = 0$$

for an incompressible fluid undergoing flow at zero Reynolds number. Use index notation only, and note that the ∇ and ∇^2 operators commute.

b. Complex problems in Stokes Flow can be solved by breaking the solution for the velocity into the sum of a particular solution and a homogeneous solution, e.g.,

$$u_i = u_i^p + u_i^h$$

where u_i^p and u_i^h satisfy:

$$\mu \frac{\partial^2 u_i^p}{\partial x_j^2} = \frac{\partial p}{\partial x_i}, \qquad \frac{\partial^2 u_i^h}{\partial x_j^2} = 0$$

but neither u_i^p nor u_i^h, in general, satisfy the continuity equation individually (only the sum does).

Normally this wouldn't do any good, but you can get a general solution for u_i^p if you know the pressure! Using the result from part (a), prove that this particular solution is just:

$$u_i^p = \frac{x_i p}{2\mu}$$

In the graduate transport class we make extensive use of this to solve complicated creeping flow problems, as harmonic functions (i.e., those for which $\nabla^2 \phi = 0$) are well known, so getting both p and u_i^h is relatively simple!

What you should learn from problem set 8

- **Problem 1: Lubrication: defined flow rate.**
  - This is a very nice example of a defined flow rate lubrication problem, although you will need to be careful in applying your boundary conditions!
  - Reinforces that, as in all lubrication problems, the velocity profile across the thin direction is just a parabola... Thus, if you get the three constants by applying boundary conditions, symmetry, etc., the problem becomes much simpler!
  - More practice rendering equations dimensionless!
- **Problem 2: Lubrication flow between two disks**
  - This makes you work through the notes on defined flow rate lubrication problems.
  - Demonstrates how you can use the results of one problem to provide information on a rather different one (e.g., tying the results together).
  - Recognizing that sometimes "push button instruments" can yield systematic errors due to unanticipated effects, and thinking through how you could fix the results by proper recognition of why the results were in error...
- **Problem 3: Scaling & Thermodynamics**
  - This problem is a useful example of applying your intuition to get insight into how a real problem (e.g., a chimney) depends on various parameters.
  - Helps to see how thermo and transport are interrelated.
- **Problem 4: Dimensional Analysis**
  - Another example of scaling, this time using strict dynamic similarity.
  - Application of scaling to a real-world modeling problem.

Lubrication Layer Between Rollers

Subject Tags: lubrication theory, scaling analysis, pressure-driven flow, similarity, dimensionless integrals

Difficulty Level: Advanced

Estimated Time: 90 minutes

Summary:
In this problem, students analyze a thin lubrication layer that forms when two sheets are drawn through a narrow gap between rollers. Using scaling arguments and the lubrication approximation, the force per width on the rollers is related to geometric and kinematic parameters. Students derive dimensionless relations for gap width, pressure, and force, and analyze how detachment geometry is governed by a self-contained integral expression.

Skills Tested:

- Scaling analysis of flow in narrow geometries
- Integral formulation of pressure-driven flows
- Application of the lubrication approximation
- Evaluation of nondimensional integrals

Lubrication Layer Between Rollers

Consider the geometry below. Two sheets are drawn with a velocity U between two rollers of radius R whose surfaces are separated by a distance $2b_0$. The idea is to fill the space between the sheets with a viscous fluid of viscosity μ, and all the action takes place in a lubrication layer between the rollers, at least in the limit $b_0/R \ll 1$. Here we analyze this problem.

a. By scaling the flow equations in the gap (use Cartesian coordinates!), show how the force per width on the rollers F/L scales with the separation distance between the rollers and the other parameters of the problem.

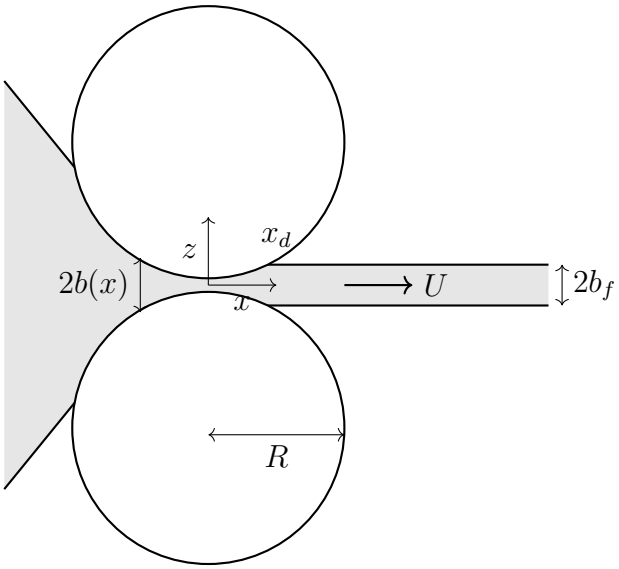
b. The sheet sandwich detaches from the rollers downstream at a point x_d and final gap width $2b_f > 2b_0$ when the pressure again returns to zero. Develop an implicit integral relation for the ratio b_f/b_0, and show that it doesn't depend on any of the other parameters of the problem.

c. Develop an integral relationship for the dimensionless pressure and force and evaluate it numerically.

Hint: The gap geometry in the lubrication limit is given by

$$b(x) = b_0 + \frac{x^2}{2R}$$

where x is the distance along the gap from the point of minimum separation. By continuity, the flow rate per extension into the paper (Q/W) through any vertical plane must be the same, and must equal the exit flow rate $U \cdot 2b_f$. Thus, this is a defined flow rate problem which makes things *much* easier!



Zero Error in Parallel-Plate Viscometers

Subject Tags: rheology, viscometer, scaling analysis, viscosity measurement, error analysis

Difficulty Level: Moderate

Estimated Time: 75 minutes

Summary:
This problem explores a subtle but important source of error in commercial viscometers that use air drag to define the zero gap condition. By analyzing the air lubrication forces as the plate descends, students determine the shift in the zero reference and estimate its impact on viscosity measurement. The problem highlights how small systematic errors can lead to substantial measurement inaccuracies and offers a practical fix using measured data.

Skills Tested:

- Scaling analysis of lubrication forces
- Order-of-magnitude estimation
- Error propagation in measurements
- Critical interpretation of experimental setup and results

Zero Error in Parallel-Plate Viscometers

In an interesting paper (Davies & Stokes, *J. Rheol.* 2005) a fundamental problem with current parallel-plate viscometers was identified. As you know, the viscosity of a fluid is measured in a parallel-plate viscometer by rotating one plate at a known angular velocity and measuring the resultant torque on the other plate. The viscosity calculation, however, requires accurate knowledge of the gap width between the plates.

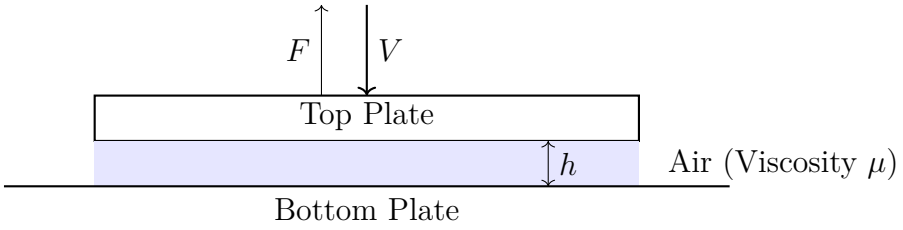
In a modern instrument it is possible to set the gap accurately automatically *provided* you have the reference point of where the two plates are in contact: e.g., where the gap is zero. For an instrument such as this, you just push a button, the upper plate descends at a constant velocity of $100\,\mu$m/s, and the gap is considered to be zero when the measured upward thrust on the upper plate exceeds 10^4 dynes, providing the reference point for all subsequent gaps—very convenient! This would be fine if there were no fluid in the gap (e.g., a vacuum), but as you know, the viscosity of air is non-zero (1.8×10^{-4} poise). In this problem we analyze what the effect of air is on the gap zeroing of 3 cm radius parallel-plates.

a. For a normal force detection threshold F, approach velocity V, air viscosity μ, and plate radius R, calculate the error in the gap set zero.

b. For the numbers given above, calculate the numerical value of the error in microns. Is it practical to reduce this error by an order of magnitude by decreasing V? Be specific!

c. If we set the gap to $250\,\mu$m using this incorrect zero (e.g., actual gap differs from $250\,\mu$m due to the zero error), what is the corresponding error in the viscosity measurement μ_{exp}/μ of a fluid?

d. How can you use the output of the instrument to get the correct viscosity? Be specific! *(Hint: It will require analysis of a couple of measurements.)*



Thermodynamics and Scaling Analysis

Subject Tags: Thermodynamics, Scaling, Buoyancy-Driven Flow

Difficulty: Moderate

Estimated Time: 45 minutes

Summary: In this problem, you estimate the fluid velocity and temperature inside a chimney with a constant energy input using fundamental conservation laws. The analysis focuses on two asymptotic regimes: when heating is weak and when it is strong. This is a good exercise in back-of-the-envelope scaling, applying both thermodynamic and fluid dynamic reasoning.

Skills Tested:

- Application of energy conservation principles
- Momentum balance under buoyancy forces
- Dimensional and limiting behavior analysis

Thermodynamics and Scaling Analysis

Consider the temperature and fluid velocity in a chimney as a function of various physical parameters. The chimney has cross-sectional area A and height H, so the total volume of air in the chimney is AH. A constant energy source at the bottom supplies energy at a rate Q, uniformly over the base. The system is idealized: we neglect friction and heat loss and assume the energy input is independent of flow rate. This energy input heats the air, which expands and rises. Using conservation of energy and momentum, estimate the mass flow rate and the resulting temperature in the chimney.

a. For small values of Q, the temperature and density changes are small. What are the mass flow rate and chimney temperature in this limit?

b. For large values of Q, the density of the gas in the chimney is much less than that of the surrounding air. What are the mass flow rate and temperature in this limit?

Hint: You will need to recall some concepts from Thermodynamics.

Dimensional Analysis: Modeling Air Entrainment in a Draining Tank

Subject Tags: Dimensional Analysis, Scaling, Fluid Mechanics, Tank Drainage

Difficulty: Moderate

Estimated Time: 35 minutes

Summary: In this problem you will determine how to scale a fluid tank drainage problem, including vortex-induced air entrainment, from a full-size system using heavy crude to a model system using water. The goal is to ensure dynamic similarity, allowing the design and safety limits of the large-scale tank to be predicted from small-scale model tests.

Skills Tested:

- Application of dimensional analysis to scale flow systems
- Determining dynamic similarity using dimensionless numbers
- Relating scale factors to flow rates and operating limits

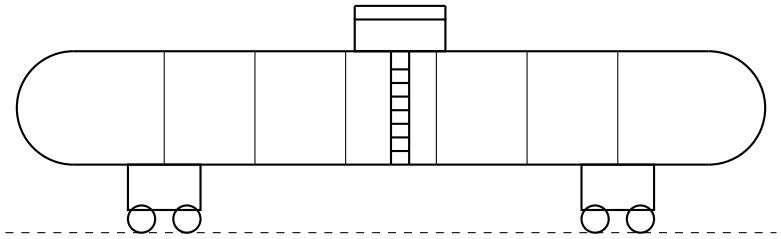
Modeling Air Entrainment in a Draining Tank

The use of rail cars to ship heavy crude down from Canada or from the fields of North Dakota has exploded (sometimes literally) in recent years. As a consequence of safety requirements, tank cars are being redesigned. You are assigned the task of performing a scale model test to determine the maximum drainage rate.

Consider the liquid tanker rail car depicted below. The 53 foot long tanker contains heavy crude with viscosity $\mu = 10$ poise and density $\rho = 0.922\,\mathrm{g/cm^3}$. As the tank drains, a vortex will form and eventually air will be entrained into the drainpipe—something we want to avoid. We want to determine the allowable operating conditions for our large tank by studying the behavior of a geometrically similar model system, but using water as the working fluid.

a. For strict dynamic similarity, what should be the scale-down ratio of the scale model?

b. How does the draw-off rate (volumetric flow rate) scale between the model and full-size tanker?



What You Should Learn From Problem Set 9

Problem 1: Streamfunction: Trough drainage.

- This problem gives you practice in using the streamfunction in cylindrical coordinates.
- Demonstration that allowing the boundary conditions to suggest the form of the solution results in a dramatic simplification of the solution procedure.
- Introduction of the general separable solution for the streamfunction in cylindrical geometries.

Problem 2: Film drainage.

- This is a nice example of film drainage, relevant to many coating problems.
- Demonstration that the "parabolic velocity distribution" can lead to complex behavior in the presence of a free surface.
- Practice simplifying problems using (once again) the flat earth limit.
- Analytic solution to a blow-up problem (the drip on the bottom).

Problem 3: Dimensional Analysis.

- Use dimensional analysis to determine the dimensional groups a problem depends on.
- Practice using additional physical insight to get much stronger dimensional analysis results, including estimates of the film thickness in two very dissimilar flow regimes.

Problem 4: Reynolds Lubrication Equation (defined pressure BCs).

- Fish out the solution from the notes and apply it with numbers.
- Calculate the drag and compare to frictional resistance: it should be *much* smaller!
- Demonstrate that the "unknown O(1) constant" left over from scaling is not always close to 1 — but scaling still gets you most of the way there!

Flow inside a 90º trough

Subject Tags: creeping flow; polar coordinates; streamfunction; biharmonic equation; symmetry

Difficulty Level: Moderate

Estimated Time: 30 minutes

This problem considers the flow in a V-shaped trough with a 90º internal angle and a center slit from which fluid is exiting. Assuming negligible inertia (low Reynolds number), we seek to determine the streamfunction and velocity distribution using symmetry arguments and solutions to the biharmonic equation in polar coordinates.

Skills Tested:

- boundary conditions
- streamfunction formulation
- eigenfunction solution
- velocity extraction

Source: CBE 30355 Problem Set 9 **File:** hw0091.tex

Flow inside a 90º trough

Consider the flow depicted below. A trough with an internal angle of 90º (e.g., the walls are at $\theta = \pm\theta_0 = \pm\frac{\pi}{4}$) is leaking out through a slit in the center with flow rate per unit extension into the paper Q/W. We want to solve for the velocity distribution. If we can ignore the inertial terms ($\mathrm{Re} \ll 1$), the flow is governed by the Biharmonic Equation.

a. Write down the boundary conditions which govern the streamfunction for this problem.

b. We anticipate that the streamfunction will be of the form

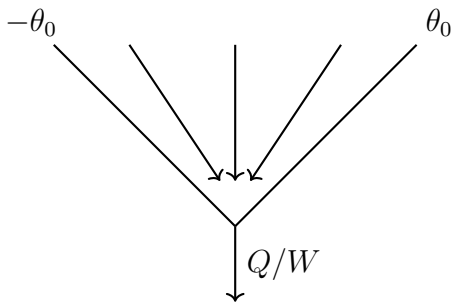
$$\psi = r^\lambda f_\lambda(\theta).$$

From the boundary conditions, what must λ be?

c. Solve for $f_\lambda(\theta)$. It is helpful to make maximum use of symmetry!

d. What is the radial velocity as a function of r and θ? Is there any flow in the θ direction?

Hint: The four homogeneous solutions for $f(\theta)$ are 1, θ, $\sin(2\theta)$, and $\cos(2\theta)$.



Film Drainage Flows

Subject Tags: film drainage; lubrication approximation; free surface flow; asymptotic analysis; gravity-driven flow

Difficulty Level: Advanced

Estimated Time: 45 minutes

In this problem, we examine the gravity-driven drainage of a thin viscous film from a cylindrical surface. The goal is to understand the evolution of the film thickness over time and position, derive a PDE governing the behavior, and estimate when and where rupture (dripping) occurs. The film is assumed thin compared to the cylinder radius and governed by creeping flow dynamics.

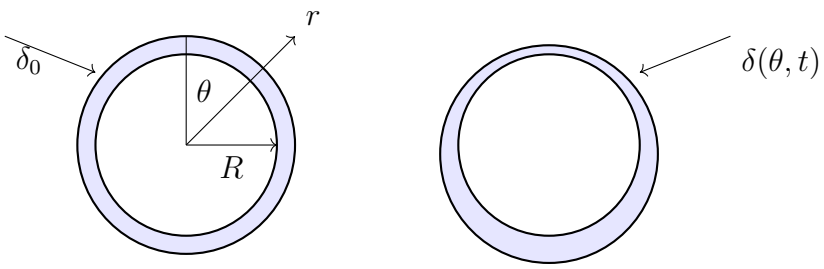
Skills Tested:

- velocity profile derivation
- time scaling
- PDE formulation
- film rupture analysis

Source: CBE 30355 Problem Set 9 **File:** hw0092.tex

Film Drainage Flows

In this problem we examine the problem of drainage from a cylinder of radius R coated with a layer of fluid of density ρ and viscosity μ, and with initial thickness $\delta_0 \ll R$. We wish to determine the evolution of the thickness of the layer as a function of θ and t.



a. Redraw your coordinates for some value of θ in the flat earth limit (e.g., Cartesian coordinates!) and solve for the velocity profile in the draining film. Remember that g is now a function of θ! This should otherwise be identical to the falling film problem we solved in class. Note that $\delta \neq \delta_0$ as the film drainage evolves, rather that is just its initial condition!

b. Recognizing that the time derivative of the film thickness is just the radial velocity (or y-velocity in your "flat earth" coordinate system) evaluated at $\delta(\theta, t)$, use the continuity equation to determine the timescale of the drainage problem t_c.

c. Now integrate the continuity equation to obtain the dimensionless partial differential equation that the film thickness must obey.

d. Solve for the time for the first drip to form (e.g., the solution blows up) at $\theta = \pi$ (the bottom of the cylinder). Note that the $\sin(\theta)$ term disappears at $\theta = \pi$, thus at the bottom (and at the top) the thickness is just a function of time.

Dimensional Analysis of Flow Down an Inclined Plane

Subject Tags: inclined flow; dimensional analysis; Reynolds number; asymptotic limits; free surface

Difficulty Level: Moderate

Estimated Time: 30 minutes

This problem applies dimensional analysis to determine the thickness of a gravity-driven flow down an inclined plane. By identifying dimensionless groups (including the Reynolds number), and considering both low and high Reynolds number limits, we develop scaling relationships between the layer thickness and flow rate. This is a two-dimensional, free-surface problem.

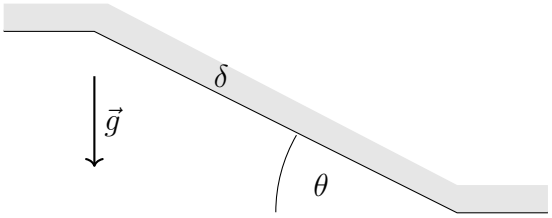
Skills Tested:

- dimensional matrix construction
- Reynolds number scaling
- asymptotic limit interpretation

Source: CBE 30355 Problem Set 9 **File:** hw0093.tex

Dimensional Analysis of Flow Down an Inclined Plane

Consider the inclined plane depicted below. The plane is inclined by an angle θ from the horizontal, and a fluid (viscosity μ, density ρ) is flowing down the plane with flow rate Q/W per unit width of the plane (normal to the plane of the paper — this is a two-dimensional problem). We wish to determine the thickness of this fluid layer as a function of the parameters of the problem using dimensional analysis.



a. Form a dimensional matrix and prove that the problem involves just three dimensionless groups. Determine three independent dimensionless groups. **Hint:** one will be the Reynolds number.

b. At low Reynolds numbers we anticipate that the flow rate will be proportional to gravity. Use this to strengthen the result of the dimensional analysis, and determine the relationship between the flow rate and fluid thickness to within some unknown function of the angle of inclination. The thickness is proportional to what power of Q/W?

c. At very high Reynolds numbers we expect the flow to be fully turbulent and to no longer depend on the viscosity. Use this to determine a new relationship between the thickness and the flow rate in the high Re limit.

The Sliding Block

Subject Tags: lubrication theory; lift force; drag force; friction; dimensionless analysis

Difficulty Level: Advanced

Estimated Time: 45 minutes

This problem investigates the hydrodynamic lubrication of a sliding block on an inclined plane, focusing on the fluid dynamics that generate lift and reduce friction. You'll determine the velocity required to support a load, compare lubrication drag to dry friction drag, and analyze the dimensionless lift behavior as a function of geometry. This highlights the physical principles behind lubrication-supported sliding motion.

Skills Tested:

- lubrication scaling
- shear stress estimation
- drag comparison
- asymptotic analysis

Source: CBE 30355 Problem Set 9 **File:** hw0094.tex

The Sliding Block

a. For the sliding block considered in class with dimensions $L = 20\,\text{cm}$, $W = 30\,\text{cm}$, and $d_2 - d_1 = 0.01\,\text{cm}$ (e.g., a fixed inclination angle), determine the velocity U at which the block can support a weight of 100 kg while maintaining a minimum separation $d_1 = 0.01\,\text{cm}$. Take the fluid viscosity to be 1 poise.

b. Calculate the drag on the block when it is fully lubricated at the conditions specified above (e.g., the lubrication equations apply!) and compare this to the drag when the lubrication fails (e.g., the drag is solely due to friction). Assume a coefficient of friction of 0.11, a typical value. **Hint:** it is much easier to determine the lubrication drag by calculating the shear stress on the plate — if you evaluate it on the block, you would also have to consider the contribution of pressure forces due to the slight incline angle of the block. Doing it right yields the same result either way, though.

c. The dimensionless lift is solely a function of the ratio $\Delta d/d_1$, as derived in class. We know that it has an asymptotic value of $0.5\,\Delta d/d_1$, but there should also be a maximum! Plot the dimensionless lift as a function of this ratio, compare it to the asymptotic relationship, and determine the optimum value of $\Delta d/d_1$ as well as the maximum lift. If you are using Matlab, use the `grid on` and `axis` commands to make the plot clear (or Python equivalent, of course).

What You Should Learn from Problem Set 10

- **Problem 1: Scaling of the energy equations.**
  - This is a key problem demonstrating the parallels between the momentum boundary layers discussed in class with the simpler (because it's linear) thermal boundary layer problem.
  - Practice scaling equations and using them to estimate desired quantities (such as wall temperature in this case) without actually having to *solve* the problem!
  - Demonstration that boundary layer thickness doesn't always scale as $L^{1/2}$!
- **Problem 2: Analysis of the momentum equations in cylindrical coordinates.**
  - This is a really nice example of a scaling problem where, again, you can learn a lot about a system (e.g., characteristic times and the dominant physical mechanisms) without actually solving the problem.
  - In part a you are looking at diffusion lengths and diffusion times for transient problems. This is analogous to pretty much all uni-directional startup (transient) problems.
  - In part b you are determining the scaling of a steady state boundary layer.
  - In c & d you are using the scaling analysis to determine the dominant mechanism for spin up. You can use exactly the same procedure (scaling of equations, determination of characteristic times, lengths, etc.) to (relatively) quickly determine what is going on in a problem without having to solve the whole thing completely! This is something I want you to apply in all of your engineering problems, or at least where appropriate!
- **Problem 3: Bernoulli's Equation / Euler flow:**
  - This is an example of why you have to tie down your roof in strong winds...
  - Reinforcing the issue of non-ideality in high Re flows (e.g., D'Alembert's Paradox).
- **Problem 4: Bernoulli's Equation Problem:**
  - A demonstration of the unexpected results sometimes obtained by application of Bernoulli's equation to inviscid flow.
  - Demonstration of how different geometries can lead to similar (or in this case identical) results.

Scaling Of The Transport Equations

Subject Tags: thermal boundary layer; energy transport; scaling analysis; convective-diffusive balance

Difficulty Level: Moderate

Estimated Time: 40 minutes

This problem explores the analogy between momentum and energy transport by scaling the energy equation for a plane Couette flow past a flat plate subject to a stepwise heating condition. Students identify appropriate characteristic scales and determine the governing nondimensional groups, arriving at a simplified model for the thermal boundary layer. It concludes with a scaling prediction for wall temperature downstream.

Skills Tested:

- nondimensionalization
- parameter identification
- thermal layer simplification

Source: CBE 30355 Problem Set 10 **File:** hw0101.tex

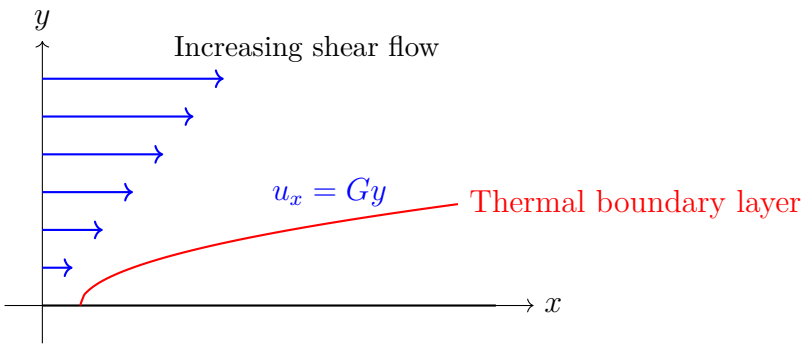
Scaling Of The Transport Equations

Energy transport is analogous to momentum transport, and the equation governing energy transport may be scaled in the same way as the momentum equations were in class. Consider flow past the flat plate $y = 0$ as depicted below. Fluid with a temperature $T = 0$ flows along the plane in the x direction with velocity $u = Gy$, where G is the shear rate (e.g., plane Couette flow without the upper plane). The plate is maintained at a temperature $T = 0$ for $x < 0$ and a constant heat flux q_0 for $x > 0$. The governing equations and boundary equations are given below.

$$Gy\frac{\partial T}{\partial x} = \alpha\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right)$$

with boundary conditions:

$$T|_{y=0,\,x<0} = 0, \quad -k\frac{\partial T}{\partial y}\bigg|_{y=0,\,x\geq 0} = q_0, \quad T \to 0 \text{ as } y \to \infty$$



a. Using L as the characteristic length in the x direction and δ the length in the y direction, render the energy transport equations dimensionless. For what value of δ is convection (the left-hand side) balanced by a diffusive term (on the right-hand side)? Using this scaling, what dimensionless parameter must be small to neglect one of the two diffusion terms (the dimensionless parameter should no longer involve δ)? Doing this, obtain the simplified thermal boundary layer equation for this problem.

b. Using scaling analysis, determine the expected temperature of the plate (i.e., at $y = 0$) at $x = L$.

Scaling Of The Transport Equations

Subject Tags: thermal boundary layer; energy transport; scaling analysis; convective-diffusive balance

Difficulty Level: Moderate

Estimated Time: 40 minutes

This problem explores the analogy between momentum and energy transport by scaling the energy equation for a plane Couette flow past a flat plate subject to a stepwise heating condition. Students identify appropriate characteristic scales and determine the governing nondimensional groups, arriving at a simplified model for the thermal boundary layer. It concludes with a scaling prediction for wall temperature downstream.

Skills Tested:

- nondimensionalization
- parameter identification
- thermal layer simplification

Source: CBE 30355 Problem Set 10 **File:** hw0101.tex

The Tea Leaf Problem

A nice demonstration of both boundary layers and inertial secondary currents is the Tea Leaf Problem. As will be demonstrated in class: if a pot (or beaker) with tea leaves is rotated, the leaves (which settle on the bottom) are initially forced to the sides, but when the beaker is stopped, the leaves accumulate in a puddle at the center of the bottom. You can observe this in a teacup after stirring. Here we examine the mechanism by which a beaker full of water, initially at rest, spins up when the beaker is rotated. We look at the effect of both the side walls of the cylindrical beaker and the bottom.

a. Consider a cylinder of radius R without a bottom where the fluid is initially at rest. At time $t = 0$ we rotate the cylinder with an angular velocity Ω. Eventually the entire fluid in the cylinder will also rotate with this velocity. By scaling the appropriate terms in the equations of motion, estimate how long this will take. If $R = 5\,\text{cm}$ and the fluid is water, what is the numerical value of this characteristic time?

b. Now for the hard part! Let us put a bottom on the cylinder (e.g., now it is a beaker). When we start to rotate the beaker, a boundary layer sets up on the bottom, which very quickly approaches steady state. This boundary layer accelerates fluid in the θ direction (via diffusion of momentum from the bottom) which is then thrown outwards radially via centrifugal force. By scaling the equations (both r and θ momentum this time), determine the characteristic radial velocity and the boundary layer thickness.

c. The boundary layer is essentially pumping accelerated fluid into the rest of the beaker. If the fluid height in the beaker is H, determine the characteristic spin-up time (e.g., how long it takes the boundary layer to fill up the vessel) as a function of the parameters in the problem.

d. If H is 10 cm and Ω is 20 radians/s, what is the numerical value of this spin up time, and how does it compare to the answer in part a?

Potential Flow Past a Cylinder

Subject Tags: potential flow; high Reynolds number; lift and drag; flow separation

Difficulty Level: Moderate

Estimated Time: 30 minutes

This problem investigates lift and drag on a cylindrical object in an inviscid, high-Reynolds-number flow. The assumptions of potential flow and flow separation are used to illustrate how pressure distributions create lift and drag forces.

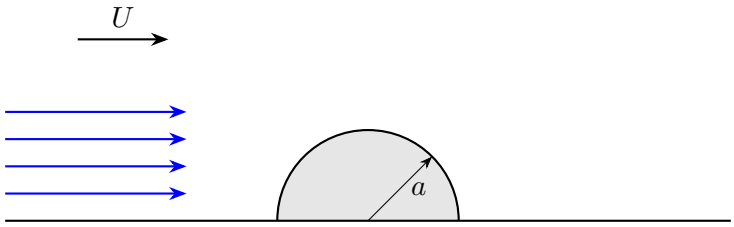
Skills Tested:

- force calculations from pressure fields
- assumptions in high-Re flows

Source: CBE 30355 Problem Set 10 **File:** hw0103.tex

Potential Flow Past a Cylinder

Consider uniform, high Reynolds number flow past a cylinder of radius a embedded in a plane as depicted below. Note that we are ignoring any no-slip condition on the plane itself, as well as on the cylinder.



a. For ideal potential flow, calculate the lift and drag on the cylinder. Assume that the pressure inside the cylinder is just the undisturbed pressure p_0 far from the cylinder.

b. Now assume that the flow separates at the top of the cylinder and that there is no pressure recovery from the ideal potential flow solution pressure minimum. What is the magnitude of the lift and drag under these assumptions? Note that the actual lift and drag will be a bit less than these values, but they are reasonable estimates.

Conical Cork in Flow

Subject Tags: inviscid flow; Bernoulli equation; variable area flow; integration in conical coordinates

Difficulty Level: Moderate

Estimated Time: 30 minutes

A conical cork is used to control airflow through a conical opening. This problem uses Bernoulli's equation and a conical coordinate system to analyze whether the cork will be blown out. Key features include transformation to streamline coordinates and evaluating pressure forces over a conical surface.

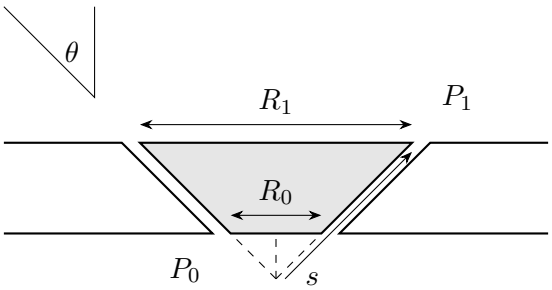
Skills Tested:

- use of Bernoulli and continuity in conical flows
- coordinate transformations
- force integration

Source: CBE 30355 Problem Set 10 **File:** hw0104.tex

Conical Cork in Flow

A conical cork is used to control the flow of air through a conical hole as depicted below. For what values of R_1 and R_0 will the plug be blown out of the hole? The flow is considered to be ideal and inviscid, and the cork is massless. You should find that the force on the cork is independent of both the conical angle θ and the width of the gap surrounding the cork.



a. Assuming parallel flow in the gap, use Bernoulli's equation and continuity to determine the velocity in the gap. To simplify the algebra, use a coordinate system s defined from the imaginary vertex of the cone, calculate $P(s)$, and integrate over s from s_0 to s_1 where

$$s_0 = \frac{R_0}{\sin\theta}, \qquad s_1 = \frac{R_1}{\sin\theta}.$$

b. Determine the net pressure force on the cork. What condition must be satisfied for the cork to be blown out of the hole? Show that this condition is independent of θ and the gap width.

What you should learn from these problems

Problem 1: Heat Loss from a plate in shear flow.

- This problem, as well as the next two, demonstrates the use of similarity transforms to convert PDE's to ODE's. Lots of practice using Morgan's Theorem and affine stretching!
- Practice actually "getting the number" - the O(1) unknown constant left over after scaling.
- Practice analytically solving a linear second order ODE, this time yielding a function a bit more complicated than a parabola... It is interesting to note the asymptotic form of the solution to this sort of problem. The solutions decay away to zero exponentially as you move away from the plate, typical for boundary layer problems. That's why "infinity" doesn't have to be all that large numerically: it's a fairly abrupt transition as you move out of the boundary layer.

Problem 2: Flow past a flat plate with suction.

- This is a nice scaling problem, demonstrating that you don't actually have to suck all that much fluid out of a boundary layer to affect its size. This is one of the three simple approaches to controlling boundary layer growth and delaying separation, the others being injection (why flaps on wings have small gaps between wing elements), and vortex generation (promoting turbulence in boundary layers, also delaying separation). Suction has been used, but not much: imagine what would happen to a plane if the holes get plugged!
- Practice breaking a problem into a set of first order ODE's and solving them numerically using the shooting method.

Problem 3: Rotating Disk Flow:

- This is the classic von Karman similarity problem. The similarity transform used here actually works even for parallel plate flow, although then the pressure gradient in the radial direction wouldn't be zero (instead you would have a no net radial flow condition that fixes the radial pressure gradient).
- A bit more practice solving a problem numerically, somewhat more involved than the flat plate problem.
- Once again demonstrating that the scaling (the easiest part) provides an excellent estimate of the desired quantity (total flux into the boundary layer and boundary layer thickness).

Heated Plate in Shear Flow: Similarity Solution and Heat Loss

Subject Tags: boundary layers, similarity solutions, heat transfer, scaling, differential equations

Difficulty: Advanced

Estimated Time: 90 minutes

Summary: This problem simplifies the thermal boundary layer problem encountered in shear flow past a heated flat plate. It asks the student to derive a similarity solution using coordinate stretching, solve the transformed differential equation analytically or numerically, and evaluate the heat loss from the plate. The physical setup involves Couette flow past a plate with a step change in surface temperature, and requires understanding of similarity transformations and integral evaluation.

Skills Tested:

- similarity methods
- integral transforms
- boundary layer analysis
- numerical evaluation of integrals

Source: CBE 30355 Problem Set 11 **File:** hw0111.tex

`https://diffusivepress.org/courseware/cbe30355/`

Similarity Solution and Heat Loss from a Heated Plate in Shear Flow

Last week you scaled the problem of shear flow past a heated plate. This week you will solve a somewhat simpler version of it.

a. By using the coordinate stretching technique illustrated in class, show that the boundary layer problem described below admits a similarity solution and obtain the similarity rule and similarity variable. Obtain the transformed ODE and boundary conditions. How does the thickness of the thermal boundary layer grow as it moves down the plate?

b. Solve the ODE. Note that $\frac{f''}{f'} = (\ln f')'$. You may leave the final result in terms of an explicit integral of a known function, or you may evaluate the integral in terms of the incomplete gamma function (you can look it up in a handbook, or online). Obtain a similar explicit relationship for the heat loss from the plate as a function of the length of the plate. Note that nearly all aspects of the solution except the final numerical value may be learned without explicitly solving the equation.

c. Evaluating the integral above, get the $\mathcal{O}(1)$ constant $f'(0)$, and thus the numerical value of the heat loss. You can solve the whole differential equation numerically as discussed in class, evaluate the integral from part (b) numerically or via Mathematica, or evaluate the gamma function integral. All methods work fine, just say how you got your answer!

Recall that the problem was flow past the flat plate $y = 0$ as depicted below. Fluid with a temperature $T = 0$ flows along the plane in the x-direction with velocity $u = Gy$, where G is the shear rate (e.g., plane Couette flow without the upper plane). The plate is maintained at a temperature $T = 0$ for $x < 0$ and a dimensionless temperature $T = 1$ for $x > 0$. The governing equations and boundary conditions are given below.

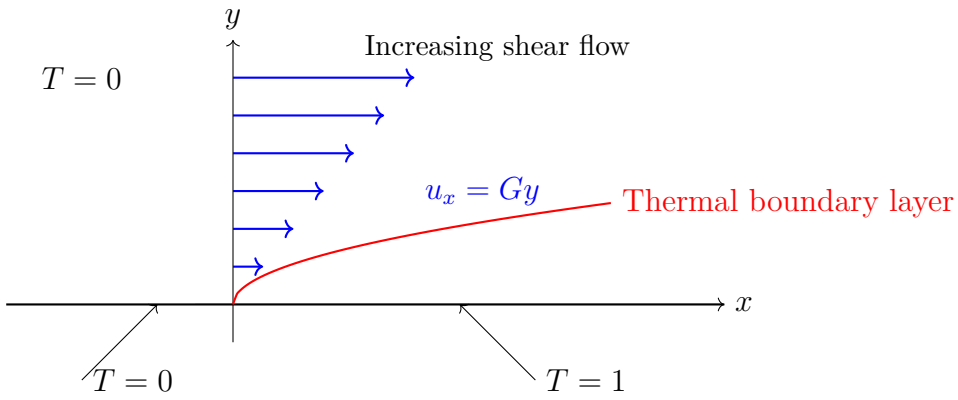
$$\frac{\partial T}{\partial x} Gy = \alpha \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right)$$

Boundary conditions:

$$\begin{aligned} T &= 0, \quad && y = 0,\ x < 0 \\ T &= 1, && y = 0,\ x > 0 \\ T &= 0, && y \gg 0 \end{aligned}$$

Heat loss from the plate:

$$Q/W = \int_0^L -k \frac{\partial T}{\partial y}\bigg|_{y=0} dx$$



Boundary Layer Growth with Suction

Subject Tags: boundary layer control, suction, Blasius equation, wall shear stress, scaling analysis

Difficulty: Advanced

Estimated Time: 90 minutes

Summary: This problem examines the use of suction as a technique to control boundary layer growth in steady flow over a flat plate. A power-law suction profile is applied through a porous plate, and the student is asked to determine the characteristic scale of the suction strength, analyze the effect on boundary layer thickness, and numerically solve a modified Blasius equation to determine the displacement thickness and wall shear stress.

Skills Tested:

- dimensional analysis
- boundary layer theory
- numerical methods
- similarity solutions

Source: CBE 30355 Problem Set 11 **File:** hw0112.tex

https://diffusivepress.org/courseware/cbe30355/

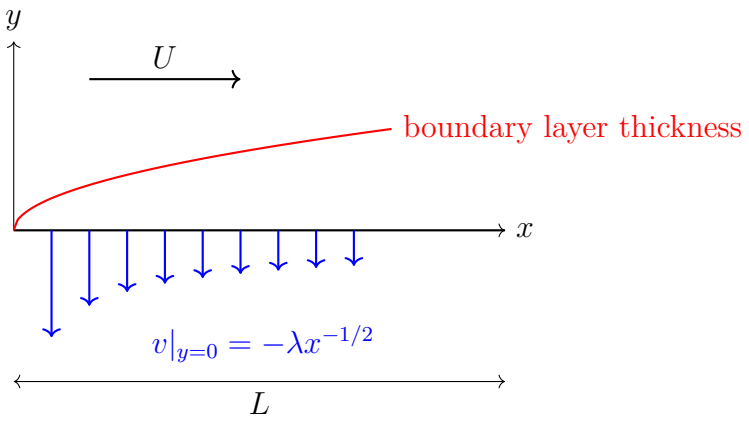
Boundary Layer Growth with Suction

Boundary layer growth with suction: One technique used to control the rate of boundary layer growth on airplane wings is suction – the wing (or plate) is porous, and fluid is sucked out of tiny holes which has the effect of keeping the boundary layer attached and preventing separation. In this problem we will examine the simple case of uniform flow past a flat plate where the vertical suction velocity is given by the power-law relation:

$$v|_{y=0} = -\lambda x^{-1/2}$$

a. What should be the characteristic magnitude of λ to affect the boundary layer thickness (e.g., how should it scale with U, μ, ρ, L, etc.) and what should be the magnitude of the total amount of gas withdrawal (the integral of v over the plate)?

b. Solve for the dimensionless displacement thickness and wall shear stress $f''(0)$ as a function of λ^* (e.g., λ divided by its characteristic scaling) and plot it up. Note that this will require a numerical solution to the Blasius Equation — where your boundary condition $f(0) = 0$ is replaced by one which involves λ^*.



Spinning Disk Electrode and von Kármán Flow

Subject Tags: electrochemistry, boundary layers, similarity solutions, von Kármán flow, numerical ODE solution

Difficulty: Advanced

Estimated Time: 120 minutes

Summary: This problem explores the hydrodynamics of the spinning disk electrode, a classic setup in electrochemistry and boundary layer theory. The flow induced by the rotation of the disk leads to a similarity solution originally derived by von Kármán. Students will estimate boundary layer thickness and flow rates, derive a reduced set of similarity ODEs, and solve them numerically using shooting and optimization methods. The ultimate goal is to assess how accurate the scaling estimate is compared to the full numerical solution.

Skills Tested:

- scaling analysis
- momentum balances
- similarity transformations
- numerical ODE integration

Spinning Disk Electrode and von Kármán Flow

An important experimental geometry in electrochemistry is the spinning disk electrode, in which a disk is spun rapidly in a fluid. The diffusion of momentum away from the surface of a disk imparts a centrifugal force which throws the fluid out radially, drawing fluid in axially. All three components of the velocity are non-zero (the Coriolis force matters too), however some components are larger than others. This problem is essentially the same as the tea leaf problem you looked at for homework last week, only now you are actually going to solve it!

a. Consider a disk of radius R spinning with angular velocity Ω in an infinite fluid at rest. By scaling the r and θ momentum equations and the continuity equation, estimate the boundary layer thickness and characteristic radial and vertical velocities as a function of the parameters in the problem.

b. We are primarily interested in the total flow rate of fluid into and out of the boundary layer. This is just the vertical velocity times the area of the disk! Based on your scalings, estimate the flow rate for a 5 cm radius disk spinning in water at an angular velocity of 100 radians/s. What is the characteristic boundary layer thickness for these values?

c. As could be demonstrated from either the scaling process or via simple affine stretching, this problem admits the classic von Kármán similarity solution: that

$$u_\theta^* = r^* f(z^*), \quad u_r^* = r^* g(z^*), \quad u_z^* = h(z^*).$$

Plug these into the r and θ momentum equations and the continuity equation to obtain a set of three ODEs for f, g, and h. Determine the five boundary conditions.

d. By breaking this problem into a set of five coupled first-order ODEs such as described in class, it is pretty straightforward to solve the problem numerically using `ode23.m` (or your favorite integrator) and `fminsearch.m` (or your favorite multidimensional optimization utility) to determine the two unknown initial conditions via the shooting method. The shooting method is more robust if you define an objective function which is the norm of the two "misses" and find the values which minimize it (it should be zero when you hit the target!). Do this and plot up f, g, and h.

Note: you can't directly integrate "to infinity" for this problem due to stability issues: the integration to large z^* is very strongly dependent on the unknown initial conditions and both the integrator and root finder tend to hang up. An easy way of fixing this is to iteratively increase your limit of integration (starting at some small initial value, such as 1) until the problem converges, using the last best fit values at the current limit as the initial guess for the next larger limit of integration. An example of such a code (and a bit fancier than required here) can be found in the senior lab experiment near the bottom of the page: `https://www.diffusivepress.org/courseware/cbe41459/`

e. You are primarily interested in the flow rate of fluid into the boundary layer. This is just the limit of u_z^* at large z^* times its scaling and the disk area. What is this value from your numerical solution? How much does the exact flow rate deviate from the answer you got for part (b), where you took the unknown $\mathcal{O}(1)$ constant to be one?

What You Should Learn from Problem Set 12

Problem 1: The Jetlev Flyer.

- This problem set is all about pipe flow, something you may well encounter as an engineer (or as a homeowner dealing with plumbing issues!). In the first part of this one you are to determine the flow rate necessary to achieve a desired thrust. Draw your control volume carefully!
- Once you have a flow rate, and if you know the piping system, you just use Bernoulli's equations (plus losses!) to determine the pressure requirements at the pump. That pressure times the flow rate is the energy requirement!
- The last bit is much trickier: Now you have to use the stuff on vectors and forces we learned at the beginning of the semester to do a momentum balance in two directions to determine the boat velocity. While you would obviously do a more careful analysis if you were designing the system (and getting paid to do so!) you actually should get pretty close to the design specs of the real system!

Problem 2: A drain problem.

- This is a nice example of pipe sizing: If you know what the flow rate is, and the driving force, you need to figure out what sort of pipe diameter to use.
- Pressure drops are pretty massively influenced by pipe diameter (for a fixed flow rate), so if you set up the problem right you can use functional iteration to calculate the diameter really quickly without having to use a computer...

Problem 3: Pump Curves:

- Knowing what a pump can do is pretty important if you are designing a flow system. This problem gets you to figure out what all the lines on a standard pump diagram mean and how to use them!
- There will be a pump curve problem on the final...

Jetlev Flyer Pump Requirements

Subject Tags: fluid mechanics, energy balance, pipe losses, thrust, drag

Difficulty: Advanced

Estimated Time: 90 minutes

Summary: This problem analyzes the mechanical and fluid dynamic requirements of a water-propelled jetpack device. Students must account for pipe fitting losses, weight support, and elevation head to estimate the necessary flow rate and pump power. The second part considers the thrust developed when the flyer is nearly horizontal, leading to towing of the pump boat. This requires applying fluid drag relationships and understanding how power is redirected toward propulsion.

Skills Tested:

- energy equation
- pipe flow resistance
- mechanical efficiency
- thrust and drag estimation

Source: CBE 30355 Problem Set 12 **File:** hw0121.tex

Jetlev Flyer Pump Requirements

In this problem you are asked to analyze the pump requirements of the Jetlev flyer jetpack. The jetpack is depicted below:

Figure 24: Source: https://jetlev-flyer.com

The pipe fittings on the back are fairly complex, consisting of a reducing Y-connector, a 135 degree bend, and a nozzle for each side. As a reasonable approximation, assume that the K value for the Y-connector is the geometric mean of a T straight through and through the side coupled with a smooth pipe reduction. The bend can be considered the sum of 45 and 90 degree short angle bends. The main feed pipe from the boat is 10 meters of 4" ID pipe, the pipe coming off of the Y is 3", and the final nozzle is 1.5" pipe. The dry weight of the jetpack is 30lb, and the rider weighs 200lb. The dry weight of the feed pipe is 10lb.

OK, now for the problem:

a. What is the required flow rate and pump output pressure for a flyer who is at the maximum elevation of 8.5 meters (e.g., going straight up, but some of the feed pipe is

used up in making the 90 degree bend to vertical). If the pump is 25% efficient, what is the required horsepower of the pump boat engine?

b. If the flyer angles forwards so that he is just out of the water (e.g., an elevation of 2m), but the pump throttle is still at a maximum, you can get quite a bit of lateral thrust, towing the pumping boat along behind. Note that the flow rate will actually be a bit higher now, because not as much of the pump energy is used up by pumping the water to 8.5m. Also, assume that you are supporting only half the weight of the water-filled feed pipe. The drag on a boat is rather complex, and depends on the Froude number. A reasonable correlation at moderate Fr is given by:

$$C_D = \frac{Resistance}{\frac{1}{2}\rho U^2 V^{2/3}} \approx 0.035$$

where U is the velocity of the boat and V is the displaced volume. The $V^{2/3}$ scaling is a multiple of the area of water the boat is pushing against. If the pumping boat weighs in at 350kg, calculate the maximum forward velocity of the boat and jetpack.

Overflow Drain Design

Subject Tags: pipe flow, gravity drainage, hydraulics

Difficulty: Moderate

Estimated Time: 30 minutes

Summary: This problem requires determining the minimum required pipe diameter to accommodate a specified flow rate from an overflow tank. The analysis assumes gravity-driven flow through the drainpipe, and requires students to estimate appropriate head losses and apply pipe flow principles or empirical relations such as the Hazen-Williams or Darcy-Weisbach equations.

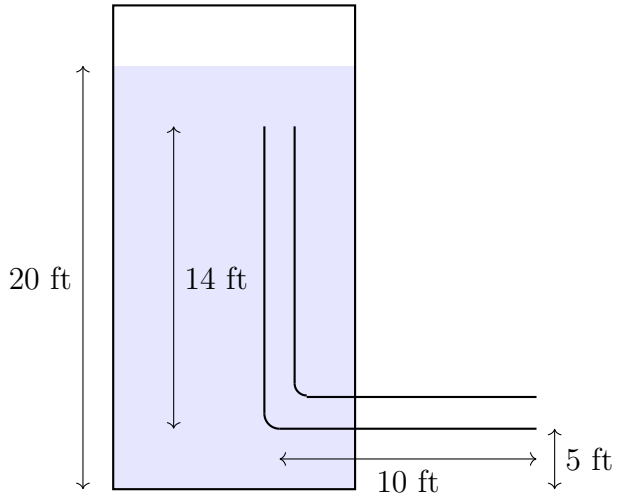
Skills Tested:

- open channel/pipe flow analysis
- head loss estimation
- flow rate calculation

Source: CBE 30355 Problem Set 12 **File:** hw0122.tex

Overflow Drain Design

You are designing an overflow drain for a tank as depicted below. It is required that the pipe must handle a flow rate of 1 gal/s. What is the minimum diameter of the drain pipe?



Pump Curve Analysis

Subject Tags: pipe flow, pump curves, energy balance, pump sizing, efficiency

Difficulty: Moderate

Estimated Time: 45 minutes

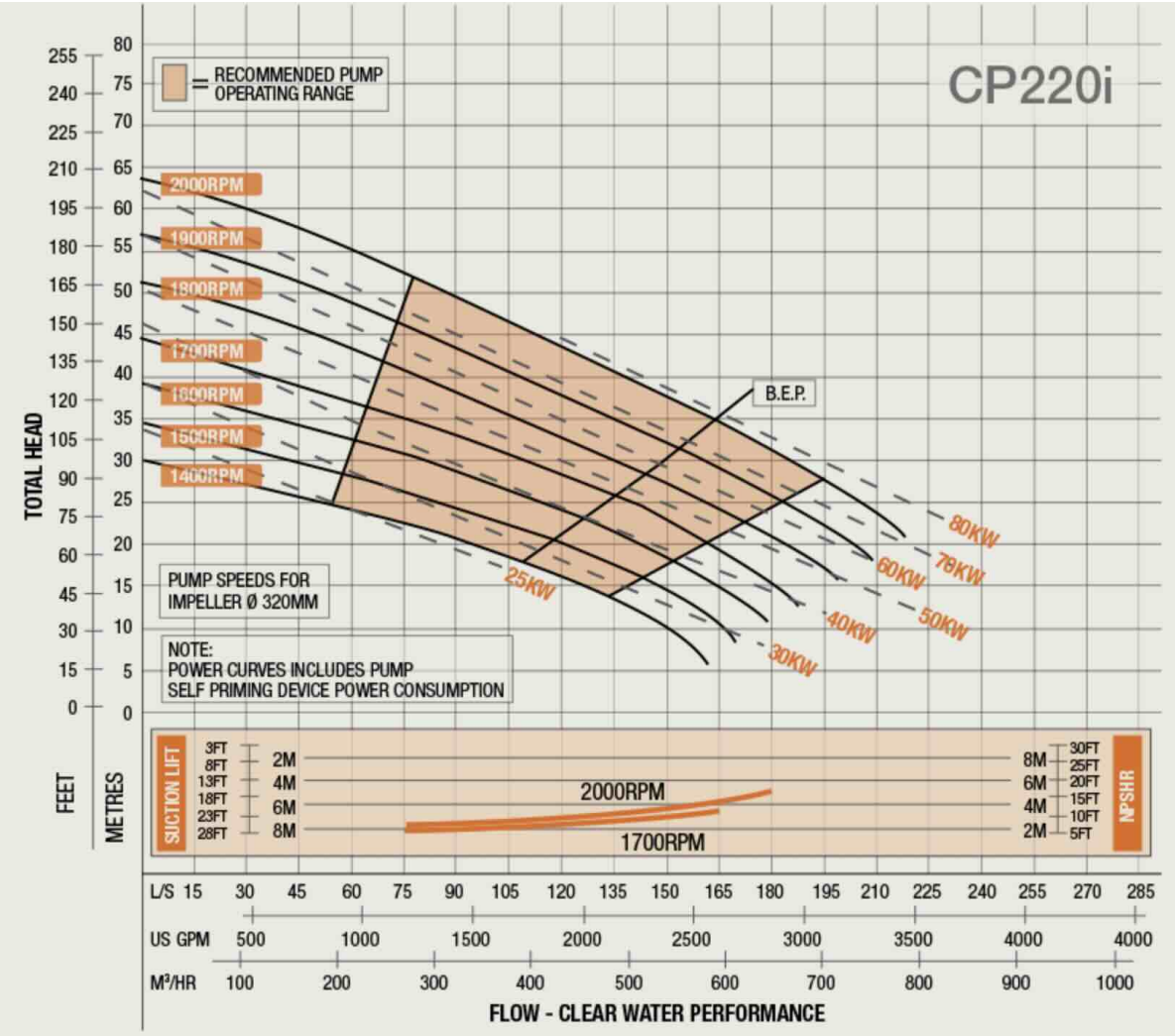
Summary: This problem focuses on analyzing the performance of a specific pump (CP200) to determine whether it is suitable for a given flow rate and elevation requirement. Students must account for pipe friction, minor losses, and acceleration effects in the system and then apply the energy equation and pump performance curve to assess the feasibility and efficiency of the setup.

Skills Tested:

- head loss estimation
- pump performance analysis
- energy equation application

Pump Curve Analysis

Pump Curves. It is desired to pump 80 liters/sec from a pond to a tank at an elevation of 20 meters through a 6" pipe. The pipe feeding the pump is 10m long, and the pipe leading to the tank is another 90m. There is a sudden contraction (K = 0.45) leading to the pump, and 3 45 degree elbows (K = 0.35 each) between the pump and the tank. Don't forget the acceleration of the fluid! Use the pump curve below to answer the following:



a. Is the pump CP220i recommended for the job?

b. What is the RPM required to do the job?

c. What is the work done by the pump on the fluid per unit time?

d. What is the efficiency of the pump at the operating conditions?

e. How far up the hill from the level of the pond can we put the pump? (Note: this could actually be negative.)

CBE 30356 Exam Questions

This section presents the problems from the CBE30356 Transport II examinations. In contrast to 355 which was taught by me for four decades, 356 was only taught by me twice. Thus, there are far fewer exam questions in this file. A forthcoming compilation of the graduate courses CBE 50544 and CBE 50545 taught by me for four decades will provide many more examples, and will be available on the `https://diffusivepress.org` website. The problems here are divided into midterm and final examinations.

CBE 30356 Mid-Term Examinations

In this section are the problems from the two mid-term examinations given by me in CBE 30356. Solutions are available on the `https://diffusivepress.org` website.

Dimensionless Groups

Subject Tags: Dimensionless Numbers, Heat Transfer, Momentum Transport

Difficulty Level: Moderate

Estimated Time: 15 minutes

Summary: This problem reinforces the student's understanding of various dimensionless groups commonly used in transport phenomena. The student is asked to match each of eight unnamed dimensionless groups with their corresponding physical descriptions and commonly accepted names.

Skills Tested:

- Recognition of physical significance of dimensionless numbers
- Matching physical descriptions with names of dimensionless groups
- Application of conceptual understanding without calculation

Dimensionless Groups

A recurring emphasis of this class is the use of dimensionless groups of parameters for describing phenomena. Listed below you will find 8 such groups we've used this term. For each of these, enter the letter corresponding to the correct physical description and name from the lists provided below.

#	Expression	Description (a–h)	Name (a–h)
1	$\frac{UD}{\nu}$	_____	_____
2	$\frac{UD}{\alpha}$	_____	_____
3	$\frac{hD}{k}$	_____	_____
4	$\frac{h_{\text{ext}}R}{k_{\text{int}}}$	_____	_____
5	$\frac{\nu}{\alpha}$	_____	_____
6	$\frac{h}{\rho C_p U}$	_____	_____
7	$\frac{\tau_w}{\frac{1}{2}\rho U^2}$	_____	_____
8	$\left(\frac{\omega a^2}{\nu}\right)^{1/2}$	_____	_____

Physical Descriptions (match with one letter *a*–*h*):

a. $\frac{Heat\,Transfer}{Convective\,Heat\,Transfer}$

b. $\frac{Heat\,Transfer}{Conductive\,Heat\,Transfer}$

c. $\frac{Inertial\,Forces}{Viscous\,Forces}$

d. $\frac{Convection\,of\,Energy}{Diffusion\,of\,Energy}$

e. $\frac{Wall\,Shear\,Stress}{Dynamic\,Pressure}$

f. $\frac{Tube\,Radius}{Penetration\,Length}$

g. $\frac{Momentum\,Diffusivity}{Energy\,Diffusivity}$

h. $\frac{Internal\,Heat\,Transfer\,Resistance}{External\,Heat\,Transfer\,Resistance}$

Dimensionless Group Names (match with one letter *a*–*h*):

a. Prandtl Number

b. Fanning Friction Factor

c. Stanton Number

d. Womersley Number

e. Nusselt Number

f. Biot Number

g. Reynolds Number

h. Thermal Peclet Number

Heat Conduction in Solids

Subject Tags: Heat Transfer, Thermal Resistance, Time Response, Conduction, Heat Transfer Coefficient

Difficulty Level: Moderate

Estimated Time: 20 minutes

Summary: This problem involves the analysis of heat conduction through a solid layer to determine the internal heat transfer coefficient in a vessel using a "heat transfer stethoscope." The student must derive a relation for h based on temperature measurements and thermal properties, and estimate the time response of the probe given its volumetric heat capacity.

Skills Tested:

- Derivation of internal heat transfer coefficient from steady-state conduction
- Understanding of thermal resistance analogies
- Estimation of dynamic time response of thermal systems

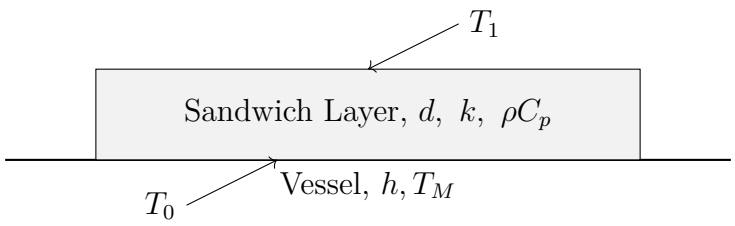
Source: CBE 30356 Mid-Term Exam 2022 **File:** problem1002.tex

Heat Conduction in Solids

Some of your classmates have been working on aspects of heat transfer in a melter/blender for HB Fuller. A key issue in improving the rate of heat transfer (and speeding up the processing time) is evaluating the heat transfer coefficient -inside- the mixer, rather tricky to do from the outside! It is proposed to make a "heat transfer stethoscope" as shown below. A sandwich (internal thickness d) is attached to the wall of the vessel, and the temperature inside the vessel and on both surfaces of the sandwich are measured (the outer part of the sandwich would have a controlled temperature).

a. If the three temperatures are T_M, T_1, and T_0 as shown, and the thermal conductivity of the material of thickness d making up the sandwich is k, derive an expression for the internal heat transfer coefficient h on the inside of the vessel.

b. The internal heat transfer coefficient could be a function of time. If the volumetric heat capacity of the material making up the sandwich is ρC_p, estimate the response time of the probe.



Convective Heat Transfer

Subject Tags: Convection, Nusselt Number, Heat Transfer, Asymptotic Analysis, Shear Flow

Difficulty Level: Advanced

Estimated Time: 30 minutes

Summary: This problem involves steady shear-driven convection with a constant wall heat flux. The student must derive the governing equations, nondimensionalize them, and determine the asymptotic temperature profile and Nusselt number. The physical scenario models heat transfer in a classic Couette flow with an imposed constant heat flux at the lower wall and insulation at the top.

Skills Tested:

- Formulation and nondimensionalization of heat transfer equations
- Asymptotic analysis of convection-diffusion systems
- Calculation of average and wall temperatures
- Evaluation of Nusselt number under non-uniform heating

Source: CBE 30356 Mid-Term Exam 2022 **File:** problem1003.tex

Convective Heat Transfer in Couette Flow

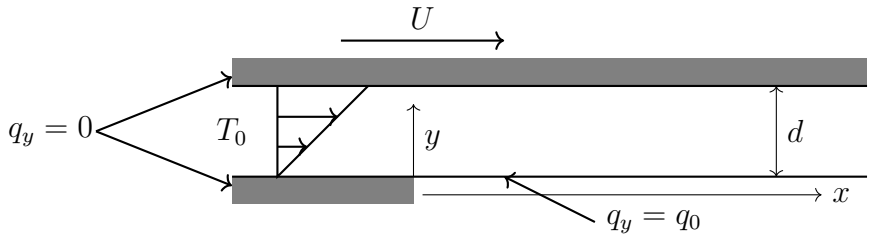
Consider the geometry depicted below. An upper plate at $y = d$ is moving with a velocity U in the x-direction, generating a simple shear flow between the two plates. The fluid is heated for all $x > 0$ with a constant heat flux q_0, and the upper surface is insulated (so the fluid is getting hotter with x!). We are interested in the asymptotic solution and Nusselt number.

a. Write down the equations and boundary conditions governing this problem and render them dimensionless.

b. Solve for the asymptotic temperature distribution at large x.

c. Using this, calculate the dimensionless flow average temperature and temperature at the wall.

d. Using these two values, calculate the asymptotic Nusselt number for these boundary conditions.



Boundary Layer Solution

Subject Tags: Boundary Layers, Scaling, Convection, Heat Transfer, Nusselt Number

Difficulty Level: Advanced

Estimated Time: 25 minutes

Summary: This problem complements the large-x asymptotic solution by exploring the small-x behavior of the thermal boundary layer. Students are asked to rescale the governing equations and boundary conditions, identify the x-dependence of the wall temperature through scaling arguments, and then estimate the local Nusselt number based on that scaling—all without solving the full ODE.

Skills Tested:

- Rescaling PDEs and boundary conditions for boundary layer analysis
- Applying scaling arguments to deduce temperature behavior
- Estimating Nusselt number from asymptotic reasoning

Source: CBE 30356 Mid-Term Exam 2022 **File:** problem1004.tex

Boundary Layer Solution

The local Nusselt number calculated in Problem 3 is only valid for large x. For small x, it is more appropriate to look at the boundary layer where energy hasn't had a chance to diffuse very far from the lower wall. In this problem we examine this limit.

a. Rescale the equation and boundary conditions in this boundary layer limit, rendering the problem dimensionless.

b. From scaling (or from affine stretching) determine how the wall temperature depends on x.

c. Recognizing that the bulk temperature in this limit is just T_0 (i.e., the boundary layer is thin), determine the local Nusselt number as a function of x to within an $\mathcal{O}(1)$ constant. Note that you are not asked to find the constant, as that would require solving the transformed ODE (which is not part of this problem). Scaling alone gets you almost all the way there!

Dimensionless Groups

Subject Tags: dimensionless groups; parameter ratios; heat and momentum transfer

Difficulty Level: Moderate

Estimated Time: 20 minutes

This problem tests your understanding of how physical mechanisms are represented by dimensionless groups. Each ratio corresponds to a commonly used group used to nondimensionalize transport equations or characterize physical regimes.

Skills Tested:

- Identification of dimensionless groups
- Interpretation of physical parameter ratios
- Matching parameters to physical mechanisms

Dimensionless Groups

A recurring emphasis of this class is the use of dimensionless groups of parameters for describing phenomena. Often we can regard dimensionless groups as ratios of physical mechanisms.

For each of the given ratios, determine the corresponding group of parameters from the list below. Note that there are a few "distractors" that won't appear!

a. $\dfrac{\text{Heat Transfer}}{\text{Convective Heat Transfer}}$

b. $\dfrac{\text{Heat Transfer}}{\text{Conductive Heat Transfer}}$

c. $\dfrac{\text{Inertial Forces}}{\text{Viscous Forces}}$

d. $\dfrac{\text{Convection of Energy}}{\text{Diffusion of Energy}}$

e. $\dfrac{\text{Wall Shear Stress}}{\text{Dynamic Pressure}}$

f. $\dfrac{\text{Tube Radius}}{\text{Penetration Length}}$

g. $\dfrac{\text{Momentum Diffusivity}}{\text{Energy Diffusivity}}$

h. $\dfrac{\text{Internal Heat Transfer Resistance}}{\text{External Heat Transfer Resistance}}$

Possible parameter ratios:

$$\frac{UD}{\alpha}, \quad \frac{UD}{\nu}, \quad \frac{h_{\text{ext}}R}{k_{\text{int}}}, \quad \frac{\tau_w}{\frac{1}{2}\rho U^2}, \quad \frac{U^2}{Lg}, \quad \left(\frac{\omega a^2}{\nu}\right)^{1/2}, \quad \frac{\nu}{D_{AB}}, \quad \frac{hD}{k}, \quad \frac{\nu}{\alpha}, \quad \frac{h}{\rho C_p U}, \quad \frac{\mu\dot{\gamma}a}{\Gamma}$$

Heat Conduction in Solids

Subject Tags: cylindrical coordinates; heat generation; energy balance; boundary conditions; steady-state conduction

Difficulty Level: Advanced

Estimated Time: 30 minutes

This problem addresses heat conduction from a radioactive energy source in a dry storage cask. You will use an energy balance and boundary conditions to determine the temperature distribution and centerline temperature in a system with internal heat generation and radial conduction.

Skills Tested:

- Application of cylindrical coordinates
- Energy balance with source term
- Boundary temperature conditions
- Derivation of centerline temperature

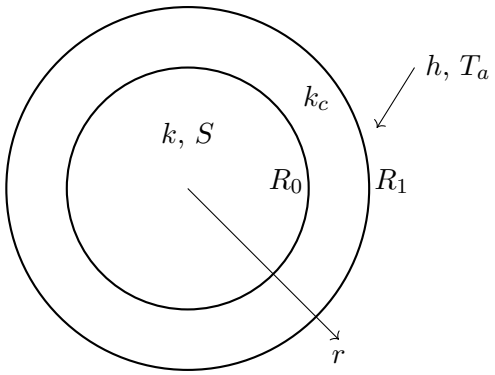
Source: CBE 30356 Midterm Exam 2023 **File:** problem1010.tex

Heat Conduction in Solids

Nuclear waste disposal is an important problem, complicated by the high level of radioactivity in a spent fuel rod that generates a lot of heat! After cooling in a pond for 5 to 20 years, sometimes these spent rods are transferred to a "dry cask" such as depicted below.

The radioactive material, with heat source per volume S, radius R_0, and conductivity k, is surrounded with a sealed cask of conductivity k_c and outer radius R_1. If the outside of the cask has a heat transfer coefficient h with the surroundings at temperature T_a, your goal is to figure out the temperature at the center of the radioactive energy source.

We'll simplify this problem by ignoring any conduction in the z (or axial) direction.



a. Using an energy balance, determine the total energy release (per unit length in the z-direction).

b. What is the temperature at the outer edge of the cask?

c. What is the temperature at the inner edge of the cask?

d. What is the temperature at the center of the radioactive heat source?

The equation of energy with source in cylindrical coordinates is:

$$\rho C_p \left(\frac{\partial T}{\partial t} + u_r \frac{\partial T}{\partial r} + \frac{u_\theta}{r} \frac{\partial T}{\partial \theta} + u_z \frac{\partial T}{\partial z} \right) = k \left[\frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial T}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 T}{\partial \theta^2} + \frac{\partial^2 T}{\partial z^2} \right] + S$$

And the heat flux components are:

$$q_r = -k \frac{\partial T}{\partial r}, \quad q_\theta = -\frac{k}{r} \frac{\partial T}{\partial \theta}, \quad q_z = -k \frac{\partial T}{\partial z}$$

Convective Heat Transfer/Scaling

Subject Tags: melting; heat transfer; external convection; energy balance; Nusselt correlation; scaling

Difficulty Level: Moderate

Estimated Time: 30 minutes

This problem involves modeling the melting of a sphere of ice using a convective heat transfer coefficient. You'll use an energy balance and a Nusselt number correlation to develop a time-dependent equation for the melting radius, then apply scaling to find the melt time. The process is assumed to be pseudo-steady with negligible internal resistance and no evaporation.

Skills Tested:

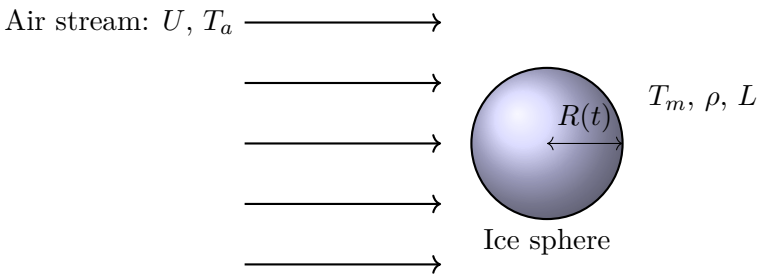
- Heat transfer modeling
- Dimensional analysis
- Energy balance with phase change
- Use of empirical correlations

Source: CBE 30356 Midterm Exam 2023 **File:** problem1011.tex

Convective Heat Transfer/Scaling

A sphere of ice of radius R_0 is suspended in a stream of air with velocity U and temperature T_a. Since $T_a > T_m$ (the melting temperature of the ice), the heat flux will cause the ice to melt, reducing the radius in time. This is a slow process, however, so we can use pseudo steady-state to figure out the melting rate. The melting process also causes the entire sphere of ice to be at an unchanging temperature T_m until it is all gone, thus all heat transfer is external.

We are also going to ignore both evaporative and condensation heat transfer of water (this is reasonable if the dew point of the air stream is at T_m). The latent heat of fusion of the ice is L (energy/mass) and the density is ρ.



The problem is essentially governed by the external heat transfer coefficient h. While many correlations can be used, for this problem assume that the Nusselt number is given by the correlation suggested in BSL at high Reynolds number (note that the diameter is twice the radius and changes in time!):

$$\mathrm{Nu} \approx 0.6\,\mathrm{Re}^{1/2}\mathrm{Pr}^{1/3}$$

a. Using an energy balance, write down the equation which describes the change in radius with respect to time.

b. Rendering R dimensionless with respect to R_0 (the initial radius) and t with respect to t_c, determine the scaling for the melt time, and how it depends on the parameters of the problem.

c. Solve the dimensionless differential equation to determine how long it takes for the ice to melt (e.g., get the $\mathcal{O}(1)$ number).

d. Suppose we humidify the air a bit (without changing the temperature or other material properties). Would this increase or decrease the melt rate? Briefly justify your answer.

Transient Boundary Layers

Subject Tags: heat conduction; similarity solution; boundary flux; dimensionless analysis; affine stretching

Difficulty Level: Advanced

Estimated Time: 30 minutes

This problem involves the unsteady heating of a semi-infinite slab where the boundary heat flux increases linearly in time. You will apply dimensional analysis and similarity transformations to derive the self-similar form of the governing equation and understand the behavior of the temperature at the wall over time.

Skills Tested:

- Transient diffusion modeling
- Dimensional analysis
- Similarity transformation
- Scaling arguments

Source: CBE 30356 Midterm Exam 2023 **File:** problem1012.tex

Transient Boundary Layers

A semi-infinite slab initially at a temperature T_0 is heated at $y = 0$ with a heat flux that increases linearly in time:

$$q_y\Big|_{y=0} = At$$

a. Write down the equation and boundary conditions which govern this problem.

b. Render the equations dimensionless, determining appropriate temperature and length scales in terms of the unknown time scaling.

c. Using the result of the scaling (or using affine stretching), show that the problem admits a self-similar solution, obtaining the similarity rule and variable in canonical form. You don't need to get the transformed ODE.

d. What is the temperature of the lower plate as a function of time (to within an unknown $\mathcal{O}(1)$ constant)?

CBE 30356 Final Examinations

In this section are the problems from the two final examinations given by me in CBE 30356. Solutions are available on the `https://diffusivepress.org` website.

An equation sheet was provided with these exams, and the LaTeX version is attached. This is the uncompressed version (which runs to four pages). You can have Chat make a compressed version that fits on a single page (with tiny, but readable print), but that requires different margins and will not compile properly with the same preamble as the rest of this document. The easiest solution is to either construct such a page and compile it separately (appending it to the exam pdf after generation) or simply grab the equations you would like to provide the students for their particular exam from this file and insert them into the problem statement. The LaTeX file (including the necessary preamble) which produces the one-page equation sheet is included in this Compendium in the file `cbe30356eqsheetcompressed.tex`.

Equation Sheet

Constitutive Relations

$$\tilde{q} = -k\nabla\tilde{T}$$
$$\boldsymbol{\tau} = \mu\left(\nabla\mathbf{u} + (\nabla\mathbf{u})^T\right)$$
$$\tilde{\mathbf{J}}_A = -cD_{AB}\nabla x_A$$

Useful Correlations and Equations

$$\mathrm{Nu} = \frac{hD}{k} \approx \left(2 + 0.4\,\mathrm{Re}^{1/2} + 0.06\,\mathrm{Re}^{2/3}\right)\mathrm{Pr}^{0.4}\left(\frac{\mu_0}{\mu_\infty}\right)^{1/4}$$
$$\mathrm{St} \equiv \frac{h}{\rho U_\infty \hat{C}_p} \approx \frac{f_f}{2}\,\mathrm{Pr}^{-2/3}$$

$$C_D \equiv \frac{F}{\frac{1}{2}\rho U^2 A}$$
$$\frac{F}{6\pi\mu U a} \approx 1 + \frac{1}{6}\mathrm{Re}^{1/2} + \frac{1}{60}\mathrm{Re}$$

$$U \equiv \frac{Q}{A} = \frac{a^2}{8\mu}\frac{\Delta P}{L} \quad \mathrm{Re} < 2100$$
$$f_f \equiv \frac{\tau_w}{\frac{1}{2}\rho U^2} \approx \frac{0.0791}{\mathrm{Re}^{1/4}} \quad 3000 < \mathrm{Re} < 10^5$$
$$\frac{1}{\sqrt{f_f}} \approx 4.0\log_{10}(\mathrm{Re}\sqrt{f_f}) - 0.40 \quad \mathrm{Re} > 3000$$

$$q_{rad} = e\sigma T^4$$

If Bi $= \infty$:

$$T^*(r^*, t^*) = 2\sum_{n=1}^{\infty}(-1)^{n-1}\frac{\sin(n\pi r^*)}{n\pi r^*}\exp(-n^2\pi^2 t^*)$$
$$\overline{T}^* = \frac{1}{\frac{4}{3}\pi}\int_0^1 T^*(r^*, t^*)\,4\pi r^{*2}\,dr^* = \frac{6}{\pi^2}\sum_{n=1}^{\infty}\frac{\exp(-n^2\pi^2 t^*)}{n^2\pi^2}$$

Fundamental Equations

$$\begin{aligned}
\tilde{\mathbf{N}}_A &= x_A\left(\tilde{\mathbf{N}}_A + \tilde{\mathbf{N}}_B\right) - cD_{AB}\nabla x_A \\
\mathbf{J}_A &= -cD_{AB}\nabla x_A = -\nabla c_A + x_A \nabla c \\
\frac{\partial c_A}{\partial t} + \nabla\cdot\tilde{\mathbf{N}}_A &= R_A \\
\frac{\partial c}{\partial t} + \mathbf{u}\cdot\nabla c &= D\nabla^2 c + R \\
\rho\left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}\cdot\nabla\mathbf{u}\right) &= -\nabla p + \mu\nabla^2\mathbf{u} + \rho\mathbf{g} \\
\rho\hat{C}_p\left(\frac{\partial T}{\partial t} + \mathbf{u}\cdot\nabla T\right) &= k\nabla^2 T + \dot{S}
\end{aligned}$$

Spherical Representation

$$\nabla^2 T \equiv \frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2\frac{\partial T}{\partial r}\right) + \frac{1}{r^2\sin\theta}\frac{\partial}{\partial\theta}\left(\sin\theta\frac{\partial T}{\partial\theta}\right) + \frac{1}{r^2\sin^2\theta}\frac{\partial^2 T}{\partial\phi^2}$$

$$\mathbf{u}\cdot\nabla T \equiv u_r\frac{\partial T}{\partial r} + \frac{u_\theta}{r}\frac{\partial T}{\partial\theta} + \frac{u_\phi}{r\sin\theta}\frac{\partial T}{\partial\phi}$$

$$\begin{aligned}
q_r &= -k\frac{\partial T}{\partial r} \\
q_\theta &= -k\frac{1}{r}\frac{\partial T}{\partial\theta} \\
q_\phi &= -k\frac{1}{r\sin\theta}\frac{\partial T}{\partial\phi}
\end{aligned}$$

$$\nabla\cdot\mathbf{u} = \frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2 u_r\right) + \frac{1}{r\sin\theta}\frac{\partial}{\partial\theta}\left(u_\theta\sin\theta\right) + \frac{1}{r\sin\theta}\frac{\partial u_\phi}{\partial\phi}$$

Cylindrical Representation

$$\nabla^2 T \equiv \frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 T}{\partial\theta^2} + \frac{\partial^2 T}{\partial z^2}$$

$$\mathbf{u} \cdot \nabla T \equiv u_r \frac{\partial T}{\partial r} + \frac{u_\theta}{r} \frac{\partial T}{\partial \theta} + u_z \frac{\partial T}{\partial z}$$

$$\begin{aligned} q_r &= -k \frac{\partial T}{\partial r} \\ q_\theta &= -k \frac{1}{r} \frac{\partial T}{\partial \theta} \\ q_z &= -k \frac{\partial T}{\partial z} \end{aligned}$$

$$\nabla \cdot \mathbf{u} = \frac{1}{r} \frac{\partial}{\partial r}(r u_r) + \frac{1}{r} \frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z}$$

Cartesian Representation

$$\nabla^2 T \equiv \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}$$

$$\mathbf{u} \cdot \nabla T \equiv u_x \frac{\partial T}{\partial x} + u_y \frac{\partial T}{\partial y} + u_z \frac{\partial T}{\partial z}$$

$$\begin{aligned} q_x &= -k \frac{\partial T}{\partial x} \\ q_y &= -k \frac{\partial T}{\partial y} \\ q_z &= -k \frac{\partial T}{\partial z} \end{aligned}$$

$$\nabla \cdot \mathbf{u} = \frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z}$$

Mathematical Operations

$$\text{If } \eta = \frac{y}{t^n}, \quad \text{then } \frac{\partial \eta}{\partial t} = -n \, \frac{\eta}{t}$$

$$\frac{d}{d\eta} f^n = n f^{n-1} \frac{df}{d\eta}$$

$$\frac{d \ln f(\phi)}{d\phi} = \frac{1}{f} \cdot \frac{df}{d\phi}$$

$$\frac{d}{d\phi} \left[f(\phi)^n \right] = n f^{n-1} \frac{df}{d\phi}$$

$$e^{it} = \cos t + i \sin t$$

$$\frac{dy}{dx} + p(x)y = f(x)$$

$$y = e^{-\int p(x)\,dx} \left[\int e^{\int p(x')\,dx'} f(x)\,dx + K \right]$$

Heat and Mass Transfer in a Stefan Tube

Subject Tags: Mass Transfer, Stefan Flow, Vapor Pressure, Heat Transfer, Evaporation

Difficulty Level: Advanced

Estimated Time: 40 minutes

Summary: This problem examines mass transfer in a Stefan tube used to measure diffusivity, and couples it with heat transfer considerations due to latent heat effects. Students must derive the molar flux under pseudo-steady assumptions, analyze thermal effects on equilibrium vapor pressure, and evaluate the temperature depression resulting from energy transport through a glass wall.

Skills Tested:

- Derivation of molar flux under pseudo-steady state conditions
- Energy balance under evaporation with latent heat
- Application of cylindrical conduction for radial heat transfer
- Dimensional consistency and unit conversions for property evaluation

Source: CBE 30356 Final Exam 2022 **File:** problem1005.tex

Heat and Mass Transfer in a Stefan Tube

As you all know, the Stefan tube is a useful way to measure the diffusivity of volatile compounds in air by determining the rate of liquid evaporation. In the implementation in Junior Lab, for example, a glass capillary is partially filled with diethyl ether. Air blows across the surface of the capillary to yield a mole fraction of zero at the top, while the mole fraction at the bottom is equal to the vapor pressure in equilibrium with the liquid divided by the atmospheric pressure. Because the partial pressure is a very strong function of temperature, the capillary is placed in a water bath so that the outer surface temperature of the capillary is controlled. So:

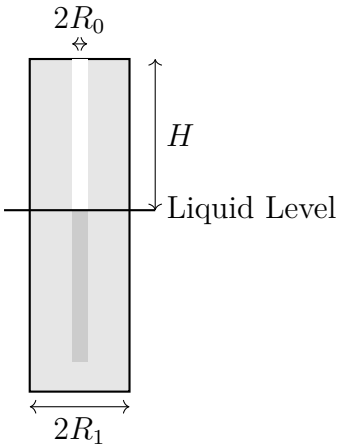
a. If the column of air is a height H above the liquid diethyl ether, derive the expression for the molar flux of the diethyl ether—the rate of evaporation of the liquid. Assume pseudo-steady state.

b. As you also know, evaporation takes energy! That means that the interface where evaporation takes place will be colder than the surroundings, and that will depress the vapor pressure at equilibrium (e.g., the "wet bulb" effect we saw for a falling drop of water). If the latent heat of evaporation is L, the thermal conductivity of the glass is k, the inner radius is R_0 and the outer radius is R_1 (enough greater than R_0 that you have to use cylindrical coordinates), develop an equation for the depression in the temperature at the interface. For purposes of estimation it is reasonable to assume that the energy required to vaporize the diethyl ether diffuses in radially via the glass capillary through a cylindrical surface of radius R_0 and length $2R_0$.

c. Evaluate the magnitude of this temperature depression for the following properties:

- $R_0 = 0.05\,\mathrm{cm}$
- $R_1 = 0.33\,\mathrm{cm}$
- $H = 1\,\mathrm{cm}$
- $D_{AB} = 0.09\,\mathrm{cm}^2/\mathrm{s}$
- $k = 0.8\,\mathrm{W}/(\mathrm{m}\cdot{}^\circ\mathrm{K}) = 0.8\times10^5\,\mathrm{g}\cdot\mathrm{cm}/(\mathrm{s}^3\cdot{}^\circ\mathrm{K})$
- $c = \dfrac{1}{24.4\times10^3}\,\mathrm{mol}/\mathrm{cm}^3$
- $L = 27\,\mathrm{kJ}/\mathrm{mol} = 27\times10^{10}\,\mathrm{erg}/\mathrm{mol}$
- $x_{A0} = 0.7$

Note that the effect of temperature on this last quantity (x_{A0}) is the primary source of error arising from the temperature depression effect!


$2R_0$
H
Liquid Level
$2R_1$

Local Nu in Shear-Heated Boundary Layer

Subject Tags: Convective Heat Transfer, Nusselt Number, Boundary Layer, Scaling Analysis

Difficulty Level: Advanced

Estimated Time: 35 minutes

Summary: This problem analyzes the early-stage thermal boundary layer that forms due to a constant wall heat flux beneath a shear-driven flow. It focuses on dimensional analysis, boundary layer scaling, and determination of the local Nusselt number under asymptotic limits at small x. Students identify the domain of validity for this approximation.

Skills Tested:

- Boundary layer scaling for thermal transport
- Non-dimensionalization of governing equations
- Extraction of Nusselt number from asymptotic behavior
- Estimating domain of applicability for similarity solutions

Source: CBE 30356 Final Exam 2022 **File:** problem1006.tex

Local Nu in Shear-Heated Boundary Layer

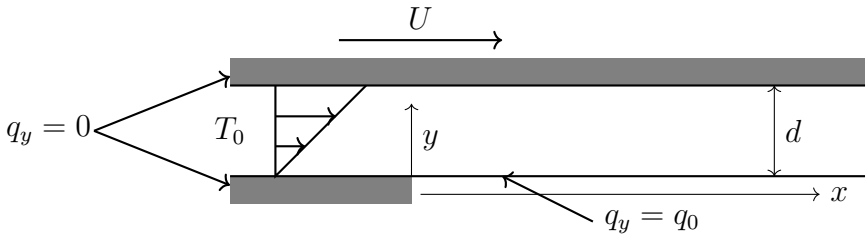
Consider the geometry depicted below. An upper plate at $y = d$ is moving with a velocity U in the x-direction, generating a simple shear flow between the two plates. The fluid is heated for all $x > 0$ with a constant heat flux q_0, and the upper surface is insulated (so the fluid is getting hotter with x!). We are interested in the local Nusselt number for small x (e.g., the boundary layer where energy hasn't had a chance to diffuse very far from the lower wall).

a. Scale the governing equation and boundary conditions in this boundary layer limit, rendering the problem dimensionless.

b. From scaling (or from affine stretching), determine how the wall temperature depends on x.

c. Recognizing that the bulk temperature in this limit is just T_0 (e.g., the boundary layer is thin), determine the local Nusselt number as a function of x, to within an $\mathcal{O}(1)$ constant.

d. What is the domain of validity (in x) over which this answer is valid (both lower and upper limits)?



Radiative Cooling of a Fin in Space

Subject Tags: Conduction, Radiation, Scaling, Heat Flux, Fins

Difficulty Level: Advanced

Estimated Time: 40 minutes

Summary: This problem models the radiative cooling of a long fin in deep space using a 1D conduction model. Students perform a shell balance or averaged formulation to derive the temperature profile, apply scaling arguments to identify a characteristic fin length and base heat flux, and solve a nonlinear ODE for the semi-infinite limit.

Skills Tested:

- Shell balance and effective 1D modeling
- Thermal scaling arguments
- Solution of nonlinear differential equations via energy integral trick

Source: CBE 30356 Final Exam 2022 **File:** problem1007.tex

Radiative Cooling of a Fin in Space

A furnace in deep space is cooled by a very long fin of cross-section A, surface area per unit length P (e.g., the perimeter of the fin cross-section), and thermal conductivity k via radiation. The base of the fin is maintained at temperature T_0, and you may neglect all radiation to the fin from the surroundings (deep space is cold!).

a. Neglecting all gradients across the cross-section, write down the equation governing the axial temperature distribution in the fin. You may derive this by averaging the heat transfer equation over the cross-section or by applying a shell balance.

b. (Most of the points!) Using scaling analysis, determine both a characteristic fin length (i.e., the axial length scale over which the temperature decays) and the characteristic heat flux at the base of the fin that provides cooling.

c. Solve the governing equation in the limit of an infinitely long fin to find the heat flux at the base. The equation is nonlinear, but can be solved by the following trick: multiply both sides by $\dfrac{dT^*}{dz^*}$, and show that both sides become perfect differentials. Integrate, apply the boundary condition at infinity, and determine the base heat flux.

Note: This trick also works with back radiation or forced convection in atmosphere, although the resulting equations are more complex.

Mass Transfer From a Falling Drop

Subject Tags: Mass transfer, Diffusion, Droplet dynamics, Henry's Law, Scaling

Difficulty Level: Advanced

Estimated Time: 45 minutes

Summary: This problem evaluates VOC removal in a spray tower by analyzing mass transfer from a falling droplet. It integrates gravitational motion, unsteady diffusion within a droplet, and internal circulation scaling to estimate pollutant removal effectiveness in atmospheric exposure.

Skills Tested:

- Terminal velocity estimation
- Transient diffusion modeling
- Nondimensionalization and scaling
- Internal circulation scaling via fluid mechanics

Source: CBE 30356 Final Exam 2022 **File:** problem1008.tex

VOC Removal by Droplet Evaporation in Spray Tower

A method for removing VOCs (volatile organic compounds) from ground water is a simple spray tower. The idea is that you spray the water into the air from some height over a pond and, if the drops are small enough, the VOCs will evaporate from the drop before it reaches the pond. In this problem, determine the fraction of the VOCs remaining in the drop when it hits the pond.

a. It is proposed to use a spray nozzle which emits 400 μm diameter drops. If the height of the tower is 10 m, how long does it take for the drops to reach the pond (ignoring drop-drop interactions)? Use the following properties for air:

$$\mu = 1.8 \times 10^{-2}\,\text{cp}, \qquad \rho = 1.23 \times 10^{-3}\,\text{g/cm}^3$$

Note that the drops are much too large for a significant fraction to evaporate over this distance and time.

b. If the Henry's Law coefficient is high enough that we are completely liquid-side limited (the usual case for a VOC), the concentration of the pollutant at the surface of the drop is essentially zero. Assuming that there is no circulation in the drop (diffusion only), write down the equation governing the transient concentration distribution and the total amount of pollutant remaining. Render the equations dimensionless and scale the problem.

c. At the temperature of the drop (reduced from that of the air due to "wet bulb" evaporation effects), the diffusivity of our VOC is

$$D = 1 \times 10^{-5}\,\text{cm}^2/\text{s}$$

For this diffusivity and the fall time calculated in part (a), what fraction of the pollutant would be remaining in the drop when it hits the pond? *Note:* While you can solve this from scratch (and should know how to do so), I would strongly recommend thinking about analogies with heat transfer and consulting the equations page.

d. It is decided that this really isn't enough removal of the pollutant. However, your colleague argues that while the viscosity of air is small, it isn't that small—particularly in relation to the viscosity of water. That means that there is likely some circulation inside the drop. Using your knowledge of fluid mechanics (and recalling what quantity involving viscosities must be continuous across an interface), estimate the circulation time inside the drop, compare it to the answer in part (a), and come up with a revised estimate of the fraction of pollutant remaining.

Convective Heat & Mass Transfer

Subject Tags: melting; heat and mass transfer; evaporation; humidity effects; Sherwood and Nusselt numbers

Difficulty Level: Advanced

Estimated Time: 40 minutes

This problem builds upon the midterm question on melting of an ice sphere, now incorporating mass transfer effects due to water vapor in the air stream. You will analyze how humidity affects melting rate by comparing latent heat from vapor transport with conductive heat flux, and apply this to determine melt time under different humidity conditions.

Skills Tested:

- Coupled heat and mass transfer analysis
- Vapor transport modeling
- Application of correlations
- Dimensional analysis

Source: CBE 30356 Final Exam 2023 **File:** problem1013.tex

Convective Heat & Mass Transfer

In the midterm you solved for the time necessary for a sphere of ice to melt in a stream of air. In order to avoid mass transfer effects we looked at the specific case where the air was at a dew point equal to the melting temperature. Here we explore how mass transfer (e.g., the humidity of the air stream) affects the melting process.

a. For this geometry and conditions (water vapor in air) we have the extremely convenient result that the Sherwood number and Nusselt number are essentially identical (e.g., the ratio is 1 to a good approximation at all Re because for water vapor in air Sc = 0.66 and Pr = 0.71, close enough to being equal).

If the latent heat of vaporization of water is H, and the mole fraction of water in the air stream is $x_{A\infty}$ and at the surface is x_{A0}, determine the ratio of the heat flux to the ice due to mass transfer (e.g., condensation or evaporation) to that arising from heat transfer. Other parameters this ratio involves include c (the molar density of air), D_{AB} (the water vapor diffusion coefficient), k (the thermal conductivity of air) and ΔT (the temperature difference between the air and the ice). You don't need any other parameters because we can take Sh/Nu = 1 for this problem.

b. Take the air stream to be at 30°C. It is observed that the ice melts in 100 s when the dew point of the air is at 0°C (e.g., the pure heat transfer result). Note that this time would depend on all sorts of parameters such as the latent heat of fusion, the radius, the air velocity, etc., as you showed in the midterm!

How long would it take the ice to melt if the air stream were instead completely dry (e.g., 0% relative humidity) so that some of the melted ice evaporates into the air?

Use the parameter values:

$$P_{\text{sat}}|_{0^\circ\text{C}} = 0.61 \text{ kPa}, \quad P_{\text{atm}} = 101 \text{ kPa}, \quad H = 45 \text{ kJ/mol},$$
$$c = 44.6 \text{ mol/m}^3, \quad D_{AB} = 2.19 \times 10^{-5} \text{ m}^2/\text{s}, \quad k = 0.0244 \text{ W/m} \cdot \text{K}$$

c. If the air stream is at 30°C (where $P_{\text{sat}}|_{30^\circ\text{C}} = 4.25$ kPa) but is at 80% relative humidity, your ice melts significantly faster! How long would it take to melt now?

The Thiele Problem

Subject Tags: effectiveness factor; reaction-diffusion; second-order kinetics; catalyst layer

Difficulty Level: Advanced

Estimated Time: 45 minutes (+2 points extra credit)

This problem extends the derivation of the Thiele modulus and effectiveness factor to second-order kinetics. Students will model coupled diffusion and reaction in a catalyst slab, perform dimensional analysis, and analyze the fast reaction limit.

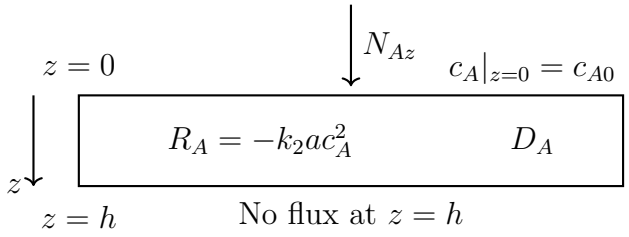
Skills Tested:

- Formulation of coupled reaction-diffusion equations
- Dimensional analysis and scaling arguments
- Catalyst effectiveness computation
- Boundary condition modeling

Source: CBE 30356 Final Exam 2023 **File:** problem1014.tex

The Thiele Problem

In class we showed how to obtain the Thiele Effectiveness Factor (the ratio of the mass flux to that which would occur with no mass transfer limitations) for a catalyst. The effectiveness factor was derived for first-order reaction kinetics (as the math is simpler!), but as you will learn next fall many reactions are not first order! In this problem we shall determine the effectiveness factor for a second-order reaction with a pseudo-homogeneous reaction rate $R_A = -k_2 a c_A^2$. The quantity a is the active catalyst surface area per volume, so the reaction rate is in moles/(volume $\cdot$ time) as usual.



a. Consider the catalyst layer depicted above. The layer is of thickness h, the concentration at the surface ($z = 0$) is c_{A0}, and the diffusivity is D_A. We have the pseudo-homogeneous reaction rate $R_A = -k_2 a c_A^2$ as discussed above. For simplicity, we shall consider the problem to be equimolar counter-diffusion (e.g., $2A \rightarrow C + D$ or similar) so that diffusion and reaction do not lead to convection.

Write down the differential equation and boundary conditions for the reactant concentration c_A with these assumptions. Write down the expression for the molar flux of A at the surface $z = 0$.

b. We are primarily interested in the fast reaction (e.g., diffusion-limited) case, where the appropriate length scale for z is much less than h. Render the equations dimensionless and via scaling determine this length scale and the scale for the molar flux at the surface.

c. The effectiveness factor is the ratio of the molar flux to that which would occur if there were no diffusional limitations (e.g., where the entire catalyst layer of thickness h is exposed to the concentration c_{A0}).

Develop an expression for the effectiveness factor in terms of the dimensionless derivative at the surface.

d. (2 points extra credit) While the differential equation is non-linear, in the limit where $h/z_c \gg 1$ it is easily solved. (Hint: use the trick of multiplying by the derivative to convert both terms into perfect differentials!).

For a two points extra credit, determine the "number" for the effectiveness factor in this limit.

A Cylindrical Cooling Fin

Subject Tags: axial conduction; external heat transfer; scaling; fins

Difficulty Level: Moderate

Estimated Time: 30 minutes

This problem considers axial conduction through a long cylindrical fin that dissipates heat by external convection. You will derive the temperature profile, perform dimensional analysis, and determine the impact of radial temperature gradients and transient behavior on heat dissipation.

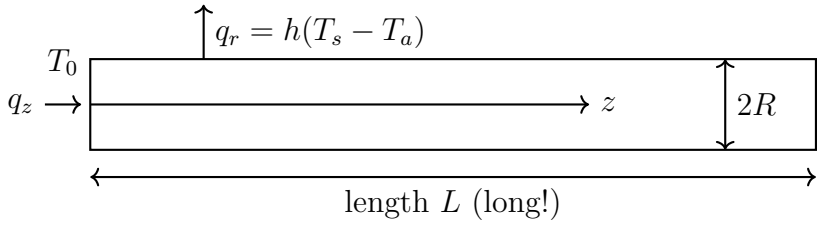
Skills Tested:

- Derivation of heat balance equations
- Dimensional analysis and scaling
- Understanding steady-state vs transient conduction

Source: CBE 30356 Final Exam 2023 **File:** problem1015.tex

A Cylindrical Cooling Fin

A cylindrical rod of length L and radius R has its base held at temperature T_0. The fin loses heat through its outer surface to ambient surroundings at temperature T_a, with heat transfer coefficient h. Assume the system reaches steady state and that radial temperature gradients can be neglected, so the temperature is averaged across each cross-section.



a. Steady-state, long cooling rods. Develop an equation for the axial variation of the cross-sectionally averaged temperature $\overline{T}(z)$. Render this equation dimensionless and use it to determine the steady-state heat flux at the base of the rod ($z = 0$) in the limit $L/z_c \gg 1$.

b. Surface vs average temperature. Your averaged expression required that the local surface temperature T_s be equal to the local average temperature $\overline{T}$, which of course isn't quite true. From scaling arguments, what would be the characteristic magnitude of this temperature difference? (Hint: consider internal and external heat transfer resistances in the radial direction.)

c. Transient behavior. It takes time for the temperature distribution and heat flux to reach steady state. How does this time scale with the parameters of the problem?

Dimensionless Groups

Subject Tags: dimensionless groups; scaling; Reynolds; Nusselt; Prandtl; Biot; Froude; Peclet

Difficulty Level: Moderate

Estimated Time: 30 minutes

Summary: This problem explores the physical basis and relevance of common dimensionless groups used in transport phenomena. Students are asked to identify the relevant physical mechanisms for each dimensionless group, construct their definitions using the given parameters, and identify a situation where each might be used in analysis.

Skills Tested:

- Dimensional analysis
- Interpretation of transport phenomena
- Application of scaling to real-world engineering problems

Source: CBE 30356 Final Exam 2023 **File:** problem1016.tex

Dimensionless Groups and Their Physical Roles

Using the parameters given below, determine the dimensionless groups corresponding to the following physical mechanisms and state a problem or equation where it would play a role.

a. Re = (Inertial Forces / Viscous Forces)

b. Nu = (Heat Transfer / Conductive Heat Transfer)

c. Sh = (Mass Transfer / Diffusive Mass Transfer)

d. Bi = (Internal Heat Transfer Resistance / External Heat Transfer Resistance)

e. St =(Heat Transfer / Convective Heat Transfer)

f. Pr = (Momentum Diffusivity / Thermal Diffusivity)

g. Sc = (Momentum Diffusivity / Mass Diffusivity)

h. Pe = (Convective Heat Transfer / Conductive Heat Transfer)

i. Fr = (Inertial Forces / Gravitational Forces)

j. Eu = (Pressure Differential / Dynamic Pressure)

Available Parameters:

- ΔP pressure differential [kg/m $\cdot$ s^2]
- ρ fluid density [kg/m^3]
- U velocity [m/s]
- d diameter [m]
- α thermal diffusivity [m^2/s]
- ν kinematic viscosity [m^2/s]
- D_{AB} mass diffusivity [m^2/s]
- g gravity [m/s^2]
- h heat transfer coefficient [W/m$^2 \cdot$ K]
- k thermal conductivity [W/m $\cdot$ K]
- k_{int} internal conductivity [W/m $\cdot$ K]
- $\hat{C}_p$ specific heat capacity [J/kg $\cdot$ K]

CBE 30356 Problem Sets

The following problem sets were used in CBE 30356. In contrast to CBE 30355 where the "what students should learn" summary was provided as a guide to students with each problem set, these had not yet been prepared for 356 by the time I had rotated off of the course. I have included them here, however, both to preserve the directory structure and as an aid to identifying problems of interest. These are generated by Chat (in the same manner as an AI will summarize a document) and have only received minor edits. The prompt used to produce these .tex files is `ai_problem_set_summary.txt` and is included in the .zip file. To improve the summary (and make it compatible with the rest of the Compendium) the prompt is followed by pasting in the contents of a 355 summary `hw004x.tex`. The script produced by Chat (which lacks the preamble by choice) is then checked and pasted into the appropriate file name. The individual problem files also have Chat generated summaries which are not part of the problem sets. The implementation of the problem sets may be seen in the LaTeX script `ahomeworktemplate.tex`. At the time of this writing narrated video solutions to all of these assignments may be found in the course directory at `https://diffusivepress.org/courseware/cbe30356/`.

What You Should Learn from Problem Set 1

Problem 1: Application of kinetic theory and models for gas-phase transport properties:

- **a.** Use of the Chapman-Enskog relation and the Eucken formula to calculate viscosity and thermal conductivity.
- **b.** Introduction to the Prandtl number and its physical meaning.
- **c.** Comparison of theoretical predictions with empirical data and the use of logarithmic scales for clarity in visualization.

Problem 2: Heat loss through building materials:

- **a.** Construction of a thermal resistance network to determine overall heat transfer coefficient.
- **b.** Application of basic energy cost estimation based on steady-state heat loss.
- **c.** Assessment of the effect of insulation and cost savings over time.

Problem 3: Thermoelectric analogy and empirical verification:

- **a.** Use of the Wiedemann-Franz Law to relate thermal and electrical conductivities.
- **b.** Implementation of a temperature-dependent correction to material properties.
- **c.** Data analysis and interpretation of trends across materials and temperature ranges using log-log plots.

Problem 4: Anisotropic conduction in layered materials:

- **a.** Identification of directional dependence of thermal properties in composites.
- **b.** Use of series and parallel analogies to compute directional conductivities.
- **c.** Quantitative estimation of anisotropy in effective thermal conductivity.

Problem 5: Tensor formulation of conductivity:

- **a.** Introduction to the full tensorial representation of k_{ij} for anisotropic media.
- **b.** Use of coordinate transformation to compute effective properties in an arbitrary direction.
- **c.** Recognition of analogy between heat conduction and hydrodynamic resistance in Stokes flow.

Source: CBE 30356 Problem Set 1 **File:** hw101x.tex

Transport Properties of Methane from 200K to 1000K

Tags: transport properties; viscosity; thermal conductivity; Prandtl number; methane

Difficulty: Moderate

Estimated Time: 90 minutes

Summary: This problem requires the student to apply kinetic theory relations to calculate temperature-dependent transport properties of methane. Using the Chapman-Enskog relation for viscosity and the Eucken formula for thermal conductivity, the student will determine these properties over a wide temperature range and compute the corresponding Prandtl number. Results will be compared graphically with literature data using a log temperature axis for clarity.

Skills Tested:

- Application of the Chapman-Enskog relation for viscosity
- Use of the Eucken formula for thermal conductivity
- Computation and interpretation of Prandtl number
- Graphical data comparison on a logarithmic temperature axis
- Use of literature and tabulated data

Source: CBE 30356 Problem Set 1 **File:** hw1011.tex

Transport Properties of Methane from 200K to 1000K

Using the Chapman-Enskog relation and the Eucken formula, calculate the viscosity, thermal conductivity, and Prandtl number for methane at atmospheric pressure from 200 K to 1000 K.

Graphically compare the results of your calculation to literature values. For clarity, plot temperature on a logarithmic scale.

Values for required constants and parameters may be found at the website `engineeringtoolbox.com` and in Tables E.1 and E.2 of BSL.

Heat Loss Through Attic Ceiling and Effect of Insulation

Tags: heat transfer; insulation; thermal resistance; energy cost

Difficulty: Moderate

Estimated Time: 60 minutes

Summary: This problem considers heat loss through an attic ceiling, first with only drywall and then with added insulation. Students calculate the overall heat transfer coefficient, determine energy cost over a winter season, and assess cost savings due to improved insulation.

Skills Tested:

- Calculation of overall heat transfer coefficient from thermal resistances
- Use of heat transfer equations to determine energy loss
- Unit conversion and cost analysis for energy usage
- Comparison of thermal performance with and without insulation

Source: CBE 30356 Problem Set 1 **File:** hw1012.tex

Heat Loss Through Attic Ceiling and Effect of Insulation

It's cold outside! In this problem we look at the effect of insulation in your attic.

a. Suppose your attic just has a thick drywall (3/4" gypsum board, thermal conductivity of $0.17\,\mathrm{W/m{\cdot}K}$) separating your living space from the attic, and there is an internal heat transfer coefficient between room air and the ceiling of $10\,\mathrm{W/m^2{\cdot}K}$ and in the attic of $20\,\mathrm{W/m^2{\cdot}K}$ (it's pretty drafty up there). Using this, calculate the overall heat transfer coefficient.

b. If the temperature inside is 72°F and that in the attic is 25°F, the area is $8\,\mathrm{m} \times 15\,\mathrm{m}$, and the cost of energy is \$0.15/kWhr, determine the total cost of this heat loss over a four month period (e.g., the typical winter heating period).

c. Now we add in 6" of fiberglass insulation (conductivity of $0.04\,\mathrm{W/m{\cdot}K}$). By what fraction is our heat loss reduced and how much does this save in heating costs over our four month winter heating period?

Testing the Wiedemann-Franz Law for Metals

Tags: thermal conductivity; electrical conductivity; Wiedemann-Franz Law; metals; temperature dependence

Difficulty: Moderate

Estimated Time: 90 minutes

Summary: This problem asks students to explore the relationship between thermal and electrical conductivity in metals using the Wiedemann-Franz Law. By comparing calculated and tabulated data for iron, copper, and aluminum over a temperature range, students will evaluate how well this theoretical relationship holds and practice extracting and manipulating physical property data.

Skills Tested:

- Application of the Wiedemann-Franz Law
- Use of temperature-corrected electrical resistivity data
- Calculation of conductivity from resistivity
- Data plotting and comparison on log-log axes
- Critical evaluation of theoretical models with real data

Source: CBE 30356 Problem Set 1 **File:** hw1013.tex

Testing the Wiedemann-Franz Law for Metals

In class we talked about the parallel between thermal and electrical conductivity in metals. This is quantified by the Wiedemann-Franz Law, which says that the thermal conductivity k and the electrical conductivity σ (in SI units of $1/\Omega{\cdot}\text{m}$) are related by:

$$\frac{k}{\sigma} = LT$$

where $L = 2.48 \times 10^{-8}\,\text{W}/\Omega{\cdot}\text{K}^2$ and T is in K.

In this problem, test this law for iron, copper, and aluminum over the temperature range 200 K to 800 K. Use the data from `engineeringtoolbox.com`, but note that you will need to use the temperature correction for the electrical conductivity (search for the page with temperature-dependent resistivity coefficients).

The resistivity of metals is roughly linear in temperature, so use a linear relationship to obtain resistivity as a function of temperature, and then invert it to get the electrical conductivity.

Plot the ratio k/σ versus temperature for all three materials on the same graph. Use a log-log scale. Include the Wiedemann-Franz Law on the same plot as a reference line. Be sure to label your graph clearly!

Anisotropic Thermal Conductivity in Laminated Materials

Tags: anisotropic conduction; laminated composite; parallel and series conduction; thermal circuits

Difficulty: Moderate

Estimated Time: 45 minutes

Summary: This problem examines anisotropic thermal conduction in a two-layer repeating laminate of epoxy and metal. Students determine effective thermal conductivities in both the direction of the laminate layers and perpendicular to them, using analogies to electrical resistance and the concept of series and parallel thermal circuits. They then compute a specific conductivity ratio using given material properties.

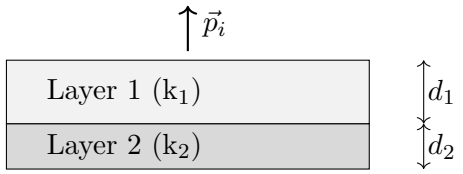
Skills Tested:

- Application of thermal resistance concepts in anisotropic media
- Series and parallel combination of thermal conductivities
- Use of geometry and material properties to compute effective properties
- Quantitative comparison of directional conductivities

Source: CBE 30356 Problem Set 1 **File:** hw1014.tex

Anisotropic Thermal Conductivity in Laminated Materials

Many materials such as wood have an anisotropic thermal conductivity. This is particularly true of laminates of dissimilar materials. Suppose we have a multilayer laminate of metal and epoxy with a unit cell depicted below (there are lots of these cells stacked up). The director $\vec{p}_i$ denotes the orientation of the planes.



a. For this material, what is the effective thermal conductivity in the $\vec{p}_i$ direction (perpendicular to the layers) and in the transverse direction (along the layers)?
Hint: Use the analogy with electrical circuits. In the $\vec{p}_i$ direction the layers act like resistors in series; in the transverse direction, like resistors in parallel.

b. If layer 1 is 0.5 mm thick epoxy ($k_1 = 0.35\,\mathrm{W/m{\cdot}K}$) and layer 2 is 1 mm thick stainless steel ($k_2 = 14.4\,\mathrm{W/m{\cdot}K}$), what is the ratio of the conductivities in the two directions?

Tensorial Conductivity in Anisotropic Laminates

Tags: anisotropic conduction; thermal conductivity tensor; coordinate transformation; analogy to fluid dynamics

Difficulty: Advanced

Estimated Time: 60 minutes

Summary: This exploratory problem challenges students to determine the full second-order tensor representation of thermal conductivity k_{ij} for a layered anisotropic material and to calculate the effective conductivity along an arbitrary direction. The case of a 45-degree orientation is considered, and the connection to low-Reynolds-number hydrodynamic resistance of a body of revolution is highlighted.

Skills Tested:

- Construction of a second-order thermal conductivity tensor for anisotropic media
- Coordinate transformation of tensors to evaluate direction-dependent conductivity
- Conceptual understanding of symmetry and principal directions
- Cross-disciplinary analogy between transport and fluid mechanics

Source: CBE 30356 Problem Set 1 **File:** hw1015.tex

Tensorial Conductivity in Anisotropic Laminates

Not for credit, but fun to do anyway:

What is the full tensorial description of k_{ij} for this material, and what is the thermal conductivity at a 45° angle?

Note that this is actually mathematically identical to the problem of a body of revolution settling in a viscous fluid (e.g., at zero Reynolds number) that we looked at last fall!

What You Should Learn from Problem Set 2

Problem 1: Hot-wire anemometry

- **a.** How electrical heating and convective cooling interact in steady state.
- **b.** Use of temperature-dependent resistance to infer heat transfer rates and fluid velocity.
- **c.** Application of empirical correlations (e.g., Whitaker) to determine convective heat transfer coefficients.
- **d.** Understanding the nonlinear coupling between voltage, resistance, and convective heat loss.

Problem 2: Annular radioactive heat source

- **a.** Application of energy balances to radially symmetric systems with internal generation.
- **b.** Use of boundary conditions including symmetry and convective cooling.
- **c.** Importance of nondimensionalization to simplify and generalize a heat transfer problem.

Problem 3: Cooling fin with heat loss on both sides

- **a.** Construction of the differential equation governing temperature variation in thin fins.
- **b.** Use of dimensionless groups to identify dominant physics and length scales.
- **c.** Solution of a canonical exponential decay problem, and interpretation of boundary behavior.

Problem 4: Exothermic reaction in a slab

- **a.** Balancing reaction heat generation and conduction to determine steady-state temperature distributions.
- **b.** Rendering nonlinear source terms dimensionless and identifying critical thresholds for stability.
- **c.** Recognition of physical instability due to thermal runaway.
- **d.** Practice solving inhomogeneous second-order ODEs using the method of particular and homogeneous solutions.

Hot Wire Anemometry and Convective Cooling

Subject Tags: heat transfer; electrical resistance; convection; correlation; plotting

Difficulty: Moderate

Estimated Time: 30 minutes

A hot wire anemometer measures airflow by observing the rate at which a heated wire is cooled by the surrounding fluid. The electrical resistance of the wire changes with temperature, providing a convenient means of temperature monitoring. In this problem, you will estimate the steady-state temperature of the wire at a fixed current and determine how the measured voltage changes with airflow speed.

Skills Tested:

- Steady-state energy balance on a cylindrical wire
- Use of empirical correlations for convective heat transfer
- Temperature dependence of electrical resistance
- Parametric plotting on semilog axes

Source: CBE 30356 Problem Set 2 **File:** hw1021.tex

Hot Wire Anemometry and Convective Cooling

A hot wire anemometer is a method for measuring air velocity by determining the rate of heat transfer and temperature from a heated wire. The idea is that because of electrical dissipation in a very thin wire it gets hot – and the temperature will depend on the air flow rate cooling it down. The temperature is measured from the resistance of the wire. Usually you measure changes in the voltage drop in what is called a Wheatstone Bridge circuit, however here we will look at a simpler problem.

Suppose your wire is $L = 1\,\text{cm}$ long and $D = 1\,\text{mm}$ in diameter. It has a resistance R_0 of 1 ohm at 20°C and a temperature coefficient α of $2.45 \times 10^{-3}\,\text{C}^{-1}$ (assume linearity). You operate the probe at a fixed current of $I = 0.5\,\text{A}$.

a. If we have a wind speed of $U = 1\,\text{m/s}$, what is the temperature of the probe? Neglect internal resistance (e.g., low Biot number as is appropriate for this problem) and use the Whittaker correlation to get the heat transfer coefficient. Ignore variation of fluid properties with temperature (just keep the temperature dependence of the electrical resistance!). Note that the power dissipation goes up a little bit with temperature too.

b. As the temperature and resistance change, so does the voltage drop V. Using a semilog scale, plot the measured voltage across the probe (the current is kept constant) as a function of wind speed over the range $1\,\text{m/s}$ to $100\,\text{m/s}$.

Heat Generation in an Annular Source

Subject Tags: cylindrical coordinates; internal generation; boundary conditions; steady state

Difficulty: Moderate

Estimated Time: 30 minutes

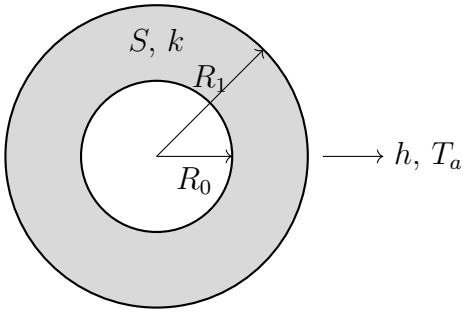
A radioactive heat source is confined to an annular region between radii R_0 and R_1, releasing heat at a constant volumetric rate S. The surrounding material has thermal conductivity k, and the outer surface at R_1 is cooled convectively to an ambient temperature T_a with a heat transfer coefficient h. The inner radius R_0 is a void, so the heat flux there must vanish. The goal is to determine the steady temperature profile.

Skills Tested:

- Steady-state energy balance in cylindrical coordinates
- Solving conduction problems with internal heat generation
- Applying symmetry and convective boundary conditions
- Use of nondimensionalization to simplify geometry

Heat Generation in an Annular Source

In this problem we will look at an annular energy source as depicted below. A radioactive source is confined to an annulus between an inner radius R_0 and an outer radius R_1, and releases heat at a rate S per unit volume. The material has a thermal conductivity k, and the outside is cooled with a heat transfer coefficient h relative to an atmospheric temperature T_a. Because there is nothing inside the radius R_0, the radial heat flux at that point is zero.



a. Using an energy balance, calculate the temperature T_1 at the outer surface R_1.

b. Now solve for the temperature T_0 at the inner edge of the radioactive source R_0. This problem is much easier if you render radius dimensionless with the outer radius!

Long Cooling Fin with Symmetric Convection

Subject Tags: heat conduction, cooling fins, dimensionless analysis, boundary value problems

Difficulty: Moderate

Estimated Time: 30 minutes

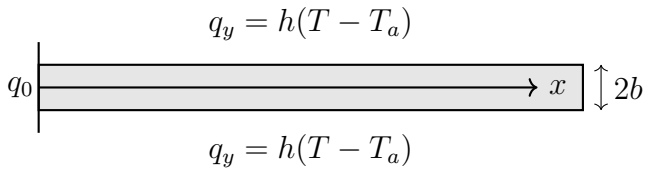
Summary: This problem involves analyzing a thin cooling fin exposed to air on both sides. Using the given heat flux at the base and thermal boundary conditions, students will derive the temperature profile, nondimensionalize the system, and solve for the base temperature assuming the fin is long compared to the thermal penetration depth.

Skills Tested:

- energy balance formulation
- applying boundary conditions
- nondimensionalizing physical systems
- solving second-order ODEs analytically

Long Cooling Fin with Symmetric Convection

A cooling fin of width 2b is cooling a reactor as depicted below. The fin is sufficiently thin that you can ignore temperature variations across the width, but of course it changes along the length due to finite thermal conductivity k. If the heat transfer coefficient between the fin and the surrounding air (temperature T_a) is h (both sides), and the heat flux at the base of the fin is q_0, we can calculate the temperature at the base.



a. Write down the governing equation and boundary conditions for the temperature $T(x)$ of the fin, averaged across its thin width.

b. Render the equation dimensionless. Identify the appropriate temperature scale and characteristic length scale.

c. Assuming the fin is long relative to this length scale, solve the problem and find an expression for the temperature at the base.

Thermal Runaway in a Reactive Slab

Subject Tags: heat conduction, exothermic reaction, boundary conditions, nondimensionalization, thermal runaway

Difficulty: Advanced

Estimated Time: 45 minutes

Summary: A reaction is occurring in a slab of width $2b$, producing heat dissipated by cooling at the walls. The reaction rate increases with temperature, creating the potential for thermal runaway. This problem explores the formulation, nondimensionalization, and solution of the resulting heat transfer problem.

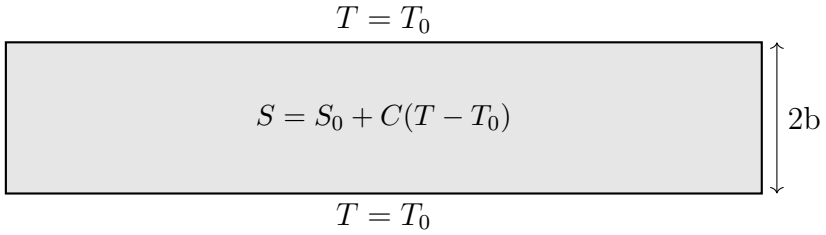
Skills Tested:

- PDE formulation from heat generation and conduction
- Application of boundary conditions
- Nondimensionalization techniques
- Analytical solution of second-order ODEs
- Critical parameter identification

Source: CBE 30356 Problem Set 2 **File:** hw1024.tex

Thermal Runaway in a Reactive Slab

A reaction is occurring in a slab of width $2b$. It's exothermic, and is producing a lot of heat which is dissipated by cooling the walls at $y = \pm b$ to a temperature T_0. The problem is that the reaction rate increases with temperature, and this creates the possibility of thermal runaway in the center! Your job is to figure out what the temperature at the centerline is under these conditions.



a. While the temperature dependence is often complex, here we shall use the simple linear relationship $S = S_0 + C(T - T_0)$. If the thermal conductivity is a constant k, write down the differential equation and boundary conditions governing the temperature at steady-state.

b. Render the equation and boundary conditions dimensionless, determine the characteristic temperature rise, and show that the problem depends only on a single dimensionless ratio.

c. Now for the fun part: Solve for the temperature distribution as a function of y^* and plot it up for a few values of the dimensionless ratio you got in part b. For sufficiently large values of C there is no solution (and your reactor just melted down). What is this critical value?

Hint: You should obtain an inhomogeneous linear second order ODE for T^*. Remember that you most easily solve this sort of equation by finding the particular solution which removes the inhomogeniety. In this case it is just a constant! The leftover bit after subtracting the particular solution is a homogeneous second order equation which should be very familiar. Apply your boundary conditions to the sum of the particular and homogenous solutions and you are done!

What You Should Learn from Problem Set 3

Problem 1: Heat transfer with time-dependent boundary conditions.

- **a.** Formulating the diffusion equation and boundary conditions for unsteady heat conduction in a slab.
- **b.** Rendering the problem dimensionless and identifying the long-time asymptotic behavior.
- **c.** Using heat flux and temperature at the insulated wall to determine the effective heat transfer coefficient at long times.

Problem 2: Transient solution of the same problem.

- **a.** Use of separation of variables and eigenfunction expansions to solve the transient diffusion equation.
- **b.** Identification of the leading eigenvalue and its role in setting the time scale for approach to steady state.

Problem 3: Heating of a sphere with uniform surface flux.

- **a.** Formulating the heat equation and boundary conditions in spherical coordinates.
- **b.** Determining the long-time solution and calculating the temperature at the surface.
- **c.** Computing the average temperature and using it to define an effective heat transfer coefficient.

Problem 4: Application of Sturm–Liouville theory to a 2D steady conduction problem.

- **a.** Recognizing the problem as elliptic and setting up the PDE accordingly.
- **b.** Using separation of variables in a steady-state 2D context.
- **c.** Solving the resulting Sturm–Liouville eigenvalue problem and constructing the full solution.
- **d.** Plotting and interpreting the temperature profile in both directions.

Source: CBE 30356 Problem Set 3 **Files:** hw103x.tex

Startup of a Heated Slab

Subject Tags: Transient conduction; boundary conditions; long-time asymptotics

Difficulty: Moderate

Estimated Time: 45 minutes

Summary: This problem explores the transient thermal response of a slab when one surface is suddenly subjected to a temperature that increases linearly with time. Students will set up and nondimensionalize the governing PDE, find the long-time asymptotic solution, and compute the heat flux and effective heat transfer coefficient across the slab.

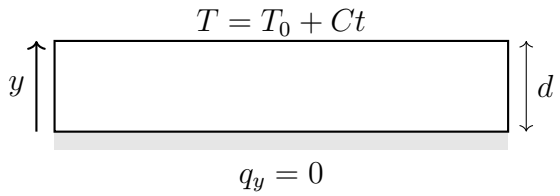
Skills Tested:

- Formulating and nondimensionalizing PDEs
- Applying appropriate boundary conditions
- Asymptotic analysis of long-time solutions
- Computing heat flux and effective heat transfer coefficients

Source: CBE 30356 Problem Set 3 **File:** hw1031.tex

Startup of a Heated Slab

Consider the slab of width d depicted below. Initially the entire slab is at a temperature T_0. The bottom at $y = 0$ is insulated ($q_y = 0$) and for $t > 0$ the upper wall at $y = d$ is maintained at a temperature that increases in time, e.g., $T = T_0 + Ct$.



a. Write down the differential equation and boundary conditions that govern this problem, and render them dimensionless.

b. Calculate the asymptotic solution at long times and determine the temperature of the lower (insulated) plate. Note that because of this boundary condition it has to be a function of temperature too!

c. Calculate the heat flux through the upper wall and using that and the answer to part (b) determine the long-time heat transfer coefficient for this problem.

Transient Solution for Heated Slab with Time-Dependent Boundary

Subject Tags: Sturm-Liouville; separation of variables; transient conduction; eigenvalue analysis

Difficulty: Advanced

Estimated Time: 60 minutes

Summary: This problem builds on the prior steady-state analysis of a slab being heated at its surface. Students will derive the full transient temperature profile using separation of variables and determine the leading eigenvalue from the Sturm-Liouville formulation. This allows them to estimate how long it takes before the asymptotic regime becomes valid.

Skills Tested:

- Application of separation of variables to PDEs with time-dependent boundary conditions
- Formulation and solution of Sturm-Liouville problems
- Identification of eigenvalues and physical interpretation in heat transfer timescales

Source: CBE 30356 Problem Set 3 **File:** hw1032.tex

Transient Solution for Heated Slab with Time-Dependent Boundary

While the asymptotic solution is the most important, the transient solution is also useful! Using separation of variables, solve the Sturm-Liouville problem for the slab and boundary conditions in problem 2 and determine the lead eigenvalue (e.g., how long do we need to wait for the heat transfer coefficient you calculated to become valid).

Heating of a Sphere with Constant Surface Flux

Subject Tags: Transient conduction; spherical symmetry; asymptotic analysis; effective heat transfer

Difficulty: Moderate

Estimated Time: 45 minutes

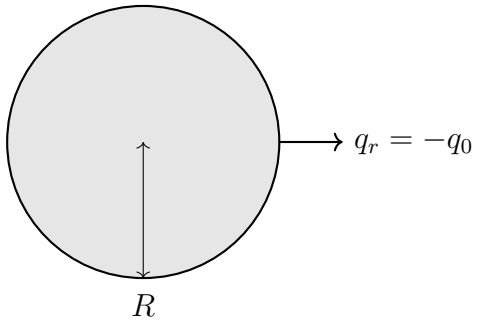
Summary: This problem examines the time-dependent heating of a solid sphere due to a uniform inward heat flux applied at its surface. Students will derive and nondimensionalize the governing PDE, compute the long-time temperature behavior, and estimate the effective heat transfer coefficient based on surface and average temperatures.

Skills Tested:

- Setting up and solving transient heat conduction in spherical coordinates
- Applying boundary conditions with specified surface heat flux
- Asymptotic analysis of long-time temperature rise
- Calculating average temperature and heat transfer coefficients

Heating of a Sphere with Constant Surface Flux

Consider the sphere of radius R depicted below. Initially it is at a temperature T_0, but we heat it uniformly at the surface with some value $q_r = -q_0$ (negative so that the temperature is going up rather than down!).



a. Write down the differential equation and boundary conditions that govern this problem and render them dimensionless.

b. Solve for the asymptotic solution at long times (which will be an increasing function of time!). Determine the temperature at the surface.

c. Calculate the average temperature of the sphere, and use that and the answer to part (b) to determine the heat transfer coefficient for this problem.

Steady 2D Heat Transfer Using Sturm-Liouville

Subject Tags: Elliptic PDE; Sturm-Liouville; steady-state conduction; separation of variables; eigenfunction expansion

Difficulty: Advanced

Estimated Time: 75 minutes

Summary: This problem introduces a steady-state two-dimensional conduction problem with boundary heating and insulated walls, which is solved using separation of variables and Sturm-Liouville theory. It provides students with experience applying eigenfunction expansions to elliptic PDEs and interpreting the resulting temperature fields.

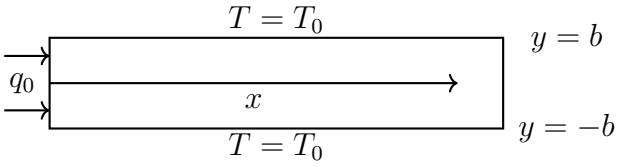
Skills Tested:

- Setup and classification of elliptic PDEs
- Use of separation of variables in steady-state 2D problems
- Formulation and solution of Sturm-Liouville eigenvalue problems
- Interpretation and plotting of 2D temperature distributions

Source: CBE 30356 Problem Set 3 **File:** hw1034.tex

Steady 2D Heat Transfer Using Sturm-Liouville

We've been using Sturm-Liouville theory and separation of variables to look at parabolic PDE's such as the heat equation, however they can be used for elliptic PDE's as well (e.g., steady heat transfer in two dimensions!). Consider the slab depicted below. The walls at $y = \pm b$ are maintained at a temperature T_0 at all time, and the end of the slab at $x = 0$ is heated with a constant heat flux $q_x = q_0$. The slab is long enough in the x-direction that the temperature in the interior just approaches that of the wall far from the heated edge (e.g., $T = T_0$ as $x \to \infty$).



a. Write down the differential equation governing the problem and render it dimensionless. (Hint: it's at steady-state, and there's only one length scale...)

b. The variable x plays the same role as t in the transient problems we've been studying (although with a second derivative rather than a first!). Using this, show that it admits a separation of variables solution and write down the solution to the x part (decaying as $x \to \infty$).

c. Set up the Sturm-Liouville problem in the y-direction and solve it, obtaining the eigenfunctions and eigenvalues.

d. Solve for the coefficients to get the complete solution.

e. Plot up the dimensionless temperature distribution in the y-direction for a few values of x, and plot the centerline temperature ($y = 0$) as a function of x.

What You Should Learn from Problem Set 4

Problem 1: Cylindrical Rod with Internal Heating — Steady State

- **a.** Practice setting up the energy balance for a radially symmetric system with internal generation and convective boundary conditions.
- **b.** Recognition that the Biot number controls the character of the temperature profile — small Bi gives nearly uniform temperature; large Bi gives boundary layer behavior.
- **c.** Insight into the similarity between the resulting parabola and classic velocity profiles from fluid mechanics (e.g., Poiseuille flow).

Problem 2: Cylindrical Rod with Internal Heating — Transient Solution

- **a.** Application of separation of variables to a radial geometry with nontrivial boundary conditions.
- **b.** Familiarity with the role of eigenfunction orthogonality and inner products in computing transient solutions.
- **c.** Recognition that analytic solutions may exist (e.g., via Bessel functions), but are often more cleanly obtained numerically.

Problem 3: Finite Difference Solution for Transient Heating of a Rod

- **a.** Implementation of a numerical scheme (finite difference with matrix solution) for solving PDEs in cylindrical coordinates.
- **b.** Understanding how to encode derivative boundary conditions into a matrix system.
- **c.** Comparison between numerical and semi-analytic solutions to verify correctness and gain confidence in numerical modeling.

Problem 4: Rectangular Slab with Internal Source — Separation of Variables Redux

- **a.** Application of steady-state energy balance in two dimensions with distributed source terms.
- **b.** Use of superposition to isolate and solve the "correction" to the large-x behavior.
- **c.** Practice in using separation of variables and eigenfunction expansion in rectangular domains.
- **d.** Connection to flow problems: understanding that this same math governs dispersion in microfluidic channels.

Heated Rod with Surface Heat Loss

Subject Tags: Cylindrical conduction, heat loss, steady-state profiles

Difficulty: Moderate

Estimated Time: 45 minutes

Summary: This problem involves the transient heating of a solid cylinder subject to internal heat generation and surface heat loss. The goal is to formulate and render the governing equations dimensionless, identify the Biot number as the key controlling parameter, and analyze the long-time steady-state profiles under various Biot number limits.

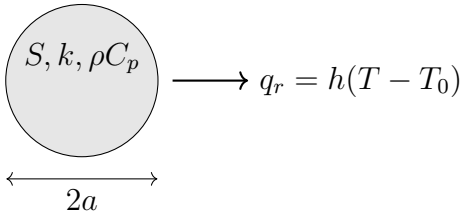
Skills Tested:

- Formulating PDEs with internal generation and convective boundary conditions
- Non-dimensionalizing with appropriate scales
- Identifying limiting behaviors using Biot number
- Visualizing steady-state profiles for different boundary conditions

Source: CBE 30356 Problem Set 4 **File:** hw1041.tex

Heated Rod with Surface Heat Loss

The first three problems deal with the heated rod depicted below. Consider a rod of radius a, thermal conductivity k, and volumetric heat capacity ρC_p. Initially the rod is at equilibrium with its surroundings at a temperature T_0. At time $t = 0$ we have a source of energy per unit volume S which warms things up. Heat is lost to the surroundings governed by a heat transfer coefficient h. So:



a. Write down the differential equation and boundary conditions governing this problem.

b. Render them dimensionless and show that the only parameter which appears in the dimensionless equations is the Biot number.

c. Solve for the asymptotic solution at long times.

d. Plot up the temperature profile for Bi = 0.1, 2, and ∞ on the same graph. Note that the Bi = ∞ parabola is the same as Poiseuille flow through a tube!

Transient Heating of a Cylindrical Rod

Subject Tags: transient conduction, numerical methods, Biot number

Difficulty: Advanced

Estimated Time: 60 minutes

Summary: This problem continues the previous one by asking you to solve the full transient problem for a heated rod with surface heat loss. Instead of solving the Bessel-based analytic solution, you will employ numerical methods, using orthogonality and integration to calculate coefficients. The final result should be a plot of the average temperature versus time for a specified Biot number.

Skills Tested:

- Transient PDE modeling and boundary conditions
- Eigenfunction expansion using orthogonality
- Numerical coefficient evaluation
- Graphical representation of average temperature vs. time

Source: CBE 30356 Problem Set 4 **File:** hw1042.tex

Transient Heating of a Cylindrical Rod

OK, now for the transient! For problem 1 above, solve the transient problem numerically and plot up the area average temperature of the rod as a function of time. Use a Biot number of 2. Note that this will require using orthogonality, integration, etc., for getting the coefficients, but the example programs will be a lot of help! This problem actually admits an analytic solution (in terms of Bessel functions) but it is pretty messy and a whole lot easier to get numerically!

Finite Difference Solution to Transient Heating

Subject Tags: finite difference, cylindrical coordinates, Biot number, numerical solution

Difficulty: Advanced

Estimated Time: 60 minutes

Summary: This problem asks you to revisit the transient heating of a rod, but now solve it numerically using a finite difference marching approach. Special care must be taken to properly implement the cylindrical geometry in the second derivative and to handle the mixed boundary condition at the surface. Use the framework and strategies described in the notes and code examples, then compare your numerical result to the previously obtained solution.

Skills Tested:

- Finite difference discretization in cylindrical coordinates
- Implementing derivative-involved boundary conditions
- Marching in time with stable schemes
- Comparison of numerical and semi-analytical solutions

Finite Difference Solution to Transient Heating

Solve problem 2 again, only this time do it using the finite difference marching solution described in the notes. There are a couple of issues here: First, because you are in the cylindrical coordinate system your (second derivative) conductive term looks a little different. The notes on how the matrix was set up in `slsolve.m` will show you what to do there. Second, your boundary condition at $r^* = 1$ now involves both the function and the derivative. Again, examining the code for `slsolve.m` will show you how to deal with that condition. Solve the problem for a Biot number of 2, calculate the average temperature of the rod, and graphically compare it to your answer for question 2.

Rectangular Slab with Volumetric Heating

Subject Tags: asymptotic steady state, separation of variables, volumetric source, rectangular geometry

Difficulty: Advanced

Estimated Time: 75 minutes

Summary: This problem involves finding the temperature distribution in a semi-infinite rectangular slab with internal heating and boundary temperatures held fixed. The task begins with formulating and nondimensionalizing the governing equation, followed by solving for the steady-state far-field temperature. You then use separation of variables to find the correction near the boundary and analyze the average profile.

Skills Tested:

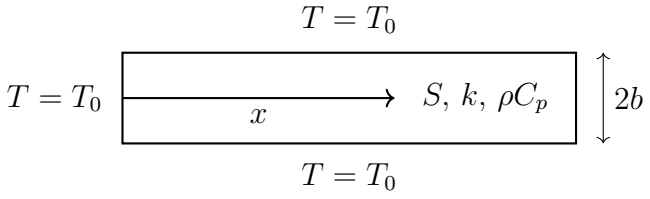
- Formulating boundary value problems with internal energy sources
- Performing nondimensionalization
- Steady-state solutions in parabolic and elliptic PDEs
- Superposition and eigenfunction expansions
- Computing spatial averages and interpreting physical implications

Source: CBE 30356 Problem Set 4 **File:** hw1044.tex

https://diffusivepress.org/courseware/cbe30356/

Rectangular Slab with Volumetric Heating

Consider the rectangular slab (semi-infinite in the x-direction) depicted below. The surface at $x = 0$ and $y = \pm b$ are maintained at a temperature T_0. There is a source of energy per unit volume S in the slab that heats things up.



a. Write down the differential equation and boundary conditions that govern this problem and render them dimensionless.

b. We are only interested in the asymptotic solution at long times here, so you can throw out the transient term! Solve for this steady-state solution when you are far from the end $x = 0$. (This solution will only be a function of y, and should be very familiar!)

c. Now for the harder part. Subtracting off the solution you obtained in part (b), solve for the temperature distribution as a function of x and y using the same sort of separation of variables solution you did in the last homework. (There will be different constants, which you will need to obtain – Wolfram Alpha works well there!) Don't forget to add the large x solution back in.

d. Plot up the depth (y-direction) average temperature as a function of x.

Note: While this problem in heat transfer isn't all that important, the mathematically identical problem for unidirectional pressure driven flow in a rectangular duct very much is! It turns out that the slowdown of fluid velocity near the side walls (corresponding to the lower depth-averaged temperature you are plotting in this problem) is *the* dominant source of dispersion in a typical microfluidic "Lab on a Chip" system and has large implications for chip design.

What You Should Learn from Problem Set 5

Problem 1: Vertical Unidirectional Flow (Transient)

- **a.** Recognition that unidirectional flow and transient heat conduction problems are governed by the same mathematics.
- **b.** Ability to write governing equations for transient problems and render them dimensionless.
- **c.** Understanding of asymptotic behavior and how to extract wall shear stress.
- **d.** Introduction to eigenvalue problems in cylindrical coordinates and the interpretation of decay timescales.

Problem 2: Transient Boundary Layers and Self-Similarity

- **a.** Setting up and rendering a semi-infinite transient conduction problem dimensionless.
- **b.** Applying affine stretching to reveal self-similar structure in PDEs.
- **c.** Identification of the proper similarity variable and canonical form.
- **d.** Use of scaling arguments to estimate heat flux magnitudes.

Problem 3: Shooting Method and Numerical Construction

- **a.** Practice in manipulating transformed PDEs.
- **b.** Use of numerical shooting methods to determine unknown constants in similarity solutions.
- **c.** Plotting of scaled solutions to reveal structure and decay behavior.

Problem 4: Periodic Forcing in Semi-Infinite Media

- **a.** Understanding of how sinusoidal forcing at the boundary propagates into a semi-infinite domain.
- **b.** Derivation of the amplitude of the oscillatory response as a function of material properties and frequency.
- **c.** Physical interpretation of penetration depth and its dependence on frequency.
- **d.** Appreciation for the mathematical duality between heat conduction and fluid mechanics problems with oscillatory boundary conditions.

Transient Start-Up of Flow in a Tube

Subject Tags: momentum transport, unidirectional flow, analogy to heat conduction, Bessel functions, wall shear stress

Difficulty: Moderate

Estimated Time: 90 minutes

Summary: This problem explores the transient start-up of unidirectional flow in a vertical cylindrical tube by leveraging the mathematical analogy with heat conduction in solids. The goal is to write down the governing equations, determine the steady-state solution, extract the lead eigenvalue for the transient response, and visualize the wall shear stress evolution with time. It also provides an opportunity to reinforce the structural parallels between different transport phenomena.

Skills Tested:

- PDE formulation and boundary condition setup for momentum transport
- Nondimensionalization and identification of governing parameters
- Long-time asymptotic analysis
- Eigenvalue problem setup and numerical evaluation
- Wall shear stress calculation and graphical representation

Source: CBE 30356 Problem Set 5 **File:** hw1051.tex

Transient Start-Up of Flow in a Tube

As has been repeatedly emphasized in class, unidirectional flow of an incompressible fluid is the same as heat transport in solids: both the equations and the solutions are the same! In this problem consider a long vertical tube of radius a filled with a viscous liquid of density ρ and viscosity μ, initially at rest. At time $t = 0$ you "let it go" (e.g., think of a filled straw when you take your finger off the end!) and it starts to move. Solve the following:

a. Write down the governing equations and render them dimensionless.

b. Determine the asymptotic solution at long times, and calculate the shear stress at the wall $r = a$.

c. Set up the problem for the decaying solution and get the lead eigenvalue, which determines how fast the asymptotic solution is reached. Note: do this numerically unless you particularly like to work with Bessel functions...

d. Plot up the wall shear stress as a function of time, including the long time asymptote.

Hint: Think about the relationship between this problem and the one you solved for homework last week...

Early-Time Boundary Layer with Internal Heating

Subject Tags: self-similar diffusion, boundary layer, internal heating, early-time asymptotics

Difficulty: Moderate

Estimated Time: 75 minutes

Summary: This problem focuses on the early-time transient response of a semi-infinite medium with internal heating, analyzed through the lens of a boundary layer formulation. Students are asked to construct the governing equations, identify the appropriate similarity transformation using affine stretching, and determine the wall heat flux behavior in the self-similar limit.

Skills Tested:

- Formulation of governing equations and thermal boundary conditions
- Non-dimensionalization of parabolic PDEs
- Use of affine stretching and identification of self-similar variables
- Extraction of wall flux scaling from similarity form

Source: CBE 30356 Problem Set 5 **File:** hw1052.tex

https://diffusivepress.org/courseware/cbe30356/

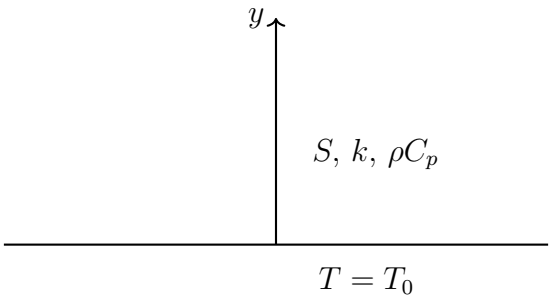
Early-Time Boundary Layer with Internal Heating

For very short times it is actually more appropriate to look at this sort of problem as a transient boundary layer. Consider the semi-infinite space depicted below. The initial temperature is T_0, and the wall is maintained at a temperature T_0 at all times. There's a heat source S in the material, however, and that means that it is getting hotter in time (but not at $y = 0$). Far from the lower wall the temperature gradient should vanish (even though it is getting warmer in time).

a. Write down the equations and boundary conditions governing this problem and render them dimensionless.

b. Using affine stretching, show that the problem admits a self-similar solution, and obtain the similarity rule and variable in canonical form.

c. Determine the heat flux at the wall to within an unknown $\mathcal{O}(1)$ constant.



Numerical Solution of Self-Similar Heating Problem

Subject Tags: similarity solution, boundary layer, shooting method, numerical methods

Difficulty: Moderate

Estimated Time: 60 minutes

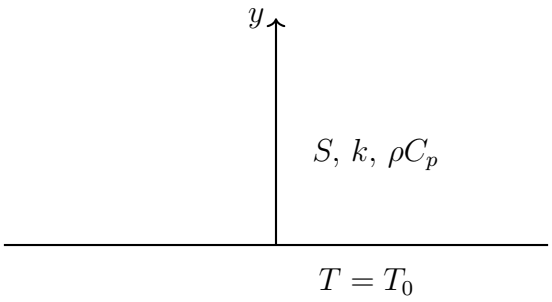
Summary: This follow-up problem asks students to numerically solve the self-similar form of the early-time boundary layer heating problem. Using the shooting method, students will determine the scaling constant and plot the resulting dimensionless temperature profile.

Skills Tested:

- Transformation of PDEs to ODEs using similarity variables
- Implementation of the shooting method for boundary value problems
- Generation of scaled temperature profile plots

Numerical Solution of Self-Similar Heating Problem

For problem 2 above, get the transformed differential equation and, using the shooting method, determine the constant. Plot the dimensionless scaled temperature profile (e.g., $f(\eta)$).



Periodic Heat Flux at the Wall of a Semi-Infinite Medium

Subject Tags: transient heat conduction, semi-infinite medium, periodic heat flux, frequency response

Difficulty: Moderate

Estimated Time: 60 minutes

Summary: This problem revisits the semi-infinite domain model from lecture, but now with a periodic heat flux rather than temperature imposed at the boundary. The task is to find the amplitude of the resulting temperature oscillation at the wall in terms of the frequency and material properties.

Skills Tested:

- Applying boundary conditions involving sinusoidal heat flux
- Solving linear PDEs with time-periodic forcing
- Understanding phase lag and penetration depth in unsteady heat conduction

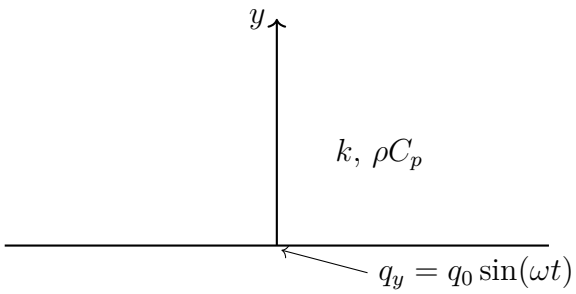
Source: CBE 30356 Problem Set 5 **File:** hw1054.tex

Periodic Heat Flux into a Semi-Infinite Medium

Consider the problem discussed in class where there was a periodic heating of a semi-infinite medium. Suppose instead of imposing a periodic temperature at the bottom, we impose a periodic heat flux of the form:

$$q_y = q_0 \sin(\omega t)$$

Solve for the amplitude of the resulting temperature at the wall as a function of the frequency ω and the properties of the material.



What You Should Learn from Problem Set 6

Problem 1: Thermal entrance region with constant wall temperature:

- **a.** Deriving governing equations and boundary conditions in the thermal boundary layer limit (flat-earth approximation).
- **b.** Use of similarity transformations to reduce PDEs to ODEs in canonical form.
- **c.** Extraction of local Nusselt number from transformed equations and interpretation in the context of boundary layer development.
- **d.** Comparison of approximate and exact solutions for physical insight.

Problem 2: Asymptotic Nu for flow between parallel plates with constant heat flux:

- **a.** Recognizing when you don't need separation of variables — exact answers can sometimes come from integral balances.
- **b.** Integration of energy equations with known velocity and boundary conditions to extract global results like Nu.
- **c.** Reinforcement of the idea that Nu depends on both geometry and boundary conditions.

Problem 3: Flat velocity profile in tube with constant wall heat flux:

- **a.** Making analogies to transient conduction problems to simplify complex convective problems.
- **b.** Use of similarity to collapse convection problems into more manageable forms.
- **c.** Interpreting flat velocity profiles as limits where analytical solutions become easier.

Problem 4: Channel with flat velocity and constant wall temperature:

- **a.** Applying separation of variables in the one case where it actually works well.
- **b.** Use of lead eigenfunction to extract asymptotic behavior without needing the full solution.
- **c.** Comparison of eigenvalue-based solutions with tabulated values (e.g., BS&L Table 14.2-1).

Thermal Entrance Region with Constant Wall Temperature

Subject Tags: Heat Transfer; Boundary Layers; Self-Similarity; Nusselt Number

Difficulty: Advanced

Estimated Time: 2–3 hours

Summary: This problem revisits the thermal entrance region in internal flow, but with a constant wall temperature instead of constant heat flux. Students will derive the governing equations under the thin boundary layer assumption, identify the self-similar form of the solution, estimate the local Nusselt number up to a constant, and compare their results numerically or analytically with the classical separation-of-variables solution. This highlights both asymptotic and numerical solution strategies for boundary-layer transport problems.

Skills Tested:

- Deriving and nondimensionalizing governing equations in thermal boundary layers
- Applying the self-similar transformation to reduce PDEs to ODEs
- Solving transformed equations using the shooting method
- Interpreting and comparing different approaches to estimating the Nusselt number

Source: CBE 30356 Problem Set 6 **File:** hw1061.tex

Thermal Entrance Region with Constant Wall Temperature

In lecture we solved for the thermal entrance region Nusselt number for flow through a pipe where we had a constant heat flux at the wall. Using the same approach (e.g., self-similar profile for a thin boundary layer near the wall), solve for the local Nusselt number for the constant wall temperature case in the thermal boundary layer development region.

a. Using the "flat earth limit" case, derive the equations and boundary conditions governing the temperature profile.

b. Show that it admits a self-similar solution in this limit, and using this determine the local Nusselt number to within an unknown constant (e.g., the solution to the transformed self-similar DE).

c. Solve the DE using the shooting method (or do it analytically if you wish!), and compare your result graphically to the separation of variables solution provided in the problem of the day notes.

Asymptotic Nusselt Number in a Channel with Constant Heat Flux

Subject Tags: Heat Transfer; Internal Flow; Nusselt Number; Laminar Flow

Difficulty: Moderate

Estimated Time: 1–1.5 hours

Summary: In this problem, you will determine the asymptotic Nusselt number for laminar flow between two plates with a constant wall heat flux. Using the given velocity profile, you will apply energy conservation to obtain an exact expression for the Nusselt number without solving the full temperature field. This illustrates how powerful results can be obtained from symmetry and scaling arguments.

Skills Tested:

- Integration of known flow profiles
- Understanding the impact of boundary conditions on heat transfer
- Application of definitions of the Nusselt number in internal flow
- Exploiting symmetry and fully-developed assumptions to simplify analysis

Source: CBE 30356 Problem Set 6 **File:** hw1062.tex

https://diffusivepress.org/courseware/cbe30356/

Asymptotic Nusselt Number in a Channel with Constant Heat Flux

The local Nusselt number depends on both the boundary conditions and the geometry. For the case of constant wall heat flux, solve for the asymptotic Nusselt number for laminar flow in a channel of width $2b$, where the velocity profile is

$$u_x = \frac{3}{2}U\left(1 - \frac{y^2}{b^2}\right).$$

Remember that this doesn't require a separation of variables solution to get an exact answer!

Asymptotic Nusselt Number for Flat Velocity Profile with Constant Heat Flux

Subject Tags: Heat Transfer; Analogy; Nusselt Number; Slip Flow; Energy Balance

Difficulty: Moderate

Estimated Time: 1 hour

Summary: This problem explores the analogy between unsteady heat conduction and unidirectional flow with flat velocity profiles. You'll determine the asymptotic Nusselt number for fully developed internal convection when the velocity profile is uniform across the cross-section and the wall heat flux is constant. Drawing analogies and connecting seemingly unrelated problems is a powerful technique in transport analysis.

Skills Tested:

- Recognition and application of analogies between conduction and convection
- Working with idealized boundary conditions (slip walls)
- Asymptotic analysis in convective systems
- Use of previously solved problems as templates for new scenarios

Source: CBE 30356 Problem Set 6 **File:** hw1063.tex

https://diffusivepress.org/courseware/cbe30356/

Asymptotic Nu for Flat Velocity Profile

As always, there are strong analogies between unsteady conduction in solids and unidirectional convective heat transfer. This is particularly true if we have a flat velocity profile such as you would get if you had perfect wall slip (e.g., $u_z = U$ rather than our usual parabola). Using this analogy, determine the asymptotic Nusselt number for a flat velocity profile for flow in a tube with constant wall heat flux.

Hint: Once you've made the analogy, it becomes pretty similar to a problem you've done before!

Asymptotic Nu for Channel Flow with Flat Velocity and Constant Wall Temperature

Subject Tags: Heat Transfer; Separation of Variables; Nusselt Number; Eigenfunction; Internal Flow

Difficulty: Moderate

Estimated Time: 1 hour

Summary: In this problem, you will analyze fully developed thermal convection in a planar channel with constant wall temperature and a flat velocity profile. Because the velocity profile is uniform, the governing equation simplifies and allows a straightforward separation of variables solution. Using the lead eigenfunction, you will extract the asymptotic Nusselt number, which quantifies the heat transfer enhancement.

Skills Tested:

- Analytical solution of heat transfer problems via separation of variables
- Extraction of Nusselt number from eigenfunction solutions
- Understanding the influence of velocity profiles on convective heat transfer
- Application of textbook reference results (e.g., BS&L Table 14.2-1)

Source: CBE 30356 Problem Set 6 **File:** hw1064.tex

Asymptotic Nusselt Number from Lead Eigenfunction

Consider again flow through a channel of width $2b$, this time with a flat velocity profile $u_x = U$ rather than the parabolic velocity distribution corresponding to Poiseuille flow. This makes it possible to solve the separation of variables problem analytically!

For the case of a constant wall temperature (e.g., what was discussed in the problem of the day), determine the asymptotic value of the Nusselt number from the lead eigenfunction. Note that you do not need to calculate the coefficient of the eigenfunction, as it should cancel out — we are only using the form of the leading term in the asymptotic limit.

Note: The results for all four of these problems are given in Table 14.2-1 of BS&L.

What You Should Learn from Problem Set 7

Problem 1: Transpirational cooling.

- **a.** Use of energy balances over spherical shells to determine conductive and convective fluxes.
- **b.** Insight into how flow through porous insulation can reduce overall heat leak.
- **c.** Appreciation for how boundary conditions drive internal evaporation rates in storage systems.

Problem 2: Heat exchanger performance and limitations.

- **a.** Understanding the concept of counter-current versus co-current exchange and how they affect minimum flow rates.
- **b.** Application of energy balances to determine required exchanger length given pipe geometry and flow rate.
- **c.** Real-world connection between heat exchanger design and practical operational constraints.

Problem 3: Natural convection from a heated vertical surface.

- **a.** Development of characteristic temperature rise using scaling arguments.
- **b.** Use of dimensional analysis to relate flux boundary conditions to flow field development.
- **c.** Reinforcement of how boundary condition changes alter asymptotic scaling results.

Problem 4: Thermal losses from glazing systems.

- **a.** Construction of thermal resistance networks incorporating convection and radiation.
- **b.** Comparison of multiple-layer window systems including radiative properties.
- **c.** Quantification of economic impact from improved thermal design.
- **d.** Recognition of the optimal placement of low-emissivity coatings and why that location matters.

Transpirational Cooling in a Spherical Tank

Subject Tags: heat transfer, conduction, convection, spherical coordinates, mass balance

Difficulty: Moderate

Estimated Time: 45 minutes

Summary: This problem introduces the idea of transpirational cooling as a mechanism for reducing heat loss through porous insulation. By performing a shell energy balance in spherical coordinates, you'll derive expressions for heat flux with and without convective mass flow and determine the corresponding rate of evaporation of the liquefied gas. The problem emphasizes analytical modeling and balance techniques.

Skills Tested:

- Constructing and interpreting energy shell balances in spherical coordinates
- Analyzing convective and conductive heat transfer mechanisms
- Applying thermal boundary conditions to derive analytical solutions
- Connecting heat flux to evaporation rate via latent heat

Transpirational Cooling in a Spherical Tank

A classic problem in convective heat transfer is transpirational cooling. The idea is that if you are trying to keep a tank of liquefied gas cold by insulating it, you can reduce the losses by forcing the boiling gas out through the insulation rather than just letting it escape.

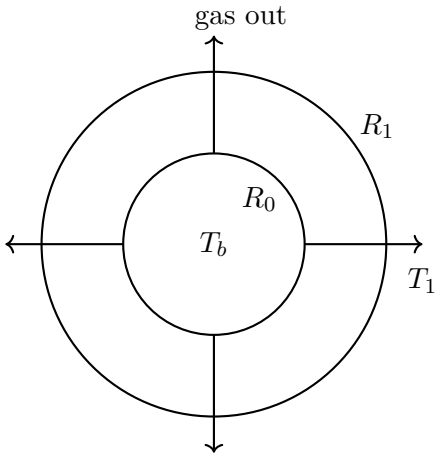
Suppose we have a spherical tank of radius R_0 surrounded by a spherical shell of porous insulation of outer radius R_1. The insulation has conductivity k, and the heat capacity per unit mass of the boiling gas flowing outwards through the insulation is C_p. If the latent heat of evaporation of the boiling liquid gas is L (units: energy/mass), the inner temperature is the boiling point T_b, and the temperature outside the insulating layer is T_1.

a. Solve for the rate of evaporation *with* transpiration (e.g., forced outward gas flow).

b. Solve for the rate of evaporation *without* transpiration (e.g., when gases escape freely through the outer shell).

c. Compare both results and comment on the effectiveness of transpirational cooling as a strategy for reducing losses.

Hint: It's actually easier to set up the temperature equation by doing a shell balance approach over a spherical surface, where you have an energy flux in via conduction through the insulation and a convective flux due to the gas (in thermal equilibrium with the local insulation temperature). These spherical shell balances are described in BS&L.



Heat Exchanger Design and Sizing

Subject Tags: heat exchangers, counter-current, co-current, thermal design, energy balance

Difficulty: Moderate

Estimated Time: 60 minutes

Summary: This problem walks through the principles of sizing a heat exchanger to achieve desired cooling. You'll compute minimum cooling water flow rates under ideal assumptions for both counter-current and co-current configurations, and then use material and geometric information to size a physical exchanger using appropriate heat transfer equations.

Skills Tested:

- Performing energy balances on multi-stream heat exchangers
- Understanding the impact of exchanger configuration on performance
- Using physical parameters to compute required surface area and length
- Applying the concept of logarithmic mean temperature difference (LMTD)

Heat Exchanger Design and Sizing

A process produces water at a rate of 2 liters/s at a temperature of 85°C. It is necessary to cool this down to 30°C for reuse, so it's heat exchanger time! You have a source of cooling water at 25°C with no restrictions on flow rates.

a. What is the minimum cooling water flow rate (in an ideal world) necessary to achieve the desired level of cooling? Assume counter-current heat exchange.

b. If all you have is a co-current heat exchanger, what would be the minimum possible flow rate?

c. OK, now we size our heat exchanger! Suppose we have a counter-current heat exchanger consisting of:

- an interior pipe of inner diameter 5 cm,
- a wall thickness of 0.32 cm steel (use $k = 36\,\mathrm{W/m \cdot K}$),
- and an inner diameter of the outer pipe of 10 cm (annular gap is thus 2.18 cm).

Using this, determine how long the heat exchanger would have to be to achieve the desired heat transfer if the flow rate of the cooling water is 50% greater than the minimum calculated in part **a**.

Natural Convection Scaling from a Heated Plane

Subject Tags: natural convection, scaling, heat transfer, vertical surface, boundary layers

Difficulty: Moderate

Estimated Time: 45 minutes

Summary: This problem explores how to estimate the surface temperature along a vertical plane subjected to a uniform heat flux. Unlike the case of a fixed wall temperature, the unknown profile arises from the boundary condition change. You'll revisit the natural convection framework, re-derive the relevant scaling laws, and deduce the functional dependence of wall temperature on height.

Skills Tested:

- Performing scaling analysis for natural convection flows
- Understanding the impact of boundary conditions on resulting temperature fields
- Applying dimensional reasoning to thermal boundary layer development
- Connecting heat flux inputs to emergent temperature profiles

Natural Convection Scaling from a Heated Plane

In lecture we used scaling to determine the heat transfer from a vertical plane due to natural convection. In this problem we do the inverse:

Suppose a plane of height H has a constant heat flux through the surface (say, we have some dissipation or reaction occurring that is releasing heat at a uniform rate per area). By scaling the equations in the same manner as was done in the notes (but now with a different boundary condition so things will change!) determine the characteristic temperature of the plane as a function of height to within some unknown $\mathcal{O}(1)$ dimensionless constant.

Heat Loss through Residential Window Glazing

Subject Tags: heat transfer, glazing, thermal resistance, energy cost, circuit model

Difficulty: Moderate

Estimated Time: 60 minutes

Summary: This problem examines various window glazing configurations to evaluate heat loss from a home and the associated energy costs. You'll calculate the thermal resistance for each setup, model the window as a series of heat transfer elements, and estimate monthly energy expenditures under realistic conditions. The final part also touches on the strategic placement of low-emissivity coatings to optimize performance.

Skills Tested:

- Constructing and analyzing thermal resistance networks
- Modeling heat loss through multi-layered systems
- Applying convective and radiative heat transfer principles
- Calculating energy costs from thermal flux

Heat Loss through Residential Window Glazing

A house (mine) is maintained at 68°F while the temperature outside is a chilly 24°F (average temperature in January). In this problem we shall solve for the heat loss for a variety of glazing options. Parameters are as follows:

- Total window area: 40 m^2
- Interior film heat transfer coefficient: 5 $W/m^{2\circ}K$
- Exterior film heat transfer coefficient: 20 $W/m^{2\circ}K$
- Gap heat transfer coefficient for double or triple glazing (per gap): 2 $W/m^{2\circ}K$
- Transmitivity for solar radiation: 1
- Emissivity of normal glass (for thermal radiation): 1
- Reflectivity of low emissivity glass (for thermal radiation): 0.75

For each of the following options calculate the energy loss and, assuming \$0.15/kWhr, the dollar value of the lost energy per month. You may find it helpful to construct a circuit diagram for each.

a. Single glazing (you don't use this in Indiana).

b. Double glazing.

c. Triple glazing (this would not fit in our existing frames).

d. Double glazing with a reflective coating on the inside of the outer pane of glass. Why is this the best place to place a single coat of Fe_2O_3? (This is what we did...)

What You Should Learn from Problem Set 8

Problem 1: Heat transfer in a large-scale process system.

- **a.** Practice computing Reynolds, Prandtl, and Nusselt numbers from raw process data.
- **b.** Application of heat transfer correlations (Dittus-Boelter) to determine film coefficients.
- **c.** Understanding of thermal resistance in composite systems and how to identify rate-limiting steps.
- **d.** Comparison of heat fluxes across film, wall, and contents—what's actually controlling?

Problem 2: Mass transfer from a spherical drop in microgravity.

- **a.** Use of spherical diffusion geometry to derive a molar flux expression.
- **b.** Incorporation of finite mole fraction correction in dilute gas-phase transport.
- **c.** Conversion from molar to mass flux and calculation of shrinkage rate for a small drop.

Problem 3: Coupling between mass and energy transfer.

- **a.** Application of the energy balance at the evaporating interface to determine surface temperature.
- **b.** Realization that significant evaporative cooling occurs even in a low-flux situation.
- **c.** Coupling between surface temperature and vapor pressure; iteration or root-finding required to solve.

Problem 4: Transient solution for total evaporation time.

- **a.** Analytic integration of a differential equation describing drop radius vs time.
- **b.** Understanding how temperature effects can significantly change system behavior.
- **c.** Final synthesis of coupled mass and energy transfer in a transient context.

HB Fuller Heat Transfer Analysis

Subject Tags: Convective Heat Transfer, Dittus-Boelter, Prandtl Number, Process Design, Heat Exchanger Performance

Difficulty: Advanced

Estimated Time: 90 minutes

Summary: This multi-part problem walks through a detailed heat transfer analysis of a process heating system used by HB Fuller to melt glue precursors in a large vessel. Students will work through hydraulic diameter, Reynolds and Prandtl numbers, and multiple heat transfer coefficients to understand which part of the system is limiting heat transfer into the vessel.

Skills Tested:

- Reynolds number and velocity calculations for internal flow
- Use of Dittus-Boelter and Nusselt number correlations
- Heat transfer area analysis and wall temperature estimation
- Understanding of conductive, convective, and composite resistances
- Diagnosing rate-limiting step in a real-world thermal system

HB Fuller Heat Transfer Analysis

Last year the CBE department worked on a project with HB Fuller to investigate ways of accelerating their glue fabrication process. One proposed candidate for the time limiting step was the rate of heating/mixing/melting in a very large vessel. This 6 ft diameter vessel is filled with plastic glue precursor which must be melted and blended. Heating is primarily accomplished by a hot oil heating system wrapping the vessel with about 300 ft of 3 inch ID half-round pipe welded to the side. So:

a. Calculate the hydraulic diameter of the half-round pipe.

b. The oil has a density of 0.831 g/cm^3 and kinematic viscosity of 0.53 cS at the operating inlet temperature (417°F). The flow rate is reported to be 114 GPM. Based on this, calculate the fluid velocity, the residence time in the pipe, and the Reynolds number.

c. The heat transfer fluid is Therminol 59, which at these temperatures has a heat capacity of $C_p = 2.3\,\mathrm{kJ/kg^\circ K}$ and conductivity $k = 0.103\,\mathrm{W/m^\circ K}$. Using this, calculate the Prandtl number.

d. Using the Dittus-Boelter correlation, calculate the Nusselt number and the heat transfer coefficient.

e. The oil is reported to have an exit temperature of 405°F. Based on this, what is the energy transfer from the fluid to the vessel? The heater for the oil has a rating of 200 kW. How does the energy loss of the oil compare to the maximum heating rate?

f. Based on the energy exchange with the vessel and the pipe area (the 3 inch x 300 ft part welded to the vessel), calculate the temperature of the vessel wall in contact with the oil (e.g., the $T_w - T_b$ temperature difference).

g. The vessel is 3/8" thick SS 304 steel. Calculate the heat transfer coefficient for the steel wall. How does the heat transfer coefficient of the steel compare to that of the oil pipe?

h. The contents of the vessel being heated are at approximately 300°F during the time the oil temperatures were being measured. Based on this, estimate the internal heat transfer coefficient.

i. We can model the heat transfer inside the vessel as heat transfer through a stagnant film (an effective film thickness). If the conductivity of the goo on the inside is $0.2\,\mathrm{W/m^\circ K}$, what would be the thickness of this layer?

j. Based on all this, what's controlling the heat transfer in this system?

Evaporation of a Floating Drop in Microgravity

Subject Tags: Mass Transfer, Diffusion, Evaporation, Microgravity, ISS, Spherical Coordinates

Difficulty: Moderate

Estimated Time: 60 minutes

Summary: This problem considers the slow evaporation of a water droplet floating on the ISS, where the absence of gravity eliminates convective effects. You'll derive and evaluate the molar diffusive flux from a spherical interface in a stagnant gas, with attention to saturation and finite mole fraction corrections.

Skills Tested:

- Derivation of molar diffusive flux from a spherical interface
- Application of saturation vapor pressure and relative humidity
- Use of mole fraction gradients and finite difference corrections
- Conversion between molar flux and rate of mass loss
- Evaluation of radius change rate for evaporating drops

Source: CBE 30356 Problem Set 8 **File:** hw1082.tex

Evaporation of a Floating Drop in Microgravity

Consider a drop of water floating in zero g on the ISS (e.g., no convection!). We are interested in determining how long a drop would float around before evaporating. For your calculations we shall assume that the air is maintained at 27°C at 1 atm pressure and a relative humidity of 50% (e.g., the fraction of saturation at that temperature). (Note that both these values are a little high, but are still in the nominal range according to the ISS website.) At this temperature the water vapor pressure is only 0.0352 atm, so it is small – but you should still use the finite mole fraction correction.

a. Develop the expression for the molar flux from the surface of a drop of radius R for an infinite medium (e.g., $R_2 \gg R$, a bit simpler than was done in class).

b. If we assume that the temperature is uniform everywhere, and the diffusivity of water vapor in air is $0.242\,\mathrm{cm}^2/\mathrm{s}$ at this temperature, calculate the rate of change in radius for a 1 mm diameter drop. Don't forget to convert from moles to mass!

Evaporation with Thermal Feedback in Microgravity

Subject Tags: Evaporation, Heat Transfer, Energy Balance, Feedback Effects, Diffusion, Non-isothermal Systems

Difficulty: Advanced

Estimated Time: 75 minutes

Summary: This continuation of the previous evaporation problem introduces a coupling between the rate of mass loss and surface cooling. You'll derive an energy balance at the drop surface to relate temperature drop to evaporation, and then use this result along with empirical correlations to compute how evaporation is suppressed at lower temperatures.

Skills Tested:

- Coupling of mass transfer and heat conduction at an interface
- Application of the latent heat to derive surface cooling
- Use of vapor pressure expressions (Buck Equation) to determine concentration
- Numerical/iterative analysis of self-consistent evaporation rate

Source: CBE 30356 Problem Set 8 **File:** hw1083.tex

Evaporation with Thermal Feedback in Microgravity

Continuing with problem 2, we recognize that the evaporation leads to cooling at the drop surface. This, in turn, decreases the equilibrium concentration at the surface as well, which suppresses the evaporation rate (there's a feedback mechanism). So:

a. If the latent heat of vaporization is L and the thermal conductivity of air is k, determine how the temperature at the surface of the drop depends on the molar flux expression. You can use the pure conduction result we derived earlier in the semester as the convection from evaporation is very small.

b. The heat of vaporization of water is $24.4 \times 10^5\,\mathrm{J/kg}$ at room temperature (it's higher the further away you get from the boiling point) and the thermal conductivity of air is $0.026\,\mathrm{W/m^\circ K}$. Based on the molar flux you calculated in question 2, what is the temperature at the surface of the drop (hint: it's way off...)?

c. Now we include the effect of temperature on x_{A1}, the equilibrium concentration at the surface of the drop. The vapor pressure of water (in kPa) is well approximated by the Buck Equation (e.g., `https://en.wikipedia.org/wiki/Vapour_pressure_of_water`). Using this, determine the instantaneous rate of evaporation of our 1 mm diameter drop. Note that this will need to be done iteratively (or via root finding) as the flux goes down as the temperature goes down, leading to a smaller temperature change.

Evaporation Time of a Microgravity Water Drop

Subject Tags: Mass Transfer, Diffusion, Heat Transfer, Iterative Modeling, Evaporation Time

Difficulty: Advanced

Estimated Time: 60 minutes

Summary: This final installment in the microgravity evaporation series asks you to use your previous results to compute the total time it would take for a water drop to completely evaporate in still air aboard the ISS. You will compare the simple isothermal prediction to the more realistic result that accounts for surface cooling.

Skills Tested:

- Analytical integration of radius decay equations
- Application of derived molar flux expressions
- Quantitative comparison of isothermal vs non-isothermal results

Source: CBE 30356 Problem Set 8 **File:** hw1084.tex

Evaporation Time of a Microgravity Water Drop

As time goes on, the drop radius gets smaller and smaller. Eventually the drop evaporates entirely! We want to determine the evaporation time.

a. Using the results of problem 2 (e.g., ignoring temperature effects), determine the evaporation time of our 1 mm diameter drop. This can be done analytically, as you should have a simple expression for the rate of change in R as a function of R.

b. Now take the results of problem 3 and include the effect of temperature on evaporation rate. How does your value compare to that in part a?

Problem Set 9: A Week to Catch Up on Sleep

After the really challenging problem set 8, this week's homework was cancelled...

What You Should Learn from Problem Set 10

Problem 1: This problem synthesizes mass transfer, thermodynamics, and fluid mechanics to analyze aerosol evaporation in air.

a. Derivation and use of an analytical solution for drop evaporation with no convection, linking sedimentation and evaporation in a closed-form expression.

b. Development of a coupled numerical solution incorporating drag (via Reynolds number), and the Ranz-Marshall correlations for simultaneous mass and heat transfer.

c. Understanding of the feedback between drop size, falling velocity, surface temperature, and evaporation rate.

d. Application of the Buck equation to relate vapor pressure to surface temperature and evaluate humidity-limited evaporation.

e. Contextualization of theoretical results in a practical setting—namely, virus transmission through evaporating droplets.

Evaporation and Sedimentation of Water Drops

Subject Tags: evaporation, aerosols, droplets, transport phenomena, heat and mass transfer

Difficulty: Moderate

Estimated Time: 60 minutes

This problem explores the evaporation dynamics and sedimentation of water droplets relevant to aerosol transmission of viruses. You will apply transport correlations including convective mass and heat transfer, use the Buck Equation for vapor pressure, and compare analytical and numerical approaches to determine the largest droplet evaporating before hitting the ground from mouth height.

Skills Tested:

- Application of Sherwood, Nusselt, Reynolds, Prandtl, and Schmidt number correlations.
- Use of vapor pressure relations (Buck Equation).
- Analytical solution of coupled sedimentation and evaporation without convection.
- Numerical solution considering Reynolds number effects via Ranz-Marshall correlations.

Evaporation and Sedimentation of Water Drops

During the pandemic it became clear that a key way (probably the key way) the virus is transmitted is via aerosols. While large droplets fall rapidly to the ground, smaller drops become aerosols that float about, waiting to be inhaled. The amount of virus a drop contains is proportional to its volume – so the transition of a large drop into a small one via evaporation is critical. In this problem I want you to answer a simple question: What is the largest drop of water falling from an altitude of 1.6 meters (e.g., about where the average mouth is) that evaporates before hitting the ground, and how does this compare to the roughly 25μm diameter drops I emit when I say "puppy" or "polymer"? For this calculation, take the air to be at 22°C at 40% relative humidity. While the Reynolds number of the falling droplets is small, it's not zero. Many correlations for mass, momentum, and energy are available, but for your calculations use the following:

$$Nu = 2 + 0.6\, Re^{1/2} Pr^{1/3}$$
$$Sh = 2 + 0.6\, Re^{1/2} Sc^{1/3}$$
$$\frac{F}{6\pi\mu U a} \approx \left(1 + \frac{1}{6} Re^{1/2} + \frac{1}{60} Re\right)$$

Use the Buck Equation for the vapor pressure of the water. Note that a saliva droplet would behave a little differently because of the stuff dissolved in it, but it is still 99% water. As a result, the droplet nuclei left behind is small enough to float about in the air, and the calculation for a clean drop of water (e.g., what I'm asking you to do) gets it about right.

a. Ignoring convection (e.g., if the Reynolds number is zero in the correlations above), you can actually solve this analytically once you've got the equilibrium concentration at the surface (which as you recall is not a function of drop size). In this limit the sedimentation velocity of the drop is proportional to the radius squared, which in turn is linear in time, so it's easy to get h as a function of t (and thus R). If you take $h = 0$ as the point where $R = 0$, then you can determine the radius of our evaporating drop at any height. Do this to determine the value of R_0 where $h = 1.6$ m.

b. The Reynolds number isn't zero, however, so your complete solution will have to be numerical. The simplest way to approach it is to pick an initial droplet size, calculate the velocity via root finding on the drag equation, and then solve the coupled mass and energy transport problem via the Ranz-Marshall equations above. This should give you an evaporation rate as a function of drop radius, which you can then use to get a change in height as well as radius as a function of time. If you then plot up h as a function of R you would get a master curve for all drops, and taking $h = 0$ as the point where $R = 0$ you can determine the radius at a height of 1.6 m, the same as was done for your analytic solution ignoring the Reynolds number dependence.

What You Should Learn from Problem Set 11

Problem 1: Air-Water Stripping Tower Design:

- **a.** Understanding the use of empirical correlations to estimate diffusivities of trichloroethylene in water and air, including the importance of temperature-dependent viscosity corrections.
- **b.** Application of mass transfer coefficients ($k_L, k_G, and\, K_L^o a$) and Henry's Law constant to practical engineering problems.
- **c.** Calculation of tower height required to meet specified contaminant removal goals under given flow conditions.

Problem 2: Equimolar Counter-Diffusion with First-Order Reaction (Boundary Layer Limit):

- **a.** Formulating the diffusion-reaction problem in a "flat Earth" boundary layer approximation and recognizing the simplification to an exponential concentration profile.
- **b.** Derivation of surface flux and calculation of the effectiveness factor for fast, diffusion-limited reactions.
- **c.** Comparison of the boundary layer limit solution to the asymptotic results from lecture notes, consolidating understanding of limiting behaviors.

Problem 3: Taylor Dispersivity Transition via Monte Carlo Simulation:

- **a.** Implementing a Monte Carlo simulation to model solute dispersion in laminar flow within a circular tube.
- **b.** Demonstrating numerically the long-time linear growth of variance consistent with Taylor's classical dispersivity formula.
- **c.** Identifying the dimensionless time (or distance) scale required to reach steady state dispersion and interpreting transient behaviors.
- **d.** Developing skills in particle boundary reflection handling and random walk diffusion in two dimensions.

Air-Water Stripping of Trichloroethylene from Groundwater

Subject Tags: Mass transfer, Environmental engineering, Air-water stripping
Difficulty: Moderate
Estimated Time: 90 minutes

This problem focuses on the removal of trichloroethylene (TCE) from groundwater at 55°F using an air-water stripping tower loaded with 1-inch metal Raschig rings. You will calculate diffusivities, mass transfer coefficients, and use Henry's Law to estimate the required tower height to achieve a given reduction in TCE concentration.

Skills Tested:

- Use of empirical correlations to determine diffusivities in air and water.
- Calculation of mass transfer coefficients (k_L, k_G, $K_L^o a$).
- Application of Henry's Law for gas-liquid equilibrium.
- Design considerations for packed tower height based on desired contaminant removal.

Air-Water Stripping of Trichloroethylene from Groundwater

You are assigned the task of stripping out trichloroethylene from groundwater (55°F) using an air-water stripping tower. Your tower is loaded with 1 inch metal Raschig Rings, and is to be operated at a water superficial flow rate of 26 gpm/ft^2 and air superficial flow rate of 48 cfm/ft^2.

a. To calculate mass transfer coefficients you will need to get the diffusivities of trichloroethylene in both water and air (at 55°F). This can be done using a number of empirical correlations, and a nice summary specific to trichloroethylene in air and water is found on p. 7 & 8 of the reference:

`https://dtsc.ca.gov/wp-content/uploads/sites/31/2018/01/tce.pdf`

Note that while this is a government publication, it actually has an error in it! For their diffusivity in water expression they forgot to include the dependence of viscosity on temperature – not huge, but not insignificant either! When using this correlation, be sure to also use the correct viscosity at our operating temperature!

b. For these conditions, calculate k_L, k_G, and $K_L^o a$. You will find table 4 on p. 75 of Staudinger's thesis to be of use to get the Henry's Law coefficient. A typical surface/volume ratio for this size Raschig Ring is 216 m^2/m^3.

Recall that Staudinger's thesis may be found at:

`https://vtechworks.lib.vt.edu/bitstream/handle/10919/101153/LD5655.V855_1986.S735.pdf?sequence=1`

c. It is desired to use the above conditions to reduce the concentration of trichloroethylene from 1 ppmw to 10 ppbw. How tall should the tower be for these flow rates?

Equimolar Counter-Diffusion with First Order Reaction — Boundary Layer Limit

Subject Tags: Diffusion, Reaction engineering, Thiele modulus, Boundary layer
Difficulty: Moderate
Estimated Time: 60 minutes

This problem examines the limiting case of a fast first order reaction under equimolar counter-diffusion conditions where the reaction is severely diffusion limited. By adopting a boundary layer approximation, you will derive the concentration profile as a simple exponential, calculate the flux at the surface and the resulting effectiveness factor, and compare your results to the known asymptotic limits.

Skills Tested:

- Applying boundary layer approximations to reaction-diffusion problems.
- Deriving exponential concentration profiles under diffusion limitation.
- Calculating surface flux and effectiveness factors.
- Comparing analytical solutions to asymptotic limits.

Equimolar Counter-Diffusion with First Order Reaction — Boundary Layer Limit

Consider an equimolar counter-diffusion / first order reaction Thiele modulus problem such as was covered in class. If the reaction is fast (e.g., you are severely diffusion limited) it is more appropriate to solve the problem in the boundary layer limit: We define

$$y = R - r$$

and examine reaction and diffusion in this "flat Earth" limit.

Show that the problem formulated in this way is just a simple exponential for the concentration distribution, and solve for the flux at the surface and resulting effectiveness factor. Compare your result to the asymptotic limit derived in the notes and show that they agree.

Taylor Dispersivity Transition via Monte Carlo Simulation

Subject Tags: Dispersion, Taylor dispersivity, Monte Carlo simulation, Diffusion, Fluid mechanics
Difficulty: Moderate
Estimated Time: 90 minutes

This problem explores the transition to the Taylor dispersivity limit in a circular tube flow using a Monte Carlo simulation approach. You will simulate the dispersion of an initially focused solute slug, analyze the long-time variance growth consistent with Taylor's classical result, and determine the time or distance scale over which steady state dispersion is achieved.

Skills Tested:

- Setting up Monte Carlo simulations of solute dispersion in tube flow.
- Applying dimensionless analysis to interpret dispersion phenomena.
- Implementing boundary reflection for particles at the tube wall.
- Computing variance growth and identifying asymptotic limits.
- Analyzing transient behavior to find steady state attainment time.

Taylor Dispersivity Transition via Monte Carlo Simulation

The Taylor dispersivity limit is reached only after solute molecules have a chance to diffuse across streamlines so that they sample the different convective velocities. In this problem we examine this transition. While this can be done analytically, the problem is more than a bit messy – so we will do it using a Monte Carlo simulation.

a. Simulate the dispersion of an initially focused solute slug in a circular tube of radius a with diffusivity D and average velocity U. After rendering the problem dimensionless, show (via simulation!) that at long times the variance grows as

$$2Kt \quad \text{where} \quad K = \frac{1}{48}\frac{U^2a^2}{D}$$

(i.e., Taylor's classic result).

b. Subtracting this asymptote off, determine the (dimensionless) time (or distance, same thing!) at which the steady state is achieved.

For this problem it is useful to do your random walk diffusivity in both the x and y directions (e.g., Cartesian coordinates), where

$$r = \sqrt{x^2 + y^2}.$$

Reflection at the tube wall is simple: your new radius would be

$$r_{\text{new}} = \min(r, 2 - r),$$

and your new x and y positions would be scaled by the new radius:

$$x = x\frac{r_{\text{new}}}{r}, \quad y = y\frac{r_{\text{new}}}{r}.$$

It's just a couple of lines of code. As was done in the class simulations, use the Trapezoidal Rule to get the displacement in the flow direction. Ignore diffusion in the axial direction, as that would be very small at large Peclet numbers. Your initial distribution can be set up in a square box (using the `rand` command to get it uniform) and then you just trim out the tracers which fall outside the circle of radius 1.

What You Should Learn from Problem Set 12

Problem 1: Concentration polarization in simple shear:

- **a.** Formulating convective-diffusion equations and applying appropriate boundary conditions in a shear flow context.
- **b.** Understanding how to nondimensionalize governing equations and identify dominant transport mechanisms.
- **c.** Using scaling arguments to justify neglecting diffusion along the flow direction.
- **d.** Setting up and solving asymptotic concentration distributions analogous to classical heat/mass transfer problems (e.g., Nusselt–Graetz analogy).
- **e.** Applying mass balance constraints to determine unknown coefficients in the solution.

Problem 2: Electrostatics and electroosmosis at a charged glass surface:

- **a.** Calculating the Debye length to characterize the thickness of the electric double layer.
- **b.** Relating surface potential to surface charge density on a charged interface.
- **c.** Applying the Boltzmann distribution to predict ion concentration profiles near charged surfaces.
- **d.** Integrating free charge distributions to verify overall electro-neutrality of the system.
- **e.** Balancing electrical body forces and viscous drag to compute electroosmotic flow velocity under an applied tangential electric field.

Concentration Polarization in Simple Shear Flow

Subject Tags: concentration polarization, shear flow, membrane filtration, transport phenomena

Difficulty: Moderate

Estimated Time: 2 hours

Concentration polarization in membrane filtration systems can lead to increased concentration at the membrane surface, causing fouling or reduced efficiency. This problem explores a tangential simple shear flow and its effect on the steady-state concentration polarization layer, including formulation of governing equations, scaling arguments, and solution techniques.

Skills Tested:

- Formulation of convective-diffusion equations with boundary conditions
- Non-dimensionalization and scaling of partial differential equations
- Analytical solution approaches to asymptotic steady-state concentration distributions
- Application of mass balance and flux boundary conditions in membrane transport problems

Source: CBE 30356 Problem Set 12 **File:** hw1121.tex

Concentration Polarization in Simple Shear Flow

Consider a tangential flow given by a simple shear velocity profile

$$u_z = \dot{\gamma} y,$$

where y is the distance normal to the membrane surface located at $y = 0$. The membrane filters out a species with diffusivity D at a constant permeate flux velocity

$$u_y|_{y=0} = -u_0.$$

The upstream concentration (and the concentration far from the membrane in the y direction) is a constant c_0, and the boundary condition at the membrane itself is a no-flux condition where diffusion balances convection.

a. Write down the convective-diffusion equation and the boundary conditions for this problem, and render the equations dimensionless.

b. Show by scaling analysis that diffusion in the flow (z) direction is negligible.

c. Set up the equation for the asymptotic concentration distribution as a function of z and y. Note that your dimensionless concentration distribution will resemble the transient dead-end filtration problem. (If the velocity profile were flat rather than a simple shear, it would be mathematically identical.)

d. Solve the equation. Note that (similar to the Nusselt-Graetz constant heat flux problem) the last unknown coefficient is determined by applying a mass balance condition at $z = 0$ (the entering flow average concentration must be c_0).

Electrostatics and Electroosmosis at a Charged Glass Surface

Subject Tags: Debye layer, surface charge, Boltzmann distribution, electroosmosis, electro-neutrality

Difficulty: Moderate

Estimated Time: 2 hours

This problem explores the electric double layer at a negatively charged glass surface in an aqueous NaCl solution. It covers calculation of the Debye length, surface charge density, ion distributions using the Boltzmann distribution, verification of electro-neutrality, and electroosmotic velocity under an applied tangential electric field.

Skills Tested:

- Calculating Debye length in electrolyte solutions
- Relating surface potential to surface charge density
- Using Boltzmann distribution for ion concentration profiles near charged surfaces
- Integrating charge distributions to confirm electro-neutrality
- Balancing electrical and viscous forces to find electroosmotic velocity

Source: CBE 30356 Problem Set 12 **File:** hw1122.tex

Electrostatics and Electroosmosis at a Charged Glass Surface

A glass surface is negatively charged with surface potential $-100\,\mathrm{mV}$. It is immersed in a 1 mM aqueous NaCl solution at 298 K. Answer the following:

a. Calculate the Debye layer thickness.

b. Calculate the surface charge density.

c. Using the relationship between the potential and ion charge concentration provided by the Boltzmann distribution, plot the distribution of Na^+ and Cl^- counterions in the diffuse layer and the net free charge (i.e., the difference between the two).

d. By integrating the free charge distribution, show that the entire system (charged membrane plus diffuse layer) satisfies the electro-neutrality condition (i.e., no net charge).

e. A tangential electric field of $100\,\mathrm{V/cm}$ is applied along the surface. This results in a force per unit volume on the net free charge in the double layer, resisted by viscous drag. By balancing these two mechanisms, calculate the electroosmotic velocity resulting from the applied field.

CBE 30356 Problems of the Day

The following problems were solved during the lectures for CBE 30356 as "Problems of the Day". The detailed contents of each lecture and the worked out solutions to the PODs may be found in the course directory at `https://diffusivepress.org/courseware/cbe30356/` as well as the narrated lecture notes. While not assigned to the students as homework or exam questions, they were quite similar to both (and can easily be adapted to such). Because of their source, unlike the other problems in this Compendium some of the files require a bit of modification (generally in consultation with the provided solutions) before they can be integrated into an exam or problem set. Also, not every class had a POD associated with it, as some reviewed material before an exam and others reviewed material necessary for that week's homework assignment.

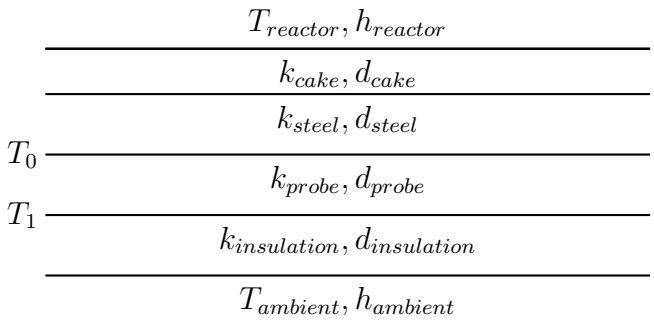
Some of the problems here are original, however many of them are classics. In particular, a number are directly drawn from examples in BSL (the assigned text for the course) with only minor modifications.

Lecture 1 POD: Calculation of Thermal Conductivity of Gases

In this problem of the day we calculate the thermal conductivity of N_2 by application of the Eucken formula. This uses the dynamic viscosity, so that needs to be calculated as well from the Chapman-Enskog equations using the expression provided in BSL. We also need the heat capacities, which are obtained from `https://www.engineeringtoolbox.com`. We compare the results graphically with the viscosity and conductivity data from engineeringtoolbox.com.

Lecture 2 POD: Design of a "Heat Transfer Stethoscope"

The Idaho National Lab waste treatment facility has a reactor which is prone to forming a sediment cake layer at the walls. It is necessary to determine when the cake layer is forming without any probe internal to the reactor. In this example we look at the possibility of an external temperature probe measuring the thickness of the internal cake layer. We do a "back of the envelope" calculation to determine the feasibility of such a "heat transfer stethoscope". The probe concept is depicted below:



The idea is that if you know the internal and external heat transfer coefficients, the conductivities and thicknesses of the insulation layer, probe, and steel wall, then by measuring T_1 and T_0 on either side of the probe you can get the extra resistance provided by the cake layer. Knowing its thermal conductivity you can estimate the thickness. Consult the solution to get approximate numbers for formulating a quantitative exploration of this concept.

Lecture 3 POD: Analysis of the SNAP-27 RTG

Energy sources for space missions have always been a problem: solar panels are great, but if the sun is very distant or you are in shade they won't do much good! One solution has been the RTG, a radioisotope thermoelectric generator. In this system a radioactive fuel (often Plutonium 238) radioactively decays via alpha emission, releasing heat and not much else (that penetrates, anyway). This heat source is coupled with a heat sink and the resulting temperature gradient is directly converted to electricity via the Seebeck effect. It's not very efficient, but it is very reliable - pretty important when you are in deep space! In this problem we will examine the design of the SNAP-27 reactors which were used to power instruments on all the manned lunar missions. A description of the device may be found at `https://www.osti.gov/servlets/purl/4758004`. From the information in this report, conduct a "back of the envelope" analysis of its heat transfer characteristics. Some specific questions to answer are:

a. What is the "effective temperature" of the radiators?

b. What is the effective thermal conductivity of the annular RTG?

c. What is the temperature of the central void?

Lecture 4 POD: Radiation Fins in Space

Cooling via radiation is very important in spacecraft. In this problem we look at the cooling provided by a single isolated cooling fin of width 2b which loses heat by radiation from both sides. Energy propagates up from the base (z=0) via conduction. We assume the fin to be a black body, with no back radiation (e.g., in shadow!). Two questions to answer: what is the relationship between the temperature at the base and the heat flux, and how long should it be before increased length no longer does you much good? Analyze the problem first by scaling and then by numerical solution to the scaled equations. Demonstrate that this numerical solution matches the exact solution obtained by the trick of multiplying the differential equation by the temperature gradient and then simply integrating.

Lecture 5 POD: Large Biot Heat Transfer from a Sphere

In this problem we investigate how the Sturm-Liouville solution for a sphere at large Biot number can be used to calculate an effective internal heat transfer coefficient. Often rather than looking at the details of the temperature distribution, it is useful to calculate transient changes in energy via a heat transfer coefficient based on the volume averaged temperature and the surface area. This is particularly useful after the "initial transient" has passed away and we are dominated by the lead term of the eigenfunction expansion. In this problem we apply it to the heating of a well stirred bucket of water with a hot rock: what fraction of the total thermal energy of the hot rock is released at short times, and what it the heat transfer coefficient at long times. The characteristic time for this transition (governed by the second eigenvalue) is also determined.

Lecture 6 POD: Transient Heating of a Slab

In this problem we investigate the transient heating of a slab of width $2b$, where heat is entering from both sides at a constant rate. Because there is no way for the heat to escape, the temperature increases linearly in time - but there is also an initial transient that must be solved via a Sturm-Liouville solution approach. As with any of these linear problems, you first obtain the asymptotic solution (which now is a function of time), and then subtract it off to obtain the Sturm-Liouville problem for the decaying part. Using this approach, obtain the complete solution at all times and plot up the temperature profile for select times up to a dimensionless time of $t^* = 0.4$, together with the asymptotic solution. Note that the important parts of the result are the asymptotic solution (which you would use to get an asymptotic heat transfer coefficient) and the lead eigenvalue of the decaying solution (which quantitatively tells you when the asymptotic solution becomes valid).

Lecture 7 POD: Quenching of a Sphere at Finite Bi

In this problem we solve the problem of the temperature distribution in a sphere of radius a which is initially at a temperature T_1 and is quenched in a bath maintained at a temperature T_0. Heat transfer to the bath is governed by a constant heat transfer coefficient h. Render the problem dimensionless and show that it is governed by the Biot number. While an analytic solution exists (the solution to this an many other heat transfer problems may be found in the classic text Carslaw and Jaeger, Conduction of Heat in Solids (1959)), however it is a little messy. Instead, solve the problem numerically using the finite difference routine introduced in class. Identify the three required functions $p(r)$, $q(r)$, and $w(r)$ required by the SL formulation, as well as the boundary condition (which involves Bi) and plug it in! Plot up the temperature profile for $Bi = 2$ at select times, as well as the lead eigenvalue as a function of Bi.

Lecture 8 POD: Neumann Instability in Finite Difference Solutions

In this problem we explore the effect of step size for the transient solution of problems using the finite difference marching method. First, obtain the exact solution to the classic problem of a slab of width $2b$ with uniform heat source per unit volume $\dot{S}$ where the temperature at $y = b$ is maintained at a constant T_0. Note that this is most easily approached by just solving for the "top half", and using an insulated condition at $y = 0$. Now implement the marching solution technique discussed in class using a second order difference scheme for boundary condition at the plane $y = 0$. Choosing a time step just below the Neumann condition, plot up the temperature at $y = 0$ for dimensionless times up to $t^* = 2$ and compare the numerical solution to the exact solution. Then do this again for a time step just above the critical value, and watch the solution explode!

Lecture 9 POD: Transient Heating of a Semi-Infinite Slab

In this problem we explore the classic problem of the heating of a semi-infinite slab (e.g., just bounded at $y = 0$). The slab is initially at a temperature T_0 and at time $t^* = 0$ it is subjected to a constant heat flux q_0 at $y = 0$. So:

a. Render the problem dimensionless

b. Show via either scaling or affine stretching that the problem admits a self-similar solution, identifying the similarity rule and variable in canonical form.

c. Determine the temperature at $y = 0$ as a function of time to within some unknown O(1) constant.

d. Obtain the transformed ODE governing the problem and boundary conditions.

e. While the differential equation actually admits an analytic solution in terms of an integral of the complementary error function, instead solve the problem numerically via the shooting method as discussed in class, obtaining the O(1) number (e.g., the unknown shooting parameter) that you need for the wall temperature.

Lecture 11 POD: Scaling: Squishing a Fluid Between Two Plates

Scaling of equations is a critical skill to master! Here we apply it to the classic fluid mechanics problem of the squishing of a fluid of volume V_0 between two plates initially separated by a distance H_0 (which would use the corresponding fluid radius R_0 from the geometry). The idea is to see how the filled radius $R(t)$ varies in time if the upper plate exerts some force Mg on the fluid. So:

a. Write down the equations and boundary conditions that govern this problem in the limit $H/R << 1$. This includes the radial momentum equation, continuity equation, force balance on the upper plate, and conservation of volume condition.

b. We ignore inertial effects here (say that the gap is small, or the viscosity is large), so the time scale is determined from a volume balance on the fluid. Using this, write down the equation for how the radius of the fluid varies with time.

c. Scale all of the equations to determine the characteristic time scale for the expansion.

d. Solve the now dimensionless problem to show that the dimensionless radius R^{*8} grows linearly with dimensionless time t^* and obtain the O(1) constant.

Lecture 12 POD: Step Change in Wall Temperature for Tube Flow

The classic problem of the heating of fluid undergoing laminar flow through a tube with constant wall heat flux is useful because it has a simple analytic solution for the heat transfer coefficient far down the tube (e.g., Nu = 48/11). In fact, however, there are many possible boundary conditions! The Graetz-Nusselt problem is classically given as a step change in wall temperature, rather than a constant heat flux. In this case the heat transfer coefficient (based on the difference between the flow average temperature and wall temperature) must be calculated from the lead term of the corresponding Sturm-Liouville eigenvalue problem. There's no analytic solution to this one because of the quadratic velocity profile, but it is easily done numerically as shown in class. So:

a. Write down the differential equation and boundary conditions governing this problem and render them dimensionless.

b. Using the usual separation of variables technique, set up the corresponding Sturm-Liouville problem.

c. Solve the problem numerically and determine the asymptotic value of the Nusselt number.

d. Plot up the local Nu as a function of distance down the tube and compare the value to the asymptote and the corresponding value for wall heating. About how far down the tube do you have to go before the asymptote becomes valid?

Note that BSL provides a table of asymptotic values of Nu for many geometries and boundary conditions.

Lecture 14 POD: Buoyancy Driven Flow for a Vertical Wall

Consider a vertical wall with characteristic heigh H. The wall is maintained at a constant temperature T_0 and the fluid far away has a temperature T_∞. The temperature difference and coefficient of thermal expansion β yields a vertical buoyancy driven boundary layer next to the wall. So:

a. Write down the differential equation and boundary conditions governing this problem and render them dimensionless.

b. From scaling alone, determine the rate of heat loss from the wall to within some O(1) value (that will be a function of the Prandtl number). Use the thermal diffusivity scaling for the boundary layer thickness (it just moves the Prandtl number around in the equations - you can't get rid of it).

c. Using either scaling or stretching, show that the problem admits a similarity solution and obtain the similarity variable and rule in canonical form.

d. Obtain the transformed differential equations and boundary conditions. Set the problem up as a system of first order ODEs and solve them numerically using the shooting method.

e. What is the O(1) constant needed to determine the heat flux for air ($Pr = 0.7$)?

Lecture 15 POD: Analysis of the Webb Telescope Sun Shield

In this problem we examine arguably the most expensive radiation shield ever made: the tennis court sized multilayer radiation shield for the Webb space telescope, and examine why it has its rather peculiar looking design. A description of the sunshield is provided at `https://webb.nasa.gov/content/observatory/sunshield.html`. So:

a. Start your analysis by considering the heat transfer between two parallel planes with some emissivity e. Determine the relationship between the radiative heat flux between the planes and their two temperatures.

b. Extend your analysis to n parallel planes in series.

c. The final layer simply emits to space with heat flux $q = e\sigma T_n^4$ with no back radiation (ideally). Use this to obtain the relationship between T_1 and T_n.

d. For the Webb radiation shield we have an emissivity of $e = 0.04$ and $n = 5$ layers. If $T_1 = 383^o K$, what is T_n?

e. While this result should match their description of the maximum temperature of layer 5, it is nowhere near good enough for telescope operation! That is why the layers are both angled and open at the outside edge (so heat leaks out the sides and isn't constant as you have been assuming). While the detailed analysis is a bit more complicated, how many times would a photon bounce between two layers (on average) before being absorbed? The idea is that you angle the layers so that they escape out the edges first...

Lecture 17 POD: Diffusion through a Stagnant Spherical Layer

Consider a spherical drop of radius R_1 of liquid A evaporating into gas B. The gas is modeled by a stagnant spherical shell of radius R_2. The equilibrium mole fraction of species A at the surface is x_{A1} and that at the edge of the stagnant film is x_{A2}. Your goal is to determine the rate of evaporation (molar flux N_{Ar} at $r = R_1$), a classic problem.

a. Assuming pseudo-steady-state and no gas phase reaction, write down the equation and boundary conditions governing the problem.

b. Solve the problem for finite values of the equilibrium mole fraction. You will get a logarithmic relationship between the two mole fractions and the total flux (e.g., $4\pi R_1^2 N_{Ar}$).

c. Take the limit as the difference in the two radii becomes small, and show that this collapses to the planar result.

Note that in this problem we have assumed some value of x_{A1}. In general this depends strongly on temperature, and that depends on the rate of evaporation. Thus the full solution for an evaporating drop needs solution to the energy equation too!

Lecture 18 POD: Thiele Effectiveness Factor for a Flat Particle

The classic result for Thiele's Effectiveness Factor (the ratio of the reaction rate to that if there were no mass transfer limitations) is derived for a spherical catalyst particle. Catalysts come in many shapes, however, so here we derive the relationship for an even simpler geometry: the flat plate. Consider a plate of thickness $2b$ where a gas A has a concentration at the surfaces $y = \pm b$ is c_{A0}. It is undergoing a first order reaction with rate (per unit volume) of $R_A = -ka\, c_A$ and is undergoing equimolar counter-diffusion with diffusivity D_A (the usual assumptions). So:

a. Write down the differential equation governing the problem and render it dimensionless. Show that it depends on a single dimensionless group (the Thiele modulus).

b. Solve the equation to determine the flux at the surface.

c. Comparing this to the rate if there were no mass transfer resistances, determine the effectiveness factor

d. Plot up the result for the flat particle and compare it to the result for a sphere.

Note that these are pretty dissimilar geometries, yet the effectiveness factor is very similar for both! This is also true for a cylindrical particle which lies between the two results.

Lecture 19 POD: Dissolution with First Order Reaction

In this problem we look at the dissolution rate of a gas from a spherical bubble in a gas-liquid sparger. The idea is that the gas diffuses into the liquid and undergoes a slow homogeneous first order reaction. The bubbles are modeled as being surrounded by a thin stagnant film of thickness $\delta << R$ where R is the bubble radius. The homogeneous reaction rate is $R_A = -kc_A$ (moles/time/vol). The equilibrium concentration at the surface of the bubble is c_{A0} and that in the liquid outside the stagnant layer is at a concentration $c_{A\delta}$. The idea is to determine how much the diffusion resistance in the (thin) stagnant layer reduces the overall reaction rate. So:

a. The layer is thin, so we work in the "flat earth limit". Using this, write down the differential equation and boundary conditions governing the problem in the thin layer (assume pseudo steady-state) and render them dimensionless. Show that in addition to the concentration ratio $c_{A\delta}/c_{A0}$ it depends on one other dimensionless group (the Hatta number, analogous to the Thiele modulus).

b. Solve the equation to determine the flux at the surface of the gas bubble in terms of these parameters.

c. To close the problem, we recognize that the flux at the surface of the stagnant layer (e.g., δ) must equal the rate of consumption in the rest of the vessel. If the volume of liquid (outside the negligible volume of the stagnant layers) is V and the surface area of all the bubbles is A, use this to determine the ratio $c_{A\delta}/c_{A0}$ in terms of the Hatta number and the dimensionless volume $V/A\delta$. This can be plugged back into the flux at the bubble surface to determine the overall reaction rate as a function of these parameters.

Note that if you get desperate, this problem is completely worked out in BSL...

Lecture 20 POD: Mass Transfer Limitations in a Biodetector

Diffusion is a slow process, and this can be a real issue in a microfluidic biodetector! In order for a probe (often on one wall of a microchannel) to see the target biomolecules they have to diffuse to the surface. We also want the process to be -fast- to improve response time, as well as -sensitive- so we don't want target molecules to get away (and where we can process a relatively large amount of fluid). We are essentially looking for the needle in the haystack! So:

a. Consider a microchannel that is 100 μm deep and 200 μm wide. If our flow rate is 0.2 $\mu l/min$, what is the average and centerline velocity of the fluid?

b. Our detector is on the bottom wall, and measures 200 μm wide by 500 μm long. it is assumed that it can detect any target molecule that gets to the surface. Based on this, what is the minimum solute diffusivity that will allow detection of half the target molecules?

c. It is concluded that this isn't good enough - we want to look at proteins for this application. Using the Stokes-Einstein diffusivity to relate the diffusion to molecular weight, determine how the molecular weight of the target molecules which can be successfully captured depends on the geometry and flow rate.

Hint: This problem is much easier if you look at how much -escapes- at the end of the channel without diffusing to the detector...

Lecture 21 POD: Calculation of H_{OX} for a Stripping Tower

In this problem you are assigned the problem of determining the height of a transfer unit H_{OX} for stripping cholorform out of water with air. The packing is taken to be 1 inch Pall rings, and the operating temperature is 25^oC. The liquid flow rate is 22.6 gpm/ft^2 and the gas flow rate is 84 cfm/ft^2. Because of the high surface tension of water it is necessary to use the Onda correlation to determine the fraction of the packing which is wetted.

a. To start, make a list of all the equations and parameters that you will need to complete this calculation.

b. Look up values for all of these parameters, and calculate the dimensionless ratios required by the correlations.

c. Calculate K_L, K_G, and K_L^o. Are we liquid side or gas side limited?

d. Complete the calculation to get H_{OX}.

Lecture 22 POD: Taylor Dispersion in Rectangular Channels

A key problem in microfluidic systems is controlling the axial dispersion of a slug as it travels down a channel. Because (for pressure driven flow) molecules move with different velocities on different streamlines, any slug spreads out due to Taylor dispersion. While the dispersivity is easily calculated for circular tubes or between parallel planes, it gets more complicated for typical rectangular channels. In fact, for large aspect microchannels the dispersion is -dominated- by the small volumes of fluid at the sides where the fluid is slowed down by the sidewalls! Here we use the very simple technique of Monte-Carlo simulation to calculate the dispersivity in such a system.

To do the simulation, we take our microchannel to have a depth of 2b and width of 2a. By symmetry we only need to consider one quarter of the channel speeding things up!. We normalize both x and y by the thin direction by $y^* = y/b$ so the side walls (which cause all the problem) are located at $x^* = ab$ where $ab = 5$ is the aspect ratio.

a. First, write down the axial velocity in terms of the average velocity in the central region (with a dimensionless value of unity).

b. We start our code by randomly distributing 1000 "particles" with y coordinates between zero and 1 and x coordinates between zero and ab. The z coordinate for all of them starts off at zero. You will want to keep track of the z variance of the particles with respect to time, so you will need to set up a t vector and a z variance vector for plotting.

c. Now we update positions inside a loop! First, calculate the z velocity based on the initial x and y positions. Then give the particles a random "kick" in both the x any y direction of magnitude $(2\Delta t^*)^{1/2}$ using the `randn` command (if doing this in Matalb). The time step needs to be very small (about $\Delta t^* = 5 \times 10^{-5}$) to keep the random walk small.

d. Reflect the particles that leave your domain back in and recalculate the velocities.

e. Using the Trapezoidal Rule, update the z position of all the particles and calculate and record the variance.

f. Plot up the variance as a function of time and compare it to the exact Taylor Dispersion limit for parallel plate flow (no side walls).

g. Now do it again including the effect of side walls. This is easily approximated by taking the z velocity to be zero for all particles within a distance of 5/8 from the sides (e.g., $x^* > ab - 5/8$). Your dispersion should now be about 8 times that for a channel without sidewalls!

Note that this code is easily modified for all sorts of cross-sections. It turns out that you can greatly reduce this pernicious effect by simply making the regions near the sidewalls a small bit deeper, increasing the velocity in those regions.

Lecture 23 POD: MC Simulation of AF4 Separations

A commercially available technique for separating large molecules and nanoparticles is the AF4 separator, which stands for Asymmetric Flow Field Flow Fractionation. The idea is that the molecules to be separated are (after focusing on one wall) subjected to a channel flow where the lower wall (where they are located) is porous and thus has a small negative velocity. At equilibrium the concentration polarization layer for the molecules with a larger diffusion coefficient is a little thicker, projecting further out into the shear flow. Thus, those molecules are convected a little faster and separation occurs.

A key innovation of this technique was to make things focused at the wall in a narrow axial band right at the start. This is cleverly accomplished by the loading procedure: initially flow of the solute containing fluid is inward at both ends and only out through the porous wall. That way all of the solute molecules are convected to one narrow band whose location depends on the ratio of the two velocities coming in (e.g., where the axial flow vanishes). Once loaded, flow is only in one direction (apart from the porous wall flow) and separation occurs.

In this problem we wish to look at the separation efficiency of such a device via simulation, calculating the rate of separation as well as Taylor dispersion due to the finite thickness of the concentration polarization layer and shear flow. This is easily done via MC simulation.

The dimensionless differential equation for a simple shear flow is given by:

$$\frac{\partial c}{\partial t} + y\frac{\partial c}{\partial x} - \frac{\partial c}{\partial y} = \frac{\partial^2 c}{\partial y^2}$$

with initial condition $c = e^{-y}$. This can be simulated with a simple shear flow of unit magnitude in the x direction and a random "kick" of $(2\Delta t)^{1/2}$ times a normally distributed random number (e.g., `randn`) in the y direction. The permeation velocity (what led to the y derivative term in the differential equation) is just -1 in the y direction (the displacement in the y direction is thus the sum of these two mechanisms), and the membrane is simulated by reflection at $y = 0$.

a. Set up the simulation as described above. The initial distribution of particles is easily accomplished by using as initial condition `y = -log(rand(n,1))`.

b. Plot up the mean position and variance of our slug as a function of time. You should find that it moves with a dimensionless velocity of 1 and a variance which grows linearly as 4t in the Taylor limit as expected from the analytical results derived in class.

c. Now add in a second molecule with dimensionless diffusivity D2 (rendered dimensionless with respect to the first particle). Note that your initial condition should also be adjusted by this diffusivity (e.g., `y2 = -D2*log(rand(n,1))`) as the thickness of the concentration polarization layer is proportional to the diffusivity.

d. Show that the mean velocity of the two species differs by D2-1, but you will find that it takes a really long time for the two peaks to be separated due to the dispersion! For

a diffusivity ratio of 1.5 you will have to carry the simulation out to a dimensionless time of O(500) with a time step of $\Delta t = 0.001$. It is possible to use the analytic result for the velocity and dispersivity to calculate the necessary separation length or time.

CBE 30356 Quizzes

The following quizzes are assigned before and after each lecture of CBE 30356. Note that when administered I used the "randomize answers" approach, and for many of these here they have **not** been randomized, so they would need to be adjusted before being placed in an exam without that option. The quizzes are administered out of Canvas, and the .xml zip file containing the quizzes (as well as the solutions) in Canvas format are available to faculty on request. The detailed contents of each lecture may be found in the course directory at `https://diffusivepress.org/courseware/cbe30356/` as well as the narrated lecture notes.

Lecture 1: Fourier's Law & the Origin of Thermal Conductivity

1. The units of energy are important! Order the following by magnitude from lowest to highest.

(1–5) Description	(A–E) Order
1 erg	A. smallest
1 joule	B. second smallest
1 cal	C. the one in the middle
1 kcal	D. largest
1 BTU	E. second largest

2. From the theory of heat transfer in low density gases, which of the following is (are) true?

   (A) The thermal conductivity increases with pressure.

   (B) The thermal conductivity increases as $T^{1/2}$.

   (C) The thermal conductivity is proportional to the temperature.

   (D) The thermal conductivity of a low density gas is much less than that of liquids and solids.

   (E) None of these statements are correct.

3. Match the following fluids to the correct mechanism for heat conduction.

(1–3) Fluid	(A–C) Mechanism
neon	A. Molecules physically changing places.
glycerin	B. Molecular interactions moving at the speed of sound.
aluminum	C. Motion of conduction band electrons.

4. For thermal conductivity in solids, mark which if any are true.

   (A) The thermal conductivity of dry sand is greater than that of wet sand.

   (B) The thermal conductivity of wood is isotropic.

   (C) The thermal conductivity is always able to be represented by a scalar.

   (D) The thermal conductivity of steel is greater than that of copper.

   (E) None of these statements is true.

Lecture 2: One Dimensional Heat Conduction in Solids

1. There are many parallels between heat, mass, and momentum transfer! Match the following examples of heat and momentum transfer.

(1–3) Description	(A–C) Term
1. plane-Couette flow	A. Heat conduction through a slab
2. plane-Poiseuille flow	B. Heat conduction in a slab with heat generation
3. Poiseuille flow	C. Radial heat conduction in a wire with electrical dissipation

2. Heat loss through a wall is reduced by adding insulation. If the heat loss Q through a wall without insulation is 10 (arbitrary units) and the loss through the same wall is reduced to 2 (same units) by adding insulation, what would be the heat loss if we double the insulation thickness?

   (A) 0.4
   (B) 1.11
   (C) 1
   (D) 3/2
   (E) There is insufficient information to answer this question.

3. One dimensional conduction through a slab can be modeled by a heat transfer coefficient h. What is the representation for h in terms of the thermal conductivity k and thickness d?

   (A) $\frac{k}{d}$
   (B) $\frac{d}{k}$
   (C) $\frac{k}{d^2}$
   (D) $\frac{d^2}{k^2}$

4. Interfacial boundary conditions are important in both heat and momentum transport (and mass transport too). What quantities are constant at an interface? Mark all that are correct.

   (A) velocity
   (B) temperature
   (C) heat flux
   (D) shear stress
   (E) temperature gradient
   (F) shear rate

Lecture 3: Heat Transfer with Generation

1. A hot water pipe is made of 1 inch OD copper and is at a temperature (at the surface) of 120°F. The ambient temperature is at 70°F. We want to reduce the heat loss from the pipe by adding foam insulation of thickness 0.5 inches and thermal conductivity $k = 0.025$ W/m°K. If the external heat transfer coefficient is 18 W/m^2°K, what is the ratio of the heat loss with insulation to that without? Assume that the external heat transfer coefficient is the same in both cases, and there is no internal heat transfer resistance.

   (A) 1.0

   (B) 0.0014

   (C) 0.146

   (D) 0.85

   (E) There is insufficient information to answer the question.

2. Consider heat generation in a slab of width $2b$. The surfaces of the slab at $y = \pm b$ are at a reference temperature of 0, the heat generation per unit volume is S, and the thermal conductivity is k. What is the heat flux at $y = b$?

   (A) $\dfrac{Sb^2}{k}$

   (B) Sb

   (C) $\frac{1}{2}\dfrac{Sb^2}{k}$

   (D) $\dfrac{1}{2}Sb$

   (E) There is insufficient information to answer the question.

3. For the problem in question 2, how much hotter is the temperature at the centerline $y = 0$ than that at the wall?

   (A) $\frac{1}{2}Sb$

   (B) $\frac{1}{2}\dfrac{Sb^2}{k}$

   (C) $\frac{1}{4}\dfrac{Sb^2}{k}$

   (D) Sb

   (E) There is insufficient information to answer the question.

4. For the slab in question 2, what is the ratio of the centerline temperature to the average temperature in the slab?

   (A) 1

   (B) $\frac{3}{2}$

   (C) 2

   (D) There is insufficient information to answer the question.

Lecture 4: Cooling Fins and Heat Transfer Coefficients

1. You are trying to increase the rate of heat transfer from a reactor by adding rods or fins to the outside. You have decided to use 2 cm diameter steel rods. If the thermal conductivity of the steel is 45 W/m$^2{}^\circ$K and the heat transfer coefficient is 20 W/m$^\circ$K, what is a reasonable length for the rods?

   (A) 10 cm

   (B) 2 cm

   (C) 100 cm

   (D) There is insufficient information to provide an estimate.

2. You are going to have to get used to knowing the solutions to simple ODE's. A differential equation which often occurs in transport is $y'' + a^2 y = 0$ where the prime denotes the derivative and a is a constant. What are the solutions to this equation?

   (A) $\sin(ax)$

   (B) $\cos(ax)$

   (C) $\sinh(ax)$

   (D) $\cosh(ax)$

   (E) $\tanh(ax)$

3. Unlike conduction from a sphere, there is no solution to conduction from a cylinder in an infinite medium (we get away with cylindrical insulation because it is of finite thickness). The corresponding problem in fluid mechanics is called:

   (A) Stokes' Paradox

   (B) Stokes' Law

   (C) D'Alembert's Paradox

   (D) Bernoulli's Equation

   (E) There are no parallels between heat and momentum transfer in this geometry.

4. Air is blowing past a cylinder of diameter 3 cm at a velocity of 4 m/s. What is the external heat transfer coefficient?

   (A) 44 W/m$^2{}^\circ$K

   (B) 64 W/m$^2{}^\circ$K

   (C) 18 W/m$^2{}^\circ$K

   (D) 0.867 W/m$^2{}^\circ$K

Lecture 5: The Nusselt & Biot Numbers and the Quenching of a Sphere

1. A classic way of producing lead shot was to pour molten lead through a porous plate and then have it fall a great height. You can still see such a "Shot Tower" built in the early 1800s adjacent to the New River Trail in Austinville, VA. The trick to making a shot tower work is to have a great enough height for the lead drops to solidify before hitting the water below, and in the case of the one in Virginia they actually built it into the New River Gorge. Anyway, all the quiz questions today deal with this problem.

   For the first question, if the drops of lead were 1.5 mm in diameter, what would be their terminal velocity in air? Assume a drag coefficient of 0.5.

   (A) 42 cm/s

   (B) 13 m/s

   (C) 19 m/s

   (D) 42 m/s

2. What is the heat transfer coefficient at this velocity?

   (A) 436 $W/m^{2\circ}K$

   (B) 237 $W/m^{2\circ}K$

   (C) 43 $W/m^{2\circ}K$

   (D) 12 $W/m^{2\circ}K$

3. What is the Biot number for our falling lead drop?

   (A) 0.0092

   (B) 36

   (C) 1.24

   (D) 0.124

4. Putting it all together, what is the characteristic fall height associated with the cooling process? Ignore the heat capacity of phase transitions - that would make it longer - but this height gets things about right...

   (A) 1.6m

   (B) 16m

   (C) 160m

   (D) 5 km

Lecture 6: The Sturm-Liouville Theorem and the Startup of a Heated Slab

1. In the lecture the transient heating of a slab was solved by subtracting off the asymptotic solution and solving for the decaying solution as a series expansion. At the centerline of the slab, what was the ratio of the first and second term of this series solution evaluated at a dimensionless time $t^* = 1$?

   (A) 1

   (B) 9:1

   (C) 27:1

   (D) 10^{10} : 1

2. A wire is heated via electrical dissipation and cooled at the wall. You want to determine the transient temperature distribution, and this yields the Sturm-Liouville boundary value problem:

$$(rf')' + s^2 rf = 0$$

$$f'(0) = 0; \quad f(1) = 0$$

   To get the coefficients in the series expansion you need to know what the weight function is (so that the eigenfunctions for different eigenvalues are orthogonal). What is it?

   (A) r

   (B) r^2

   (C) 1

   (D) s^2

3. Water (kinematic viscosity 1cs) is initially at rest in a channel of half-width 1mm. At time $t = 0$ you impose a pressure gradient and the water begins to move. Based on today's lecture, how long do you have to wait for the centerline velocity to reach 99% of its asymptotic value?

   (A) 1.9 s

   (B) .019 s

   (C) 0.94 s

   (D) The lecture was on heat transfer and has no bearing on fluid flow.

4. A cylinder is initially at a temperature of T_0. At time $t = 0$ you begin heating it at the outside (raising it to a temperature T_1 at the surface) and you also introduce a uniform heat source per unit volume in the interior (e.g., you stuffed it in the microwave oven). You obtain the asymptotic solution at long times, but you want to obtain the transient solution too! What would be the differential equation and boundary conditions of the appropriate Sturm-Liouville problem?

(A)

$$(rf')' + c^2 rf = 0$$
$$f'(0) = 0; \quad f(1) = 0$$

(B)

$$(rf')' + c^2 rf = S$$
$$f'(0) = 0; \quad f(1) = 0$$

(C)

$$(rf')' + c^2 rf = 0$$
$$f'(0) = 0; \quad f(1) = 1$$

(D)

$$f'' + c^2 f = 0$$
$$f'(0) = 0; \quad f(1) = 1$$

(E) This problem will not admit a Sturm-Liouville solution.

Lecture 7: Matrix Methods for Numerical Solutions to SL Problems

1. You are trying to obtain an estimate of the derivative of a function and have evaluations at three node locations $x_1 = 1$, $x_2 = 1.5$, and $x_3 = 2$. If the values of the function at these locations are $y_1 = 0$, $y_2 = 1$, and $y_3 = 1.5$, what is the best estimate of the derivative at x_2?

   (A) 1.5
   (C) 1
   (B) 2
   (D) 0.75

2. You are trying to evaluate the derivative at a boundary (e.g., $x_0 = 0$). You have function evaluations there and at $x_1 = 0.5$ and $x_2 = 1$ which have the values $y_0 = 0$, $y_1 = 1$, and $y_2 = 1.5$. What is the best estimate of the derivative at x_0 from these values?

   (A) 2.5
   (C) 1.5
   (B) 2
   (D) 1

3. You will shortly be solving the convective diffusion equation for flow through a pipe governed by the differential equation:

$$2(1-r^2)\frac{\partial T}{\partial z} = \frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right)$$

   If you use separation of variables you get a Sturm-Liouville problem for the spatial part. What would be the weight function for this problem used in obtaining the coefficients?

   (A) $2(1-r^2)r$
   (C) $2(1-r^2)$
   (B) r
   (D) 1

4. You are trying to improve the accuracy of the lead eigenvalue obtained using the numerical solver for a Sturm-Liouville problem. You used 40 intervals, but wanted to do better. About how many intervals should you use to reduce the error in the calculated eigenvalue by a factor of 5?

   (A) 20
   (C) 90
   (B) 40
   (D) 200

Lecture 8: Finite Difference / Marching Methods for Numerical Solutions to Non-Linear Problems

1. You are solving the dimensionless heat transfer equation:

$$\frac{\partial T}{\partial t} = \frac{\partial^2 T}{\partial x^2}$$

using the finite difference approach described in class. If your spatial discretization is $\Delta x = 0.01$, what is the maximum Δt you can use?

(A) 0.00005
(B) 0.0001
(C) 0.005
(D) 0.01

2. Suppose that the problem in question 1 was non-linear so that the conductivity varied from a dimensionless value of 1 to a dimensionless value of 1/2 over the domain. What would be the limit on Δt in this case? (Hint: a bit of a trick question: think about it!)

(A) 0.00005
(B) 0.0001
(C) 0.005
(D) 0.01

3. In a marching solution the values of the dependent variable at the boundaries are determined from the boundary conditions after updating the values at all the interior points. Suppose that we have the no-flux condition at $x = x_0 = 0$ (left edge of the domain). In updating the temperature T_0 at $x = x_0$ using this boundary condition, what should we use?

(A) $T_0 = T_1$
(B) $T_0 = 0$
(C) $T_0 = \frac{4}{3}T_1 - \frac{1}{3}T_2$
(D) $T_0 = \frac{4}{3}T_1 - \frac{1}{3}T_2 + \frac{2}{3}\Delta x$

4. More practice setting up Sturm-Liouville problems. In the (possibly) melting reactor problem you solved for homework at steady-state, the corresponding transient problem is governed by the dimensionless differential equation:

$$\frac{\partial T}{\partial t} = \frac{\partial^2 T}{\partial x^2} + 1 + C^* T$$

With boundary conditions:

$$T|_{x=1} = 0; \quad \left.\frac{\partial T}{\partial x}\right|_{x=0} = 0; \quad T|_{t=0} = 0$$

What would be the ODE corresponding to the Sturm-Liouville problem governing the decaying solution?

(A) $f'' + C^* f + \lambda f = 0$

(B) $f'' + 1 + C^* f + \lambda f = 0$

(C) $f'' + 1 + \lambda f = 0$

(D) $f'' + \lambda f = 0$

(E) The problem as written is non-linear, and thus will not admit a Sturm-Liouville solution.

Lecture 9: Boundary Layer Solutions to Transient Problems

1. A key benefit to obtaining a self-similar solution to a problem is that you can learn a tremendous amount about how things behave without getting the complete solution — just from knowing the similarity rule and similarity variable in canonical form you can determine quantities to within an "O(1) constant" — which you would actually have to solve the now ODE to get, of course. For the problem discussed in the notes, use this concept to determine how the heat flux at the plate varies with time.

   (A) t^{-1}
   (B) $t^{-\frac{1}{2}}$
   (C) t^0 (e.g., constant in time)
   (D) $t^{\frac{1}{2}}$
   (E) t

2. The analogous problem to the step change in temperature at the wall is the constant heat flux case, e.g.,

$$-k\left.\frac{\partial T}{\partial y}\right|_{y=0} = q_0$$

   This problem yields the same similarity variable, but a more complicated similarity rule. Which of the following similarity rules would satisfy Morgan's Theorem for this boundary condition and yield a similarity solution? Mark all that are correct as there is more than one!

   (A) $\dfrac{T^*}{y^*} = f(\eta)$
   (B) $\dfrac{T^*}{t^{*1/2}} = f(\eta)$
   (C) $\dfrac{T^* t^*}{y^{*3}} = f(\eta)$
   (D) $T^* = f(\eta)$

3. Which of the similarity rules in question 2 is in canonical form?

   (A) $\dfrac{T^*}{y^*} = f(\eta)$
   (B) $\dfrac{T^*}{t^{*1/2}} = f(\eta)$
   (C) $\dfrac{T^* t^*}{y^{*3}} = f(\eta)$
   (D) $T^* = f(\eta)$

4. For the problem posed in question 2 (the constant heat flux case), how does the temperature at the wall vary with t?

   (A) t^{*0} (e.g., constant)
   (B) $t^{*1/2}$
   (C) t^*
   (D) $t^{*3/2}$

Lecture 10: Periodic Heating / Penetration Depth

1. The average temperature in South Bend over the course of a year is about 49°F and pipes, of course, tend to freeze about 32°F. That means that you need to determine the *frost line:* the location where it is likely to propagate down to during the winter months. If the average temperature in January is 24°F, how deep is the frost line? Take the thermal diffusivity of the soil to be 0.006 cm^2/s (it varies a lot with soil type). Note that recommendations are usually to bury a foot below the average frost line, and of course adding in a bit for a really cold winter or conductive soils is a good idea!

   (A) 3.1 ft
   (B) 5.7 ft
   (C) 1.2 ft
   (D) 4.3 ft

2. Suppose you have a probe located at the frost line calculated (correctly!) for problem 1. Around here the coldest day of the year is (on average) January 22. Based on this, when will the probe likely reach its minimum temperature?

   (A) Valentine's Day
   (B) Ground Hog Day
   (C) New Year's Day
   (D) Christmas Day
   (E) St Patrick's Day

3. A plate is oscillating with an angular frequency of 10 radians/s (in plane, not up and down!) with an amplitude of 1 cm. On top of the plate you have a 0.5 cm thick layer of corn oil (kinematic viscosity of 0.56 cm^2/s). You scatter parsley flakes on the surface and examine their motion. What would be the approximate amplitude of their oscillation?

   (A) 0.3 cm
   (B) 0.5 cm
   (C) 0.7 cm
   (D) 1.0 cm
   (E) This is fluid mechanics and has no relation to heat transfer!

4. In many areas the Trans-Alaska Pipeline is supported above ground on pillars sunk into the permafrost: the depth at which the ground is always frozen. Since this is often a frozen peat bog (hard as a rock until it melts) global warming can be a bit of a problem! Suppose your yearly average temperature rises above the freezing point. What is the most cost effective solution?

   (A) Sink your foundations deeper so that you are further away from the surface.
   (B) Add chillers so that you locally keep the bog around your pipe support foundations frozen.
   (C) Reroute your pipeline.

Lecture 11: Convective Heat Transfer: Derivation and Scaling of Equations

1. Suppose we have a dimensionless temperature field given by:

   $$T = 6t - 3x$$

   (e.g., increasing with time and decreasing in the x-direction.)

   Our local velocity vector is given by $(2, 1, 0)$ in the x, y, and z directions.

   Given this, what is the time rate of change in temperature of a fluid element?

   (A) 6
   (B) 3
   (C) 0
   (D) -3
   (E) There is not enough information to answer the question.

2. A fluid is flowing into a pipe which is heated at the wall. Which is greater, the flow average temperature or the area average temperature?

   (A) Flow average temperature.
   (B) Area average temperature.
   (C) There is insufficient information to answer the question.

3. A 0.1 cm radius heated sphere is falling through a viscous fluid (low Reynolds numbers!) with thermal diffusivity of 0.001 cm^2/s. At what velocity does convection play a significant role in the heat transfer?

   (A) 0.01 cm/s
   (B) 0.0001 cm/s
   (C) 0.1 cm/s
   (D) 1 cm/s

4. A shaft 5 cm in diameter is rotating at a speed of 1000 RPM. It is lubricated by a fluid with viscosity of 1 poise (0.1 Pascal-seconds, the SI measure of viscosity) and a thermal conductivity of 0.15 W/m°K. The shaft is cooled, but you are asked to determine if viscous heating is likely to build up a significant temperature rise in the lubricating gap. From the *scaling of the equations*, estimate the magnitude of the temperature rise. Note that the numerical value would be a bit less than this depending on boundary conditions, so this is a bit of an upper bound. Hint: you can assume that the lubricating region is just a simple shear flow (thin gap) and it turns out the gap width doesn't matter...

   (A) 4°C
   (B) 0.2°C
   (C) 46°C
   (D) 630°C

Lecture 12: The Graetz-Nusselt Problem

1. What is the weight function for the Sturm-Liouville equation associated with the Graetz-Nusselt problem?

   (A) $2r(1-r^2)$ (C) 1

   (B) r (D) 0

2. Far down a conduit of arbitrary (but uniform) cross-section with constant heat flux at the wall the Nusselt number approaches an asymptotic value. What can we say about the transient leading to this asymptote? Mark all that are correct!

   (A) Nu will be larger regardless of cross-section.

   (B) Nu will be larger because the developing thermal boundary layer causes the temperature profile near the wall to be compressed.

   (C) Nu is always smaller in the entrance region.

   (D) Sometimes Nu would be larger, sometimes smaller in this region depending on the geometry.

3. Suppose you are designing a heater for a microfluidic channel of depth 100μm. An aqueous solution is flowing through the channel with velocity of 1 cm/s. Your heating element is 5 mm long. Can you use the asymptotic heat transfer coefficient, or are you in the thermal boundary layer region?

   (A) Yes, your heater is long enough that the asymptotic value of Nu is appropriate.

   (B) No, you are in the thermal boundary layer development region because your heater is so short.

   (C) Hard to say, it looks like you are in the transition region.

4. You are processing a viscous liquid (with the same thermal diffusivity as water) that requires pumping it through a heater. Your heater is a 2 cm ID pipe 1 m long, and your fluid velocity is 10 cm/s. Your heater is operating at constant heat flux. You find that some of the fluid is being thermally degraded, and you want to reduce the near-wall temperature rise by a factor of two. What can you do?

   (A) Using a recycle pump, increase the fluid velocity by a factor of two.

   (B) Using a recycle pump, increase the fluid velocity by a factor of four.

   (C) Using a recycle pump, increase the fluid velocity by a factor of eight.

   (D) There is no way to reduce the wall temperature without decreasing the heat flux.

Lecture 13: Nusselt Number Correlations and Heat Exchangers

1. Water is flowing through a 1cm ID pipe at a velocity of 1 m/s. Assume that the temperature is 30 ^{o}C. What is the internal heat transfer coefficient based on the Colburn analogy? Use the properties from the engineeringtoolbox.com website.

   (A) 5000 W/m^2 ^{o}K
   (B) 2000 W/m^2 ^{o}K
   (C) 1000 W/m^2 ^{o}K
   (D) 8748.6 W/m^2 ^{o}K

2. Glycerin is flowing through a 1cm ID tube at a velocity of 1 m/s. We have the following properties at 25 degrees C:

   - $k = 0.284$ W/m ^{o}K
   - $\nu = 7.1$ stokes
   - $\alpha = 9.5 \times 10^{-4}$ cm^2/s

   If the tube is 1 m long, what would be the heat transfer coefficient if we kept the tube wall at a fixed temperature?

   (A) 482 W/m^2 ^{o}K
   (B) 3925 W/m^2 ^{o}K
   (C) 127 W/m^2 ^{o}K
   (D) 66 W/m^2 ^{o}K

3. A double pipe heat exchanger is operated in **co-current** mode. The hot stream enters with a temperature of 80 ^{o}C and exits with a temperature of 55 ^{o}C. The cold stream enters with a temperature of 20 ^{o}C and exits with a temperature of 50 ^{o}C. For these conditions, what is the log-mean delta T?

   (A) 22 ^{o}C
   (B) 30 ^{o}C
   (C) 35 ^{o}C
   (D) 60 ^{o}C
   (E) 5 ^{o}C

4. You decide to operate the heat exchanger from question 3 in counter-current mode, yielding the same inlet and outlet temperatures. This can be done with a smaller area heat exchanger (for the same heat transfer coefficient). What would be the ratio of the area of the counter-current exchanger to co-current exchanger yielding the same temperatures? Hint: If the flow rates and temperatures don't change, neither does Q.

   (A) 0.68
   (B) 0.35
   (C) 0.93
   (D) 0.82

Lecture 14: Nusselt Number Correlations for External Flows

1. A cube of 10 cm on a side is suspended in a stream of air moving at 2 m/s. If the cube is at a temperature of 40°C and the air is at 20°C, what is the approximate rate of heat loss? Note that you can easily look up the properties of air on engineeringtoolbox.com, but take the properties to be:

$$k = 0.0259 \text{ W/m}^\circ\text{K}$$
$$\nu = 0.15 \text{ stokes}$$
$$\alpha = 0.22 \text{ cm}^2/\text{s}$$

(A) 1.2 W
(B) 18 W
(C) 425 W
(D) 3 W

2. Consider the cube of problem 1, only now you turn off the air flow. What is the approximate rate of heat loss in this case?

(A) 18 W
(B) 6 W
(C) 2 W
(D) 245 W
(E) 0 W: There's no flow!

3. As was described in the lecture, laminar flow up a plane due to natural convection is self-similar: the scaling parameter for x (e.g., H, the height of the plane) disappeared from the problem but did not have to be specified. Using the results of scaling (or you can do affine stretching to check if you wish!), what is the variation of the vertical velocity u_x with x?

(A) $x^{1/4}$
(B) x^0 (e.g., constant)
(C) $x^{1/2}$
(D) $x^{3/4}$
(E) x

4. A window pane measures 1 m tall by 0.5 m wide. It has a surface temperature of 5°C (it's cold out there!) and the room temperature is 20°C. Given this and the properties of air, what would be a rough estimate of the flow rate of the draft rolling of the window pane? Note that I just want the scaling here: you would need to solve the whole problem to get the O(1) number multiplying it!

(A) 2.2 liters/s
(B) 43 liters/s
(C) 0.1 liters/s
(D) There is not enough information provided to answer the question.
(E) 8.7 liters/s

Lecture 15: Spectral Effects in Radiative Heat Transport

1. Electromagnetic radiation can arise from the following sources. (mark all that are correct)

   (A) A change in vibrational energy state.
   (B) A change in rotational energy state.
   (C) Displacement of outer shell electrons.
   (D) Displacement of core shell electrons.
   (E) Nuclear interactions.
   (F) AC current in a conductor.

2. A sphere of radius 5 cm is suspended in air. It is coated with carbon black, so its emissivity is 1 (a black body). The surroundings are at room temperature (20°C) as is the air. If the surface temperature of the sphere is 100°C, what is the ratio of the net energy transfer due to radiation to that from natural convection? (Hint: Use the Churchill correlation on p. 213 of the class notes.)

   We have the following properties of air at 20°C:

   - $\nu = 0.15$ cm^2/s
   - $\alpha = 0.22$ cm^2/s
   - $k = 0.026$ W/m $\cdot$ K

   (A) 0.26
   (B) 1.25
   (C) 1.86
   (D) 0.13
   (E) 0.80

3. Air is blowing past the sphere of problem 1 at a wind speed of 10 m/s. What is the ratio of the net energy flux due to radiation to that arising from forced convection? (Hint: use the Whittaker relation on p. 202 of the course notes. Because it is air, you can ignore the correction due to the viscosity ratio — the correction would be less than 5% for these temperatures.)

   (A) 0.18
   (B) 1.25
   (C) 0.62
   (D) 0.02

4. Two gray planes with emissivity of 0.4 are exchanging energy via radiation (e.g., they are in a vacuum!). We place a radiation shield between them, but the only one available was terrible: it had an emissivity of 1 (e.g., a black body). What is the ratio of the energy transmitted with the shield to that without?

   (A) 0.8 - Not great, but better than nothing.
   (B) 1.0 - Because it is a black body it doesn't do any good.
   (C) 0 - The black body absorbs all the radiation, so nothing gets through.
   (D) 1.2 - Strangely, having a black plane in between actually increases the amount of radiation transferred.

Lecture 16: Fick's Law and the Conservation Equations

1. A 20nm (that's nanometers) diameter ribosome is diffusing in water at 25°C. What is its diffusivity?

   (A) 2.2×10^{-7} cm^2/s

   (B) 1.8×10^{-8} cm^2/s

   (C) 1.6×10^{-5} cm^2/s

   (D) 2.2×10^{-9} cm^2/s

2. A small amount of ethanol is poured into the bottom of a 10 cm long test tube (filled with air) and capped off. In the absence of convection (which is really hard to do!) approximately how long would it take for the air to become uniformly saturated?

   (A) 1 second

   (B) 10 seconds

   (C) 20 minutes

   (D) 1 day

   (E) 4 months

3. You do the same experiment, but now the tube is filled with water rather than air. In the absence of convection about how long would it take for the ethanol to become uniformly distributed by diffusion alone?

   (A) 1 s

   (B) 10 s

   (C) 10 minutes

   (D) 1 day

   (E) 4 months

4. Benzene has a viscosity of 0.65cp at room temperature and a density of 0.876 g/cm^3. Using this, estimate the self-diffusivity of benzene at this temperature.

   (A) 2×10^{-5} cm^2/s

   (B) 2.2×10^{-7} cm^2/s

   (C) 2.2×10^{-3} cm^2/s

   (D) 2.2×10^{-6} cm^2/s

Lecture 17: The Stefan Tube

1. In the lecture the relation between the evaporative flux and material parameters was obtained. What is measured in the laboratory is the change in the fluid level (or the rate with which fluid has to be replenished to keep the level constant). What is this rate of change?

   (A) $-N_{Az}\Big|_{z=0}\frac{M}{\rho_l}$

   (B) $-N_{Az}\Big|_{z=0}\frac{1}{M\rho_l}$

   (C) $-N_{Az}\Big|_{z=0}$

   (D) $-N_{Az}\Big|_{z=0}/c$

2. At 25°C the vapor pressure of diethyl ether is 0.7 atm. If you are measuring the diffusivity of diethyl ether in air, what should you use as x_{A0}?

   (A) 0.7

   (B) 0.3

   (C) 1.4

   (D) 0.7 moles/liter

3. What is the total molar concentration c at 25°C and 1 atm?

   (A) 0.0244 moles/cm^3

   (B) 4.1×10^{-5} moles/cm^3

   (C) 22.4 moles/cm^3

   (D) 0.0224 moles/cm^3

4. OK, now we use all this! You conduct a diffusion experiment of diethyl ether in air in a 2 cm high Stefan tube at 25°C. If the diffusivity of diethyl ether in air is 0.079 cm^2/s, the molecular weight M is 74.12 g/mol and the liquid density is 0.713 g/cm^3, what is the rate of change of the liquid level in your tube?

   (A) 2.0×10^{-4} cm/s

   (B) 1.2×10^{-4} cm/s

   (C) 2.7×10^{-6} cm/s

   (D) 4.0×10^{-4} cm/s

Lecture 18: Gas Phase Mass Transfer with Reaction: The Thiele Modulus

1. A heterogeneous reaction of the form $A+B \rightarrow C$ occurs at a surface. The three species A, B, and C are diffusing through a gas film of thickness δ in the z direction. What can we say about the molar flux of the three species? Mark all that are correct!

   (A) $N_{Az} = N_{Bz}$

   (B) $N_{Az} = N_{Cz}$

   (C) $N_{Az} + N_{Bz} + N_{Cz} = N_{Az}$

   (D) $N_{Cz} = -\frac{1}{2}\left(N_{Az} + N_{Bz}\right)$

   (E) $N_{Az} = \frac{-cD_A}{1-x_A}\frac{\partial x_A}{\partial z}$

2. Diffusion in a porous medium is modeled by an effective diffusivity D_A. What can we say about this quantity?

   (A) If pore diameters are large with respect to the mean free path length of the gases, then it should be proportional to D_{AB}.

   (B) It is always equal to D_{AB}.

   (C) It is roughly proportional to the void fraction.

   (D) It is roughly inversely proportional to the tortuosity of the pores.

3. Suppose we have a dimerization reaction $2A \rightarrow B$ occurring in a porous spherical catalyst pellet. Which of the following are true?

   (A) $N_{Ar}|_{r=0} = 0$

   (B) There is a net molar flux into the pellet.

   (C) There is a net molar flux out of the pellet.

   (D) $r^2 N_{Ar} = \text{constant}$

   (E) This is an example of equimolar counter-diffusion.

4. Ernest Thiele was a famous Chemical Engineer. Which of the following are true?

   (A) He was known for developing the graphical method for solving staged operation (distillation) problems.

   (B) He was known for codifying the balance between diffusion limited and reaction limited reactions in catalyst pellets.

   (C) He has a lectureship at Notre Dame named in his honor, in which he attended the first one at age 90.

   (D) He lived so long ago that none of your professors could have met him.

Lecture 19: Diffusion with Homogeneous Reaction in Liquids

1. The Hatta number was introduced in the diffusion with reaction problem discussed in this lecture. What is its physical interpretation? Mark all that are correct.

   (A) It equals the film depth/ diffusion length.

   (B) It is the square root of the ratio of the rate of consumption of A to the diffusion of A.

   (C) It is the ratio of the rate of reaction of species A to the rate of convection of A.

   (D) It is the square root of the ratio of reaction to convection.

2. Suppose we had a second order consumption reaction rather than a first (e.g., $R_A = -k_2 c_A^2$). Which of the following are true?

   (A) Scaling still works fine, and would yield a new dimensionless group equivalent to the Hatta number, but which involves c_{A0}.

   (B) The asymptotic concentration can still be obtained from an ODE, solved numerically this time.

   (C) The transient problem still admits a separation of variables solution.

3. For the Sb_2S_3 hole diffusion problem the decay of the hole concentration will quickly be exponential in time (e.g., dominated by the lead eigenvalue). If there were no interfacial mass transfer resistance what would be this decay rate?

   (A) $e^{-(\lambda+\pi^2/4)t^*}$

   (B) $e^{-\lambda t^*}$

   (C) $e^{-\pi^2/4\, t^*}$

   (D) There is no decay.

   (E) There is insufficient information to answer the question.

4. In the opposite limit (large interfacial resistance, low Biot number) what would be the observed decay rate?

   (A) $e^{-(\lambda+\pi^2/4)t^*}$

   (B) $e^{-\lambda t^*}$

   (C) $e^{-\pi^2/4\, t^*}$

   (D) There is no decay rate in the low Biot number limit.

   (E) There is insufficient information to answer the question.

Lecture 20: Convective Diffusion into a Falling Film

1. A layer of water is falling down a vertical wall with a film depth of 500 μm. What is the Reynolds number?

   (A) 400
   (B) 4
   (C) 0.04
   (D) 4000

2. About how long would a 100 μm deep film of water have to be before we approach the asymptotic mass transfer coefficient for a species with diffusivity 10^{-5} cm^2/s? (Hint: don't forget to include the appropriate eigenvalue in your estimation!)

   (A) 1.5 cm
   (B) 33 cm
   (C) 3.3 km
   (D) 15 m

3. Now let's look at the thin penetration calculation for the mass transfer coefficient. The diffusivity of 111-trichloroethane in water (an unfortunately common pollutant) is 0.96×10^{-5} cm^2/s. If water is flowing down a plate at a velocity of 1 m/s (and we assume that it is thin enough to still be laminar!) and the length of the plate is 5 cm, approximately what would be the mass transfer coefficient between the bulk fluid and the interface?

   Note: This is really an approximation of the film flowing down a packing element in a stripping tower.

   (A) 0.02 cm/s
   (B) 0.002 cm/s
   (C) 2 cm/s
   (D) There is not enough information to make an estimate, as it depends on how thick the film is.

4. The Sherwood number is a dimensionless mass transfer coefficient analogous to the Nusselt number for heat transfer. It is defined as:

$$Sh = \frac{k_m d}{D_{AB}}$$

   For typical operating conditions of a gas-liquid contactor (e.g., the discussion at the end of the lecture), how should the Sherwood number depend on the diffusivity in the liquid?

   (A) $D_{AB}^{1/2}$
   (B) $D_{AB}^{-1/2}$
   (C) D_{AB}^{-1}
   (D) D_{AB}
   (E) It is approximately independent of D_{AB}.

Lecture 21: Gas-Liquid Stripping: Nox, Hox, and Overall Mass Transfer Coefficients

1. There are many parallels between heat and mass transfer at interfaces! Key to understanding these parallels is knowing your boundary conditions. Which of the following are true?

   (A) In heat transfer, the temperature is continuous across an interface.

   (B) In heat transfer, the heat flux is continuous across the interface.

   (C) In mass transfer, the concentration is continuous across the interface.

   (D) In mass transfer, the mass flux is continuous across the interface.

2. Consider the case where we are liquid side mass transfer limited (e.g., $K_L^o = k_L$). The height of a transfer unit H_{ox} depends on the liquid flow rate. Based on the Onda model, how should it scale with the superficial liquid flow rate L?

   (A) $L^{1/3}$

   (B) L

   (C) $L^{-1/3}$

   (D) $L^{-2/3}$

3. We want to remove 99% of the pollutant in a water stream. If we choose the gas flow rate so that the stripping ratio $R = 2$ (e.g., twice the minimum value for removal), what would be the required number of transfer units N_{ox}?

   (A) 8

   (B) 4

   (C) 50

   (D) 1

   (E) There is not enough information to answer the question, as the Henry's Law constant is not specified.

4. You are stripping a VOC out of water with air at atmospheric pressure and 15°C. You are told that the Henry's law constant on a molar basis (e.g., $m = y/x$) is 500 at that temperature. If you poured your polluted water into a bottle with equal volume of air and water in it and shook it up, what fraction of the total amount of pollutant would be in the air?

   (A) 0.28

   (B) 0.38

   (C) 1.0

   (D) 1/500

   (E) There is not enough information to answer the question.

Lecture 22: Taylor-Aris Dispersion

1. All four questions today deal with the diffusion of colloidal particles in a capillary with a radius of 25 μm filled with water at 25°C. The variance of the distribution of the particles is being observed as a function of time. For this question, the fluid is at rest and the variance is found to be (after a really really long time) fit by $\sigma^2 = 0.01 + 4.4 \times 10^{-8} t$ cm^2 where t is measured in seconds. Based on this, what is the radius of the colloidal particles? (Hint: Remember Stokes-Einstein diffusivity, and you might want to look through the pod20.pdf notes from last week...)

   (A) 100 nm
   (B) 1 μm
   (C) 1 nm
   (D) 10 nm

2. OK, now we have some flow! If the fluid is driven through the tube via a pressure gradient with a velocity of 0.1 cm/s, what would be the half-width of the distribution at a detector 1 m downstream? The half-width of the peak can be taken to be the square root of the variance.

   (A) 11 cm
   (B) 1.1 cm
   (C) 110 cm
   (D) 1.1 mm

3. The asymptotic Taylor-Aris dispersivity is reached only after the particles (or molecules, or tracers, etc.) have a chance to sample all the different streamlines. From scaling alone, about how far down the capillary would this be achieved? Note that there is an eigenvalue to the SL problem of about 10 or so in this geometry, so the actual distance is about an order of magnitude less than you get for this scaling! I just want the scaling answer here...

   (A) 30 cm
   (B) 3 cm
   (C) It isn't reached before you run out of capillary.
   (D) 3 mm

4. The observed spread occurring in pressure driven flow is quite convenient for observation and calculation of the particle dispersivity and thus size. If the goal is separation, however, one would like to keep the slug more compact! A way of doing this is drive the flow via electroosmosis, which (ideally, anyway) yields a flat velocity profile that can be approximated by perfect slip at the walls. For the same velocity, initial variance, etc., of our slug of particles, what would be the peak half width (square root of the variance) at our detector 1 m from the entrance?

   (A) 0.1 cm
   (B) 1 cm
   (C) 10 cm
   (D) 0.007 cm
   (E) There is insufficient information to make an estimate.

Lecture 23: Concentration Polarization and Field-Flow Fractionation

1. The Asymmetric Flow Field Flow Fractionation technique discussed in class is particularly well suited to the separation of nanoparticles. It can be used to get a good measure of the size distribution of such particles by examining the elution as a function of time. In all the problems of today's quiz we are going to look at the "numbers" for getting the distribution of liposomes with average diameter of 100 nm using AF4. So:

   If we are doing the separation of these liposomes in water at room temperature, what is the expected diffusivity?

   (A) 4.4×10^{-8} cm^2/s
   (C) 4.4×10^{-10} cm^2/s
   (B) 4.4×10^{-12} cm^2/s
   (D) 4.4×10^{-6} cm^2/s

2. An AF4 unit with a channel height of 250 μm is to be used for the separation. If it is desired to have a resolution of about 20% on the diameter of the liposomes, about what should the cross-flow through the membrane at the accumulation wall be?

   (A) about 10^{-4} cm/s
   (C) about 10^{-6} cm/s
   (B) about 10^{-5} cm/s
   (D) about 10^{-7} cm/s

3. If the length of your separation cell is 5 cm and half of your flow is going through the membrane, about what should be the tangential sweep velocity at the inlet (averaged over the depth, as was done in the notes)?

   (A) about 0.04 cm/s
   (C) about 0.0004 cm/s
   (B) about 0.004 cm/s
   (D) about 0.4 cm/s

4. What would be the expected elution time of our 100 nm liposomes under these conditions?

   (A) 30 min
   (C) 3 hr
   (B) 3 min
   (D) 1.2 min

Lecture 24: Concentration Polarization in Rotating Membrane Systems

1. It is proposed to filter a dilute aqueous solution of $CaSO_4$ using a dead end RO membrane filter. If there is no stirring (a really bad idea, of course), and the concentration is 0.01 g/liter, about how long will it take before our membrane plugs? The permeate flux is 0.001 cm/s, the diffusivity of $CaSO_4$ is 7×10^{-6} cm^2/s and the solubility limit is about 2 g/liter.

   (A) 20 minutes
   (B) 2 seconds
   (C) 3 hours
   (D) 20 seconds

2. A commercial track-etched membrane has the following properties: pore density = 6×10^8 1/cm^2, pore diameter = 100 nm, and membrane thickness = 25 μm. If we are running water through at a pressure of 1 bar, what would be the expected permeate flux?

   (A) 0.006 cm/s
   (B) 0.3 cm/s
   (C) 6×10^{-5} cm/s
   (D) 0.01 cm/s

3. In our calculation of the expected concentration polarization layer for a rotating membrane we assumed that the flow was laminar. If the rotation is fast enough, however, it will be turbulent — and mixing would be very different! The transition in this geometry occurs at a Reynolds number of about 500 based on the radius, the rotation rate, and the boundary layer thickness. For the 1000 RPM rotation rate of the 1.45 radius filter in Prof. Bruening's lab, what is the Reynolds number at the outer edge?

   Note: you can actually get vortices at a lower Reynolds number than the transition at 500, but their effect on mass transfer would be small depending on the relative thickness of the boundary layer and the concentration polarization layer.

   (A) 148
   (B) 14.8
   (C) 724 (e.g., it's turbulent!)
   (D) 67

4. It is observed that in a rotating membrane system the excess concentration in the concentration polarization layer at the surface is too high, thus it is proposed to decrease the transmembrane pressure by a factor of two. What would this do to the excess concentration at the surface?

   (A) decrease it by 8x.
   (B) decrease it by 4x.
   (C) increase it by 2x.
   (D) increase it by 8x.

Lecture 25: Cross-Flow Filtration: From Desalination to Hemodialysis

1. What statements below are true for current seawater desalination processes?

   (A) Membrane losses (e.g., low K_H) dominate energy costs.

   (B) If your recovery ratio is 1/2, the minimum feed pressure for a single pass SWRO is 54 atm plus losses.

   (C) Increasing K_H may actually increase energy costs by increasing concentration polarization.

   (D) The feed spacer is designed to minimize concentration polarization by increasing k_m.

2. Which of the statements below are true for general reverse osmosis systems?

   (A) K_H is more significant for treating river water than seawater.

   (B) The minimum pressure required for treating river water with an RO system is comparable to that for treating seawater.

   (C) RO systems can be used to desalinate wastewater at a (relatively) low energy cost as compared to seawater.

   (D) The energy cost for treating river water with an RO system is roughly 4 times the osmotic pressure.

3. In order for dialysis to be effective, molecules have to be able to diffuse across the lumen to the membrane surface. For the hollow fiber module specifications given in the notes (and ignoring the eigenvalue which would help a lot!) about what diffusion coefficient would be necessary for a molecule to diffuse to the wall during the time the blood is resident in the unit?

   (A) 5×10^{-6} cm^2/s

   (B) 5×10^{-7} cm^2/s

   (C) 5×10^{-8} cm^2/s

   (D) 5×10^{-9} cm^2/s

4. Even small proteins have a diffusivity in plasma far less than the correct answer to question 3. Can hemodialysis work for such molecules?

   (A) Yes, because their diffusivity is augmented by shear induced diffusion produced by the RBCs in the shear flow.

   (B) No – the scaling says that there's no hope.

Lecture 26: Elementary Electrostatics in Aqueous Media

1. For two charged spheres in a vacuum, how does the force between them vary with distance?

   (A) $1/r^2$

   (B) $1/r^3$

   (C) $1/r$

   (D) r^2

2. Two plates in air separated by a distance d have a measured capacitance of 1 mF (milliFarad). What would be the capacitance if the gap were filled with deionized water?

   (A) 80 mF

   (B) 1 mF

   (C) 12.5 μF

   (D) There wouldn't be any capacitance if DI water is in the gap — you need a conductor to make it work.

3. A porous membrane with 10 nm diameter pores is functionalized with charged groups to provide ion selectivity. It is found that the membrane is selective at an (NaCl) ionic strength of 1 mM. At what salt concentration would this selectivity be expected to degrade?

   (A) 10 mM

   (B) 1 mM

   (C) 0.1 mM

   (D) 100 mM

4. What is the Debye length in a 1 mM solution of Copper(II) Sulfate?

   (A) 5 nm

   (B) 10 nm

   (C) 3 nm

   (D) 1 nm

Lecture 27: Electrokinetics: Electroosmosis and Electrophoresis

1. Suppose you have a 25 μm diameter capillary with surface potential of 50 mV filled with an aqueous solution. The capillary is 10 cm long, and you apply a voltage drop of 500 V across the length. How long would it take for the EO velocity to drive the fluid the length of the channel?

   (A) about 10 minutes

   (B) about 10 seconds

   (C) about 10 hours

   (D) 0.35 days (EO velocity is slow!)

2. Now consider the same capillary, but the flow is driven by a pressure drop of 1 psi. What would be the convection time in this case?

   (A) 74 s

   (B) 1.2 hr

   (C) 2.8 s

   (D) 5.3 min

3. In a capillary electrophoresis setup the solute to be analyzed is often loaded in a buffer with a lower salt concentration than the rest of the capillary. Why?

   (A) Because the local salt concentration is lower, the local electric field will be higher and the solutes will focus to the front or back end of the initially loaded slug.

   (B) Because the local salt concentration is lower, the electrophoretic mobility of the analyte will be higher and will focus to the front or back end of the initially loaded slug.

   (C) There is no effect of local salt concentration on the initial focusing of the analyte.

   (D) I have no idea what the answer is, but I'll make a guess...

4. How can you reduce the "racetrack effect" in electroosmotic or electrophoretic flow through curved large aspect ratio micro channels?

   (A) Adjust the profile around the inner edge of the curve so that it increases the path length and reduces the electric field strength.

   (B) Adjust the profile around the inner edge of the curve so that it decreases the path length and reduces the electric field strength.

   (C) Adjust the profile around the inner edge of the curve so that it increases the path length and increases the electric field strength.

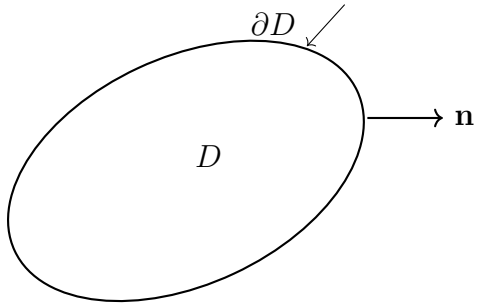
   (D) There is no way to reduce dispersion from this mechanism.

My Problems

This section is a placeholder for problems which you may wish to add to the Compendium. You would add either problems alone, or problems and a summary page using the input command. The files would need to have a toc entry at the subsection or subsubsection level depending on the file structure you wish if they are to be included in the table of contents. A placeholder problem is added in below.

This is a Placeholder for Your Problem

Verbiage here.



a. First part to be asked.

b. Second part to be asked.

c. Third part to be asked.