AguaClara Textbook Documentation

Release 0.0.8

AguaClara Cornell

Jul 21, 2018

ACKNOWLEDGEMENTS

This textbook is written and maintained in Github via Sphinx. It uses and refers to AguaClara code and functions in aide_design. Listed below are the versions of the programs we use:

Table 1: These are the software versions used to compile this textbook

Software	version
Sphinx	1.7.5
aide_design	0.0.12
Anaconda	4.5.4
Python	3.6.5

CHAPTER

ONE

ACKNOWLEDGEMENTS

We gratefully acknowledge the funding provided by the Environmental Protection Agency and the National Science Foundation. Together they have provided over $1 million in support of developing the next generation of sustainable drinking water treatment technologies.

1.1 Environmental Protection Agency statement

This textbook was developed under numerous Assistance Agreements awarded by the U.S. Environmental Protection Agency to Cornell University. It has not been formally reviewed by EPA. The views expressed in this document are solely those of the authors and do not necessarily reflect those of the Agency. EPA does not endorse any products or commercial services mentioned in this publication.

1.2 National Science Foundation statement

This material is based upon work supported by the National Science Foundation under Grant numbers CBET-1704472 and CBET-1437961. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the authors and do not necessarily reflect the views of the National Science Foundation.

Table 1.1: Table of funded research projects that contributed to the knowledge in this textbook.

Agency	Proposal Title
NSF	Wrf: Experimental Observation and Modeling of Coagulant Mediated Contaminant Removal: Flocculation, Floc Blankets, and Sedimentation
USEPA	AguaClaras Ram Pump for Zero Electricity Drinking Water Treatment
USEPA	Environment & Community Friendly Wastewater Treatment
USEPA	High Rate Sedimentation
USEPA	Novel Reactor Design for Enhanced Removal of Fluoride Using A Modified Nalgonda Method
USEPA	Novel Reactor Design for Enhanced Removal of Fluoride Using A Modified Nalgonda Method
NSF	Experimental Evaluation And Modeling Of Hydraulic Flocculation Systems Under Conditions of Turbulent Flow
USEPA	Application of Foam Filtration to Water Treatment for Rapid Emergency Response
USEPA	Stacked Rapid Sand Filtration - A Robust Filtration Process for Sustainable Drinking Water
USEPA	Sustainable Water Treatment Facility for Communities with Arsenic Contaminated Groundwater
USEPA	Smart Turbidimeters for Remote Monitoring of Water Quality
USEPA	Stacked Rapid Sand Filtration - A Robust Filtration Process for Sustainable Drinking Water Infrastructure
USEPA	Developing A Point-of-Use Filter Utilizing Polyurethane Foam
USEPA	Dose Controller for AguaClara Water Treatment Plants
USEPA	Dose Controller for AguaClara Water Treatment Plants
USEPA	AguaClara: Clean Water for Small Communities

More gratitude below! Ken Brown x 100 students who gave their time and creativity so that others could have safe water on tap.

CHAPTER

TWO

AUTHORS

This text is a collaborative effort involving hundreds of people. Innovation requires collisions of ideas and the AguaClara program was designed to foster global and multidisciplinary interactions between students, faculty, field engineers, plant operators, implementation partner organizations, and community members. These interactions have provided a continuous and rich source of ideas that make it clear that in a social network it is impossible for anyone to claim ownership of an idea. Thus the inventions, equations, and reactor designs that are described in this text are the product of a large, collaborative, open-source community and none of us can claim that we are the sole authors. The list of authors below have contributed directly to this text.

- Juan Guzman
- Monroe Weber-Shirk
- Clare OConner
- Ethan Keller

CHAPTER

THREE

INTRODUCTION TO RST AND SPHINX FOR TEXTBOOK CONTRIBUTORS

3.1 What is RST?

RST stands for ReStructured Text. It is the standard markup language used for documenting python packages. Sphinx is the Python package that generates an html website from RST files, and it is what we are using to generate this site. To read more about why we chose RST over markdown or Latex, read the following section, *Why RST?*.

3.1.1 Why RST?

In the beginning, we used markdown. As we tried to add different features to markdown (colored words, image sizes, citations), we were forced to use raw html and various pre-processors. With these various band-aid solutions came added complexity. Adding sections became cumbersome and awkward as it required ill-defined html. Additionally, providing site-wide style updates was prohibitively time-consuming and complex. Essentially, we were trying to pack too much functionality into markdown. In the search for an alternative, restructured text provided several advantages. Out of the box, RST supports globally-defined styles, figure numbering and referencing, Latex function rendering, image display customization and more. Furthermore, restructured text was already the language of choice for the AIDE ecosystems documentation.

3.2 Setting up RST for Development

There are two ways to *quickly* view an RST file. The first is using an Atom plugin that renders the view alongside the source code. This is a good initial test to make sure the RST is proper RST and looks *mostly* correct. However, some functionality, such as any extensions provided by Sphinx wont run in the preview. In order to see the final html that will display on the website, youll need to use the second method, running sphinx locally to fully generate the html code. Once you are satisfied with your work and want to push it to the textbook, youll need to incorporate it to the master branch. To do so, refer to *Publishing online*.

3.2.1 Installing the Atom Plugins

If you are using the Atom IDE to write RST, you can use the rst-preview-pandoc plugin to auto-generate a live RST preview within atom (much like the markdown-preview-plus preview page.) To get rst-preview working, youll need to install language-restructuredtext via atom and Pandoc via your command line (`pip install pandoc`). If everything worked, you can use `ctrl + shift + e` to toggle a display window for the live-updated RST preview.

3.2.2 Building RST Locally with Sphinx

We use Sphinx to build RST locally and remotely. Follow these steps to get Sphinx and run it locally:

1. Install Sphinx, disqus, and a sphinx visual theme using pip: `pip install sphinx --user -U` and `pip install git+https://github.com/rmk135/sphinxcontrib-disqus`.
2. Generate all the html by navigating in the command line to the source directory /Textbook and creating the build in that directory with the command line `make html`.
3. View the html generated in the /Textbook/_build directory by copying the full file path of /Textbook/_build/html/index.html and pasting it into your browser.

Note: Regarding **1.** the master branch for the package implementing disqus in sphinx is broken, which is why we use a non-standard pip/online installation. If you already have the incorrect sphinx-disqus version installed, uninstall it with `pip uninstall sphinxcontrib-disqus` before installing the functioning version.

3.2.3 Publishing Online

We use Travis to ensure this site will always contain functional builds. To publish online, you need to:

1. Always test your build by first :ref:‘ building RST locally <heading_building_rst_locally>‘, and then following the *testing online* instructions. Once you like how your build looks, follow the steps below to introduce it into the master branch.
2. Submit a pull request to master. Youll need to ask for someone else to review your work at this stage- request reviewers. Every pull request **must** be reviewed by at least one other person.
3. Travis will build the site using Sphinx, and if there arent any errors, Travis will report success to GitHub on the checks part of the pull request.
4. All your requested reviewers must now approve and comment on your commit before the merge is allowed.
5. Once the PR passes Travis and is approved by another author, feel free to merge to master.
6. **To release the master branch, (build the html, pdf, and latex, and upload the pdf to Pages) youll need to publish a GitHub**
    - Tag name: 0.1.5
    - Release title: Filtration section maintenance
    - Description: Added filter code from aide_design 0.2.6. Also updated all broken external links.
7. Travis will rebuild the site and push the html to Pages, and the PDF and LaTeX to GitHub Releases under the tag name.

Important: If your changes to the master branch arent pushing to gh-pages, then check the status of the Travis build here.

3.2.4 Testing Online

To test exactly what will be published, we have a test branch. The test branch is built by Travis and contains all the processed html that Travis produces in _build/html. This branch is populated when ANY COMMIT IS PUSHED. Meaning, the last commit to be pushed, if it passes the Travis tests, will be built and the output will be placed in the test branch. Also, if the PDF=True environment variable is triggered for a Travis build, the PDF will also be generated and placed in the test branch. To do this, use the Trigger Build option in Travis and put:

```
script:
    - PDF=True
```

The website output is viewable here.

3.2.5 Sharing Test Output

if you want to share what your latest branch developments look like without having whoever is viewing it actually have to build it, you can push a commit, and find the rawgit URL with this site by entering the URL of the git file within the test branch that youd like to share. Furthermore, if you want to point to the commit so that even if someone else pushes, the URL will still point to the code you intend it to, make sure to include the commit SHA within the rawgit URL like so: https://rawgit.com/AguaClara/Textbook/e5693e0485702b95e11d4d6bdf76d07c42fdbf99/html/index.html. That link will never change where it is pointing. To share the PDF output, follow the *testing online* instructions to build the PDF, and point to the commit with the PDF. Happy testing!

3.3 Brief Best Practices

When writing RST, there are often many ways to write the same thing. Almost always, the way with the fewest number of characters is the best way. Ideally, never copy and paste.

3.3.1 How do I write RST?

RST is friendly to learn and powerful. There are many useful cheatsheets, like this one and the next page on this site: Functionality in RST and AguaClara Convention, which is specifically for AguaClara and this textbook project. When you start writing RST, look at the cheat sheets all the time. Use `ctrl-f` all the time to find whatever you need.

Things not covered in most cheat sheets which are of critical importance:

- A document is referred to by its title, as defined between the `*****` signs at the top of the page, **NOT** the filename. So it is critical to have a title.
- Anything else youd like to add for the future

3.3.2 Example to Start From

This file is written in RST. You can start there! Just click on View page source at the top of the page.

Also, the next page contains the convention, and is where we specify all AguaClara RST best practices: Functionality in RST and AguaClara Convention. I recommend looking at the raw RST and the rendered html side by side.

3.4 Converting Markdown to RST

Ideally, use pandoc to do the conversion in the command line: `pandoc --from=markdown --to=rst --output=my_file.rst my_file.md`. Raw html will not be converted (because it is not actually markdown), and tables are converted poorly. Youll need to carefully review any page converted with pandoc.

CHAPTER

FOUR

PARAMETER CONVENTION LIST

Table 4.1: Relevant Dimensions

Dimension	Abbreviation	Base Unit
Length	$[L]$	meter
Mass	$[M]$	kilogram
Time	$[T]$	second

If you would like to be able to `ctrl+f` some variables, click on View page source on the top right of this window. If you want to know what a greek variable is but dont know what its called, you can view the source text on the file where you found the variable. nu, mu, eta, who actually remembers what these all look like? The letter v should sue nu for copyright infringement. Or is it the other way around?

Table 4.2: Parameter Guide

Parameter	Description	Units
m	Mass	$[M]$
z	Elevation	$[L]$
L	Length	$[L]$
W	Width	$[L]$
H	Height	$[L]$
D	Diameter	$[L]$
r	Radius	$[L]$
A	Area	$[L]^2$
$\rlap{-}V$	Volume	$[L]^3$
v	Velocity	$\frac{[L]}{[T]}$
Q	Flow rate	$\frac{[L]^3}{[T]}$
n	Number, Amount	Dimensionless
C	Concentration	$\frac{[M]}{[L]^3}$
p	Pressure	$\frac{[M]}{[L][T]^2}$
g	Acceleration due to Gravity	$\frac{[L]}{[T]^2}$
ρ	Density	$\frac{[M]}{[L]^3}$
μ	Dynamic viscosity	$\frac{[M]}{[T][L]}$
ν	Kinematic viscosity	$\frac{[L]^2}{[T]}$
h	Head, Elevation	$[L]$
h_L	Headloss	$[L]$
$h_{\rm f}$	Major Loss (friction)	$[L]$

Continued on next page

Table 4.2 – continued from previous page

Parameter	Description	Units
ϵ	Surface roughness	$[L]$
f	Darcy-Weisbach friction factor	Dimensionless
Re	Reynolds Number	Dimensionless
h_e	Minor Loss (expansion)	$[L]$
K	Minor Loss coefficient	Dimensionless
Π	Dimensionless Proportionality Ratio	Dimensionless
Π_{vc}	Vena Contracta Area Ratio	Dimensionless
Π_{Error}	Linearity Error Ratio	Dimensionless
M	Fluid Momentum	$\frac{[M][L]}{[T]^2}$
F	Force	$\frac{[M][L]}{[T]^2}$
t	Time	$[T]$
θ	Residence Time	$[T]$
G	Velocity Gradient/Fluid Deformation	$\frac{1}{[T]}$
ε	Energy Dissipation Rate	$\frac{[L]^2}{[T]^3}$
$\Pi_{\bar{G}}^{G_{Max}}$	$\frac{G_{Max}}{\bar{G}}$ Ratio in a Reactor	Dimensionless
$\Pi_{\bar{\varepsilon}}^{\varepsilon_{Max}}$	$\frac{\varepsilon_{Max}}{\bar{\varepsilon}}$ Ratio in a Reactor	Dimensionless
Π_{HS}	Height to Baffle Spacing in a Flocculator	Dimensionless
H_e	Height Between Flow Expansions in a Flocculator	$[L]$
S	Spacing Between Two Objects	$[L]$
B	Center-to-Center Spacing Between Two Objects	$[L]$
T	Object Thickness	$[L]$
P	Power	$\frac{[M][L]^2}{[T]^3}$
η_K	Kolmogorov Length Scale	$[L]$
λ_ν	Inner Viscous Length Scale	$[L]$
$\Pi_{K\nu}$	Ratio of Inner Viscous Length Scale to Kolmogorov Length Scale	Dimensionless
Λ	Distance Between Particles	$[L]$

CHAPTER

FIVE

INTRODUCTION TO AGUACLARA WATER TREATMENT DESIGN

5.1 A Different Kind of Textbook

This textbook represents our cumulative insights from our journey that has been motivated by a quest to make the world a better place where everyone has access to safe water on tap, the engineering challenge of optimizing the design of drinking water treatment plants, and the curiosity to understand what controls their performance. We would like to understand what determines which contaminants make it the whole way through a water treatment plant. If we could understand what allows some contaminants to sneak the whole way through a water treatment plant, then we suspect that we could create better designs to more effectively remove contaminants.

Engineering textbooks provide a venue for authors to share what theyve learned, to organize ideas, and to provide a guide for engineers as they design solutions for real world problems. Engineering textbooks are often intended to document the established core of knowledge. It seems reasonable to assume that what is in textbooks and in peer reviewed literature is mostly true.

5.1.1 The edge of knowledge may be closer than we thought

The assumption that what is written and passed down in oral history through the scientific community is true can lead to missed opportunities and lost insights. The hypotheses from one generation of scientists can too easily evolve into new theories in the next generation and then into established theories for the next. The history of drinking water treatment science is cloudy (think high turbidity!) with hypotheses that miss or misrepresent key concepts.

You might wonder why getting the science right matters. After all, the core drinking water treatment technologies were invented before we were born and many of us have safe drinking water coming from our taps. Environmental Engineers have known how to design municipal drinking water treatment plants since they early 1900s. We hypothesize that there are many opportunities to significantly improve drinking water treatment technologies and that improved understandings of each unit process have the potential to lead to new breakthroughs.

Our contention is that no one has ever optimized the design of a drinking water treatment plant! We are reasonably certain of this because we dont yet have models (with equations) that describe performance of most of the core unit processes (rapid mix, flocculation, floc blankets, sedimentation, sand filtration) used for surface water treatment. The only possible exception is lamellar sedimentation which can be characterized if we know the size and density distribution of the particles entering the sedimentation tank.

Traditional drinking water treatment textbooks can too easily miss the opportunity to advance the science of drinking water treatment technologies by presenting certainty where there should be skepticism. For example, rapid mix is described as process that occurs in a few seconds, flocculation is described as a process that should be fastest for high turbidity waters and slowest for low turbidity waters, and filtration performance is described by a model that predicts first order removal with respect to filter bed depth. We will demonstrate why each of these assumptions doesnt match observations, we will discuss new insights into these processes, and we will identify high priority research questions that have the potential to lead to major improvements in drinking water treatment.

We want to encourage skepticism and we want to develop insights to guide thoughtful skepticism. A key skill for successful engineering is the ability to identify the location of the edge of knowledge. The ability to distinguish between what is reasonably certain and what is still in question is what powers the scientific method of slowly extending knowledge. New insights are difficult to obtain if the research is based on a faulty premise.

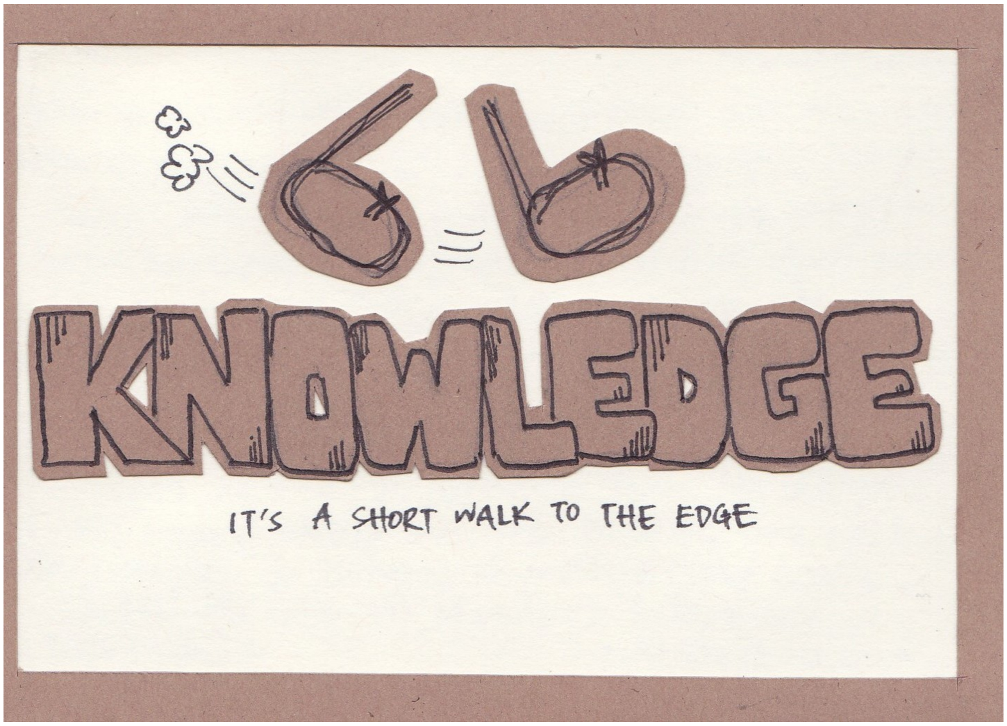


Fig. 5.1: Weve learned that we can find the edge of knowledge very soon after we begin researching a water treatment technology (artwork created by Yi Wen Ng in 2012).

There are significant knowledge gaps in every process that we cover in this textbook. We arent yet able to optimize surface water treatment processes because we dont yet understand the fundamental physics of many of the processes. We are getting closer, join us on the journey.

We need the brightest and the best to create new and better solutions so we can meet the goal of providing everyone with safe drinking water. This challenge is apparently more difficult than building a space station, designing a fuel cell, or inventing the world wide web. So lets role up our sleeves and begin.

5.1.2 Tools to Find the Edge of Knowledge

- Dont believe everything we say
- Ask lots of questions
- How do you know that? The goal here is to identify the difference between what is known and what is hypothesized.
- What is the equation that describes the physics of this process? If there isnt an equation the describes the process and that can be used to design the reactor for the process, then it is likely that the equation doesnt yet capture the physics.
- How could we improve this process? If the physics of a process are fully understood, then dimensionally correct equations can be used to obtain the optimal design for that process.

- Is the process design based on rules of thumb or on physics? Rules of thumb or empirical design guidelines often can be identified by the use of physical parameters that have units. For example, if the design guideline specifies a length, time, or velocity then it is likely that the guideline is not based on physics. If the design guidelines are based on a dimensionless parameter then it is possible that it is based on physics.
- Evaluate the data to see if it matches predictions of the hypothesized model. Assess whether the authors acknowledge when their data doesnt match hypothesized models.
- Beware of the use of words that are poorly defined and that hide uncertainty. For example, creating a name for a supposed mechanism to describe all of the observations that dont fit with your theory does NOT mean that you understand that mechanism. The ability to name something doesnt mean it is understood.
- Does this theory provide insights that have led to new discoveries or new applications?
- Does the theory include equations that are based on the fundamental laws of nature?
- Does the theory use dimensionless constants that are close to one?
- Is it an elegant theory with no need for special cases?

5.1.3 Myth in Environmental Engineering

The following list is designed to get you thinking. These are concepts that present in the Environmental Engineering community

- Dead bodies cause disease
- Slow sand filters ripen (improve in ability to remove contaminants over time) because of biological growth in the filter bed
- If a 20 cm deep sand filter removes 90% of influent particles, then a 40 cm deep filter will remove 99% of influent particles
- If water is dirty, then you should filter it.
- Chlorine disinfects dirty water and makes it safe to drink
- Chlorination and filtration eliminated typhoid fever from the US
- Cessation of chlorination due to fear of disinfection by products caused the cholera outbreak in Peru in 1993
- Sedimentation is simple
- We already know how to solve the problem of the billions of people who do not having access to safe drinking water

A challenge for authors is to recognize the difference between what is known with a reasonably high degree of certainty and what is assumed to be true without a solid basis. We struggle to tell the difference between fact and hypothesis. The time-honored approach in science is to rely on the peer review process. But that process for vetting knowledge has been shown to be flawed.

Your question could be whether the distinction between fact and hypothesis really matters. If the hypothesis is widely accepted as fact and if it has been accepted for decades what benefit is there to calling it a hypothesis rather than a fact?

This question is at the core of our educational philosophy. Is this text the repository of knowledge that we are providing for you to drink or is this text a conversation where we invite you to join the effort to discover better ways to provide safe water on tap?

5.1.4 Integrating Educational Philosophy with an Evolving Textbook

This is an evolving textbook. We dont intend to ever print this book. This book has version numbers just like software with the idea that revisions are rapid and frequent. We commit to helping to accelerate the pace of knowledge generation and to revising this text as you help us identify places where we have presented hypotheses as theory and places where research provides a basis for better theoretical models of the water treatment processes.

Socrates said Education is the kindling of a flame, not the filling of a vessel. Our goal is to bring the spirit of play, discovery, and mystery into the challenge of improving the quality of life of everyone on the planet by sharing better methods to produce safe drinking water.

In We Make the Road by Walking: Conversations on Education and Social Change, Paulo Freire said, The more we become able to become a child again, to keep ourselves childlike, the more we can understand that because we love the world and we are open to understanding, to comprehension, that when we kill the child in us, we are no longer.

We commit to playing together in a relationship where we are all learning and we are all teaching. Education must begin with the solution of the teacher-student contradiction, by reconciling the poles of the contradiction so that both are simultaneously teachers and students. - Paulo Freire

5.1.5 Respect, Empathy, Love and Curiosity power the AguaClara Innovation System

The AguaClara network of organizations has been methodically inventing improved water treatment technologies since 2005. Our success is based on respect, empathy and love. Innovation requires flocculation of ideas. The transport of ideas between organizations and individuals is mediated by respect. Respect as a cornerstone of organizational culture foster rapid and honest exchange of ideas.

Any large organization will require a leadership hierarchy and any hierarchy will rely on respect based on fear or respect based

- Theyll Teach You, Whether You Like It or Not
- Everyone is a Friend or a Foe
- Its All about the Trophies
- They Dont Step Outside Boxes
- Theyre Addicted to Yardsticks

Love-based hierarchies foster honesty and a free-flow of information. Reflection is encouraged across the organization and truth, honesty, and integrity are valued. Staff at the bottom of the hierarchy know that their opinions and reflections are valued and thus they will be willing to report problems to organization leaders.

Love-based leaders relate to others based on true respect for the other. They will take the time to converse with people at all levels of the organization and will value the opportunity to speak with people who are the interface between the organization and the rest of the world. A persons value is based on being a person, not based on position in the hierarchy.

As water treatment plant designers it is critical that we spend time with a diverse set of stakeholders including community members and water treatment plant operators. Those relationships must begin with respect and valuing their insights. As we spend time together we can develop trust so that they communicate both the good and bad.

Weve learned much from plant operators. They figured out how to reduce rising flocs at Agalteca, Honduras where we learned that conventional sedimentation tank inlet manifolds generate large circulation currents. Plant operators added curtains to the windows at Moroceli, Honduras because they noticed that direct sunlight on the sedimentation tanks caused an increase in settled water turbidity.

Empathy enables us to consider reality from anothers perspective. Empathy is fundamental in design. Empathy enables us to bring the people who will use or benefit from a technology into the design considerations. Empathy brings the insight that water treatment plants need to have roofs and provide a secure work environment both day and night. Empathy brings the insight that replacement parts must be readily available and that generic components are preferred over specialty proprietary components.

Curiosity can flourish in a culture of love, respect, and empathy. Asking why and why not and pondering an ever growing number of questions has empowered student teams to take on the quest for new knowledge and new solutions.

5.2 The Global Context for Drinking Water Treatment

The Sustainable Development Goals: SDGs and specifically SDG 6 provide the context and motivation for this text. The first SDG 6 target is: By 2030, achieve universal and equitable access to safe and affordable drinking water for all. That goal is daunting and wont be met using the approaches of the past 5 decades. This text is about creating a new paradigm for the design of high performing water treatment technologies with the goal of making a real contribution toward SDG 6.1.

Fig. 5.2: Sustainable development goal 6 is all about clean water and sanitation.

The number of people who currently lack access to reliable safe water on tap is not known. Estimates range from 1.8 billion who use a source of drinking water that is fecally contaminated to the Centers for Disease Control recommendations for where it is usually safe to drink tap water.

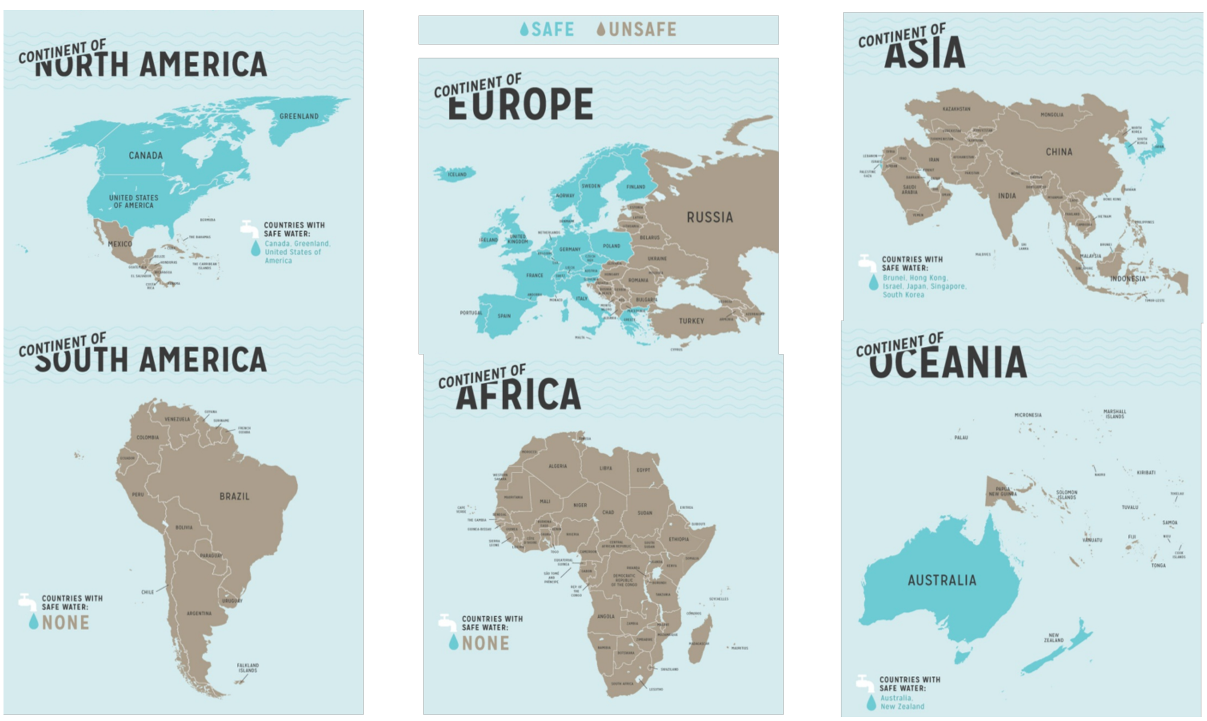


Fig. 5.3: There are relatively few countries where it is almost always safe to drink the tap water.

The UN estimate in 2017 was that 2.1 billion lack access to safe water. By 2030 there will be an additional 1.2 billion from population growth.

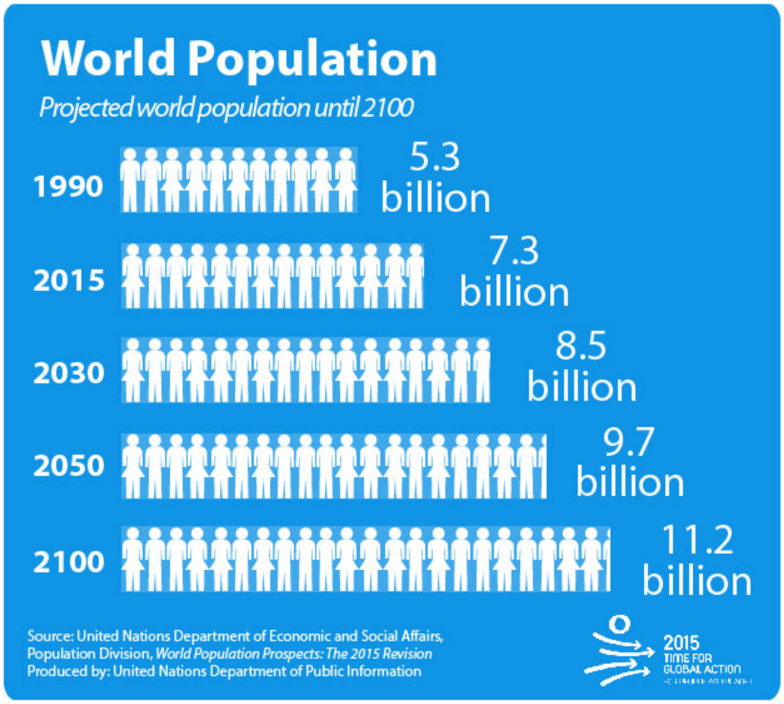


Fig. 5.4: 1.2 billion people will be added to the global population between 2015 and 2030.

Thus by 2030 we need to provide safe water for at least 3.3 billion people AND maintain the water supply systems for the 5.2 billion who currently have access to safe water. That is a daunting number that requires some exploration!

```
from aide_design.play import*
import datetime
People_needing_water_2030 = 3.3*10**9
now = datetime.datetime.now()
Task_time = (2030 - now.year)*u.year
#If we assume we will meet this demand by building the same amount of new capacity␣
↪each year, then we have
People_per_year = People_needing_water_2030/Task_time
People_per_year
#The percapita demand for water
Per_capita_demand = 3*u.mL/u.s
Per_capita_demand.to(u.L/u.day)
Per_capita_demand
Rate_new_water_supply_capacity = (People_per_year * Per_capita_demand).to(u.L/(u.s*u.
↪year))
Rate_new_water_supply_capacity
NYC_water_supply = 44000 * u.L/u.s
NYC_per_year = Rate_new_water_supply_capacity/NYC_water_supply
NYC_per_year
```

If we provide 260 L/day per person, then we need to provide the equivalent of 19 water supplies for New York City every year between now and 2030. The planet needs approximately 800,000 L/s of capacity each year. AguaClara water treatment plants cost approximately $10,000 per L/s of treatment capacity. Thus the budget for global water treatment needs to be 8 billion USD per year. Note that this doesnt include any other aspects of supplying water. Managing water sources, transmission lines, storage, and distribution systems are even more expensive than water treatment.

The need for drinking water supplies isnt limited to the global south. The California Urban Water Agencies estimate that 530,000 or more people in rural areas of California are unable to turn on their tap and access clean, safe water.

5.2.1 Why dont 2 billion people have access to safe water?

The simple answer is that they are too poor and are unable to afford safe water on tap. But it isnt that simple! Families without access to safe water on tap often spend more for water than families with safe water on tap.

As we work to solve a global challenge that has been plaguing humanity since the dawn of human civilization, then it will serve us well to understand a bit of the history that has led to our current reality. Water treatment history includes amazing successes, persistent failures, fortuitous discoveries, a heavy reliance on empiricism, and an occasional myth. Our goal is learn from and reflect on our history and then create even better solutions.

5.3 Introduction to Surface Water Treatment

Table 5.1: History of Surface Water Treatment Technologies

Technology	Description	Prerequisite	Owner	Year
Simple sedimentation	particles settle	none	public	unknown
Flocculation	aluminum and iron salts	none	public	1757
Sedimentation	horizontal flow	flocculation	public	unknown
Lamellar sedimentation	plate or tube settlers	flocculation or floc blanket	public	1904
Slow sand filtration	single step treatment for low NTU water	none	public	1829
Rapid sand filtration	depth filtration	sedimentation	public	1920
Stacked rapid sand filter	gravity powered backwash	lamellar sedimentation	AguaClara Cornell open source	2012
Floc blanket	upflow fluidized suspension of flocs	flocculation	public	1930
Jet reverser floc blanket	first fully fluidized floc blanket	flocculation	AguaClara Cornell open source	2012
Ballasted sedimentation	•	•	Actiflo Veolia	1995
Superpulsator	pulsing flow through floc blanket	rapid mix	Degremont	1958 1991
Dissolved air flotation	•	•	•	•

See Pretreatment Processes for Potable Water Treatment Plants by Jeff Lindgren for an excellent overview of available technologies, May 2014 (not including AguaClara innovations).

5.3.1 Treatment Trains

The prerequisites for the unit processes in the table above reveal that surface water treatment almost always requires a series of treatment steps. A treatment train is a series of treatment steps (or unit processes) designed to convert a contaminated source water into a purified water meeting the design objectives.

The AguaClara Treatment Train Why does flocculation precedes sedimentation? Which process removes the largest quantity of contaminants?

Sedimentation is the process of particles falling because they have a higher density then the water, and its governing equation is:

$$\bar{v}_t = \frac{D_{particle}^2 g}{18\nu}\frac{\rho_p - \rho_w}{\rho_w} \tag{5.1}$$

Such that:
$\bar{v}_t$ = terminal velocity of a particle, its downwards speed if it were in quiescent (still) water
$D_{particle}$ = particle diameter
ρ = density. The p subscript stands for particle, while w stands for water

todo add graph showing sed velocity of organic and inorganic particles as function of diameter.

5.3.2 Design Evolution

During the later half of the 20th century surface water treatment technologies evolved slowly. The slow evolution was likely a product of the regulatory environment, the high cost of water treatment infrastructure, and the low profit margin. The high cost of municipal scale water treatment infrastructure made experiments at scale infeasible and thus there was no mechanism to introduce disruptive innovations. With little opportunity for a significant return on investment there was little incentive to invest in the research and development that could have advanced the technologies. A final disincentive was the widely held belief that surface water treatment was a mature field with little opportunity for significant advancement. The advances of the latter half of the 20th century focused primarily on mechanization and automation (Supervisory Control and Data Acquisition - SCADA).

Design standards such as the [Great Lakes - Upper Mississippi River Board 10 States Standards](http://10statesstandards.com/) are evolving very slowly and retain an empirical approach to design. The empirical design methodology is a direct result of two confounding factors. The physics of particle interactions based on diffusion, fluid shear, and gravity are complex and given the challenges of characterizing surface water particle suspensions it was natural to assume that a mathematical description of the processes would be intractable.

Mechanized and automated water treatment plants performed reasonably well in communities with ready access to technical support services and supply chains that could reliably deliver replacement parts. In the global south municipal water treatment plants havent faired as well. In 2012, one of the main water treatment plants serving Kathmandu, Nepal had failed chlorine pumps and were using a red garden hose to siphon chlorine from the stock tank. They crimped the end of the hose to control the flow rate of the chlorine solution.

The ingenious and simple chemical dosing system that uses a siphon to completely bypass the failed pumps begs the question of whether design engineers could have invented a better option than the short lived pumps that they specified. We will investigate a gravity powered chemical dosing system that is far more reliable than chemical dosing pumps and that borrows from the simplicity of the garden hose solution used by the Nepali plant operators.

Chemical dosing systems are particularly vulnerable and their failures make plant operation very challenging. Providing the right coagulant dose is critical for efficient removal of particle and dissolved organics. Chemical dosing systems commonly rely on pumps and those pumps require regular maintenance and have relatively short mean times between failures.

The AguaClara Cornell program was founded in 2005 with the goal of creating a new generation of sustainable technologies that would perform well even in the rugged settings of rural communities. The goal wasnt simply to create technologies that would work for communities with very limited resources. The goal was to create the next generation of technologies that would both perform well in communities with limited resources and would be the highest performing technologies on multiple metrics for all communities.

Fig. 5.5: Failed chlorine doing system bypassed with a red tube that siphons the chlorine solution at a plant in Kathmandu, Nepal in 2012.

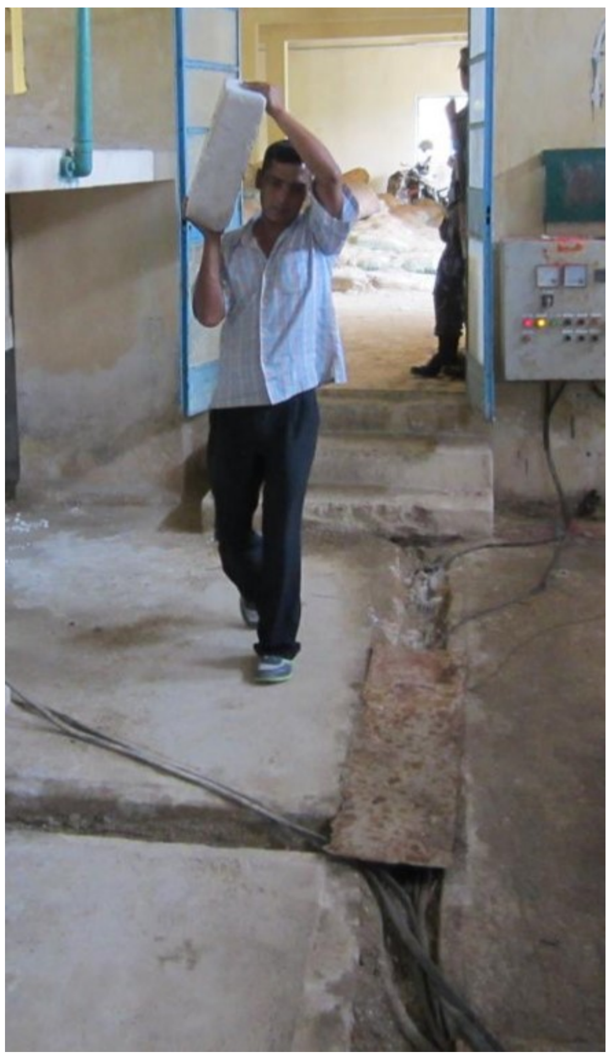


Fig. 5.6: Alum dosing system based on the rate that 25 kg blocks of alum are placed in the inlet channel of the plant.

5.3.3 Empirical Design

For the past several decades surface water treatment technologies have been considered mature and when I (Monroe) took a design course on drinking water treatment in 1985 I had the impression that there was little room for further innovation. This perspective is remarkable given that with the exception of lamellar sedimentation there were no equations describing the core treatment processes.

Empirical design guidelines dont provide insight into how designs could be optimized or even what the performance of a water treatment plant will be. Thus designers cant compare alternative designs because there is no way to compare designs that would deliver comparable results.

5.4 Design for the Financers, Venders, Client, or Context

Tours of water treatment plants suggest that it is common for designs to be driven by the vender goal of a stable revenue stream for replacement parts rather than by a goal of meeting the clients needs. Mandatory software upgrades, mechanical valves, chemical pumps, mixing units provide a steady demand for proprietary components. Financers often prefer projects that can be implemented quickly either because they have target expenditures for a fiscal year or because loan repayment begins when the facility is turned over to the client.

Design for the client would strive to reduce capital, operating, and maintenance expenses. Clients also place a high value on reliability, ease of maintenance, and the ability to handle repairs with their staff. Design for the context would account for the capabilities of local and national supply chains. A key design consideration is to ensure that the treatment capabilities of the treatment plant match the variable water quality of the proposed water source. There are numerous slow sand filtration plants installed in the global south that are attempting to treat water sources that can not be effectively treated by slow sand filtration. The cost of the failure to consider the client and the context is born by the communities who end up with water treatment systems that arent able to provide reliable safe water.

Design for the client requires empathy and a commitment to listen to and learn from plant operators. It also requires attention to detail and watching how plant operators interact with water treatment plants. Empathy leads to the goal of creating a work environment that makes it easy for the plant operators to do their routine tasks. This isnt just to make the plant operators work easy. A plant that is designed with the plant operator in mind will also engender pride and that pride will lead to better plant performance.

Fig. 5.7: A plant operator built a makeshift ladder to enable easier access to the flocculation and sedimentation tanks in a package plant. This ladder considerably shortened the distance between the coagulant dose controls and the observation of flocculation.

5.4.1 Design Bifurcations

Seemingly small decisions can have a profound effect on the evolution of design. Traditionally in tropical and temperate climates, flocculation and sedimentation units are built without an enclosing building. Without protection from the sun the materials used for plant construction must be UV resistant and thus plastic cant be used. This requires use of heavier and more expensive materials such stainless steel and aluminum. Metal plate settlers are heavy and thus they cant be easily removed by the plant operator.

Conventional sedimentation tank cleaning must be done by providing operator access below the plate settlers. This in turn requires that the space below the plate settlers be tall enough to accommodate a plant operator. Thus sedimentation tanks that are built in the open have to be deeper than sedimentation tanks that are built under a roof and they are more difficult to maintain because the operator has to enter the tank through a waterproof access port.

Fig. 5.8: Plant operators opening hatch below plate settlers in a traditional sedimentation tank.

Fig. 5.9: Plant operator removing plate settlers from an AguaClara sedimentation tank settlers.

Dramatically different designs are also created when we choose gravity power and smart fluids rather than electricity for each of the unit processes.

5.5 Water Contaminants

Particles transported by rivers are composed of resistant primary minerals (e.g., quartz and zircon), secondary minerals (clays, metallic oxides and oxyhydroxides) and biogenic remains.

5.5.1 Turbidity

Define Turbidity Show optical systems explain rough correlation with suspended solids concentration and why that relationship varies depending on the particle size distribution and density.

5.5.2 Pathogens

Pathogens are particles

5.5.3 Dissolved Species

Natural Organic Matter (NOM) plays a supersized role in influencing performance of surface water treatment plants. Discuss charge density. Observed stabilization of inorganic particles

5.6 Chlorine Saved the World

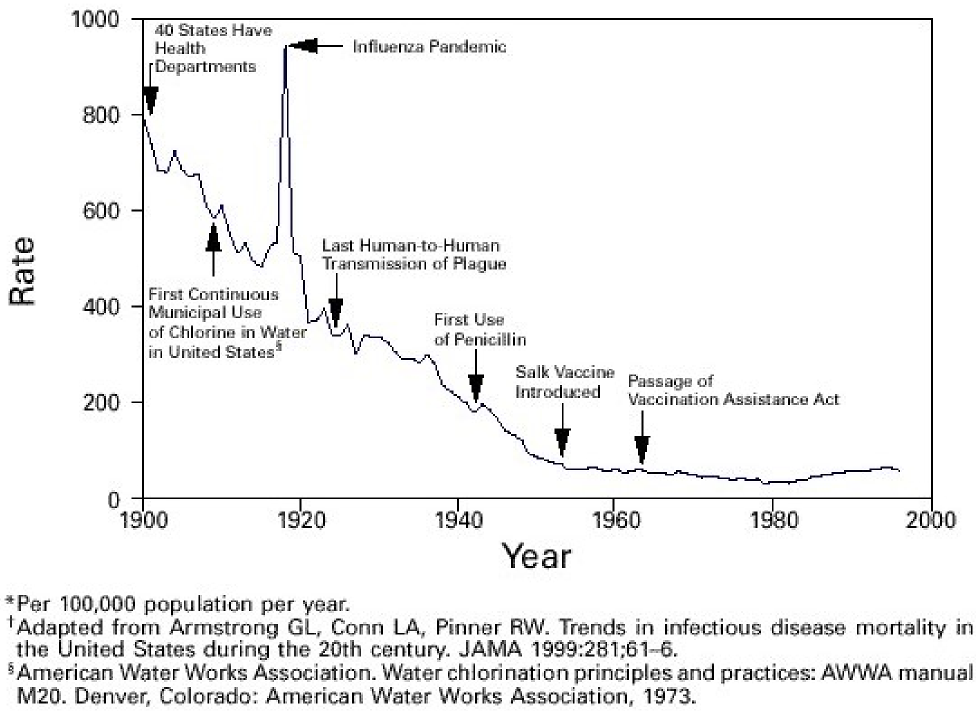


Fig. 5.10: Classic graph showing the reduction in the death rate for the United States from 1900 to 1996.

CHAPTER

SIX

REVIEW: FLUID MECHANICS

This document is meant to be a refresher on fluid mechanics. It will only cover the topics of fluids mechanics that will be used heavily in the course.

If you wish to review fluid mechanics in (much) more detail, please refer to this guide Note that to view this link, you will need a Github accounts. If you wish to review from a legitimate textbook, you can find a pdf of good book by Frank White here.

6.1 Important Terms and Equations

Terms:

1. *Laminar*
2. *Turbulent*
3. *Viscosity*
4. *Streamline*
5. *Control Volume*
6. *Head*
7. *Head loss*
8. *Driving head*
9. *Vena Contracta/Coefficient of Contraction*

Equations:

1. Continuity equation: (6.1)
2. Reynolds number (6.6)
3. Bernoulli equation (6.7)
4. Energy equation (6.9)
5. Darcy-Weisbach equation (6.10)
6. Swamee-Jain equation (6.13)
7. Hagen-Poiseuille equation (6.17)
8. Orifice equation (6.34)

6.2 Introductory Concepts

Before diving in to the rest of this document, there are a few important concepts to focus on which will be the foundation for building your understanding of fluid mechanics. One must walk before they can run, and similarly, the basics of fluid mechanics must be understood before moving on to the more fun (and exciting!) sections of this document.

6.2.1 Continuity Equation

Continuity is simply an application of mass balance to fluid mechanics. It states that the cross sectional area A that a fluid flows through multiplied by the fluids average flow velocity $\bar{v}$ must equal the fluids flow rate Q:

$$Q = \bar{v}A \tag{6.1}$$

Note: The line above the v is called a bar, and represents an average. Any variable can have a bar. In this case, we are adding the bar to velocity v, turning it into average velocity $\bar{v}$. This variable is pronounced v bar.

In this course, we deal primarily with flow through pipes. For a circular pipe, $A = \pi r^2$. Substituting diameter in for radius, $r = \frac{D}{2}$, we get $A = \frac{\pi D^2}{4}$. You will often see this form of the continuity equation being used to relate the a pipes flow rate to its diameter and the velocity of the fluid flowing through it:

$$Q = \bar{v}\frac{\pi D^2}{4} \tag{6.2}$$

The continuity equation is also useful when flow is going from one geometry to another. In this case, the flow in one geometry must be the same as the flow in the other, $Q_1 = Q_2$, which yields the following equations:

$$\bar{v}_1 A_1 = \bar{v}_2 A_2 \tag{6.3}$$

$$\bar{v}_1 \frac{\pi D_1^2}{4} = \bar{v}_2 \frac{\pi D_2^2}{4} \tag{6.4}$$

Such that:
$Q =$ fluid flow rate
$\bar{v} =$ fluid average velocity
$A =$ pipe area
$r =$ pipe radius
$D =$ pipe diameter

An example of changing flow geometries is when a change in pipe size occurs in a circular piping system, as is demonstrated below. The flow through $\text{pipe}\,1$ must be the same as the flow through $\text{pipe}\,2$.

6.2.2 Laminar and Turbulent Flow

Considering that this class deals with the flow of water through a water treatment plant, understanding the characteristics of the flow is very important. Thus, it is necessary to understand the most common characteristic of fluid flow: whether it is **laminar** or **turbulent**. Laminar flow is very smooth and highly ordered. Turbulent flow is chaotic, messy, and disordered. The best way to understand each flow and what it looks like is visually, like in the wikipedia figure below or in this video. Please ignore the part of the video after the image of the tap.

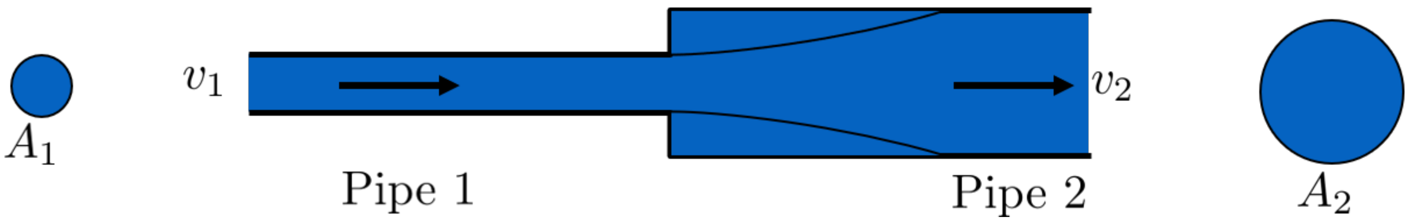


Fig. 6.1: Flow going from a small diameter pipe to a large one. The continuity principle states that the flow through each pipe must be the same.

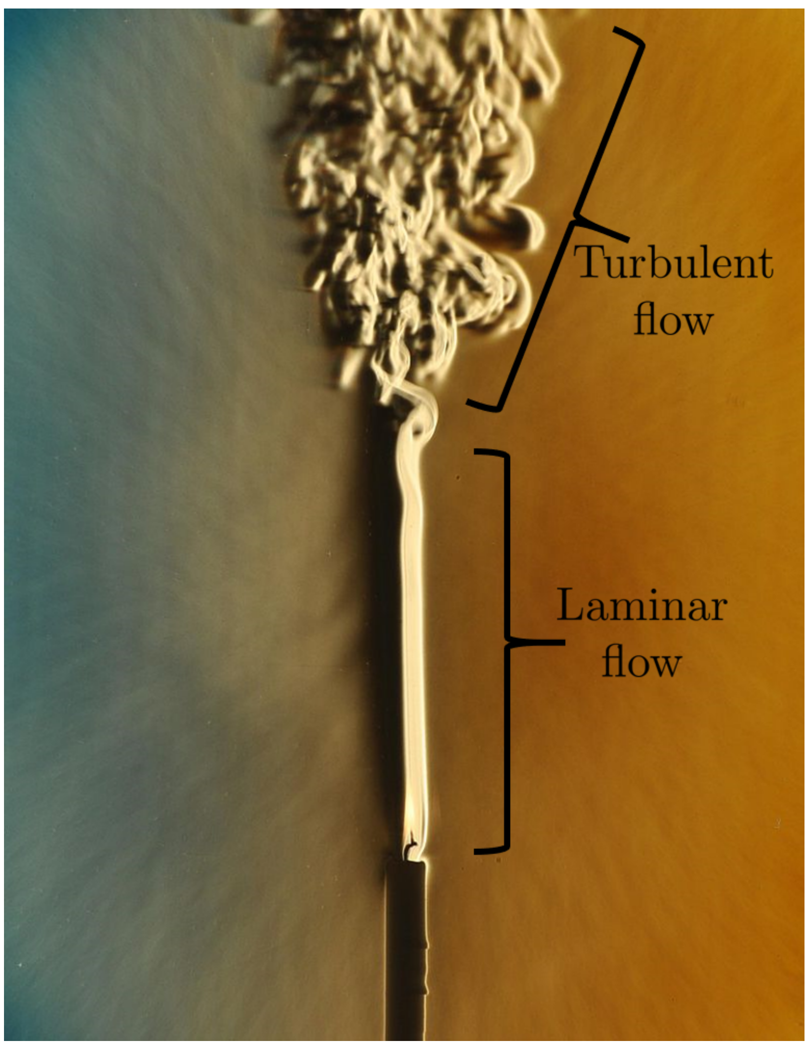


Fig. 6.2: This is a beautiful example of the difference between ordered and smooth laminar flow and chaotic turbulent flow.

A numeric way to determine whether flow is laminar or turbulent is by finding the Reynolds number, Re. The Reynolds number is a dimensionless parameter that compares inertia, represented by the average flow velocity $\bar{v}$ times a length scale D to viscosity, represented by the kinematic viscosity ν. Click here for a brief video explanation of viscosity. If the Reynolds number is less than 2,100 the flow is considered laminar. If it is more than 2,100, it is considered turbulent.

$$\mathrm{Re} = \frac{\text{inertia}}{\text{viscosity}} = \frac{\bar{v}D}{\nu} \tag{6.5}$$

The transition between laminar and turbulent flow is not yet well understood, which is why the concept of transitional flow is often simplified and neglected to make it possible to code for laminar or turbulent flow, which are better understood. We will assume that the transition occurs at $\mathrm{Re} = 2100$. In aide_design, this parameter shows us as `pc.RE_TRANSITION_PIPE`.

Fluid can flow through very many different geometries, like a pipe, a rectangular channel, or any other shape. To account for this, the characteristic length scale for the Reynolds number, which was written in the equation above as D, is quantified as the hydraulic diameter, D_h when considering a general cross-sectional area. For circular pipes, which are the most common geometry youll encounter in this class, the hydraulic diameter is simply the pipes diameter, $D_h = D$.

Here are other commonly used forms of the Reynolds number equation *for circular pipes*. They are the same as the one above, just with the substitutions $Q = \bar{v}\frac{\pi D^2}{4}$ and $\nu = \frac{\mu}{\rho}$

$$\mathrm{Re} = \frac{\bar{v}D}{\nu} = \frac{4Q}{\pi D \nu} = \frac{\rho \bar{v} D}{\mu} \tag{6.6}$$

Such that:
Q = fluid flow rate in pipe
D = pipe diameter
$\bar{v}$ = fluid velocity
ν = fluid kinematic viscosity
μ = fluid dynamic viscosity

See also:

Function in aide_design: `pc.re_pipe(FlowRate, Diam, Nu)` Returns the Reynolds number *in a circular pipe*. Functions for finding the Reynolds number through other flow conduits and geometries can also be found in physchem.py within aide_design.

Note: Definition of Flow Regimes: Laminar and turbulent flow are described as two different **flow regimes**. When there is a characteristic of flow and different categories of the characteristic, each category is referred to as a flow regime. For example, the Reynolds number describes a flow characteristic, and its categories, referred to as flow regimes, are laminar or turbulent.

6.2.3 Streamlines and Control Volumes

Both streamlines and control volumes are tools to compare different parts of a system. For this class, this system will always be hydraulic.

Imagine water flowing through a pipe. A streamline is the path that a particle would take if it could be placed in the fluid without changing the original flow of the fluid. A more technical definition is a line which is everywhere parallel to the local velocity vector. Computational tools, dyes (in water), or smoke (in air) can be used to visualize streamlines.

A **control volume** is just an imaginary 3-dimensional shape in space. Its boundaries can be placed anywhere by the person applying the control volume, and once set the boundaries remain fixed in space over time. These boundaries are usually chosen to compare two relevant surfaces to each other. These surfaces are called *Control Surfaces*. The entirety of a control volume is usually not shown, as it is often unnecessary. This is demonstrated in the following image:

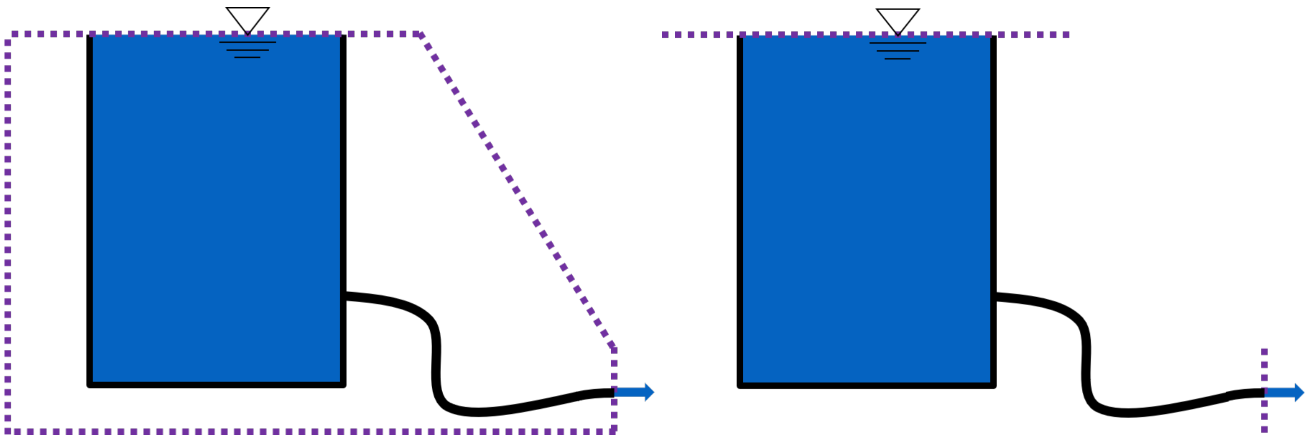


Fig. 6.3: While the image on the left indicates a complete control volume, control volumes are usually shortened to only include the relevant control surfaces, in which the control volume intersects the fluid. This is shown in the image on the right.

Important: Many images will be used over the course of this class to show hydraulic systems. A standardized system of lines will be used throughout them all to distinguish reference elevations from control volumes from streamlines. This system is described in the image below.

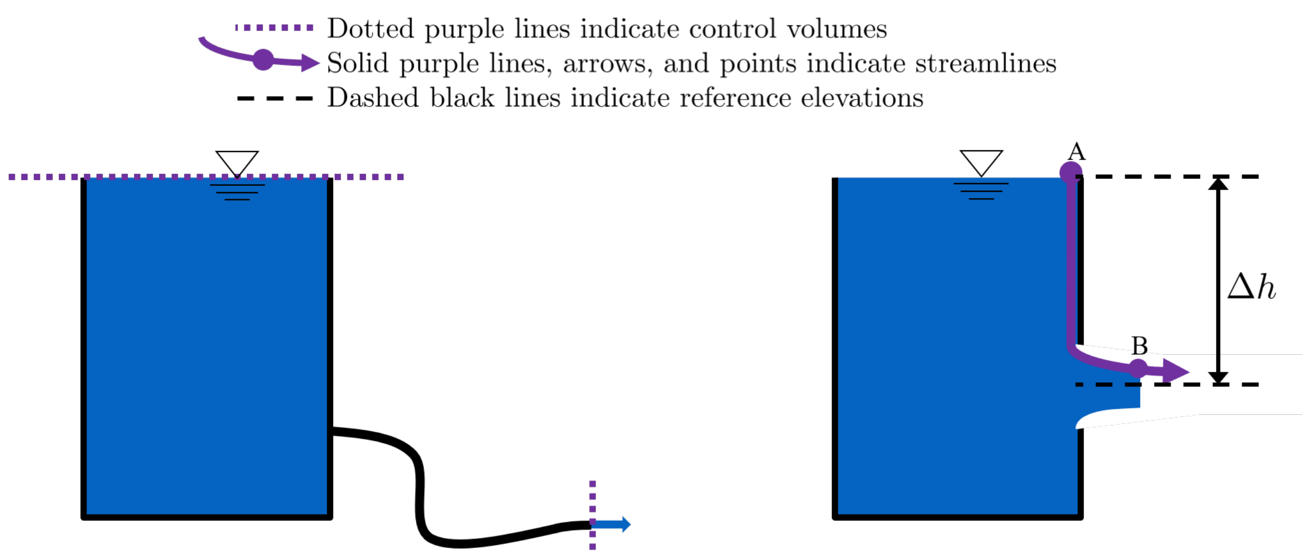


Fig. 6.4: On the right, a control volume is applied to a hydraulic system. On the left, a streamline is applied to a hydraulic system. A figure-convention for control volumes and streamlines will be very helpful throughout this course as there will be very, very many figures.

6.3 The Bernoulli and Energy Equations

As explained in almost every fluid mechanics class, the Bernoulli and energy equations are incredibly useful in understanding the transfer of the fluids energy throughout a streamline or through a control volume. The Bernoulli equation applies to two different points along one streamline, whereas the energy equation applies to fluid entering and exiting a control volume. The energy of a fluid has three forms: pressure, potential (deriving from elevation), and kinetic (deriving from velocity).

6.3.1 The Bernoulli Equation

These three forms of energy expressed above make up the Bernoulli equation:

$$\frac{p_1}{\rho g} + z_1 + \frac{v_1^2}{2g} = \frac{p_2}{\rho g} + z_2 + \frac{v_2^2}{2g} \tag{6.7}$$

Such that:
p = pressure
ρ = fluid density
g = acceleration due to gravity, in aide_design as `con.GRAVITY`
z = elevation relative to a reference
v = fluid velocity

Notice that each term in this form of the Bernoulli equation has units of $[L]$, even though the terms represent the energy of the fluid, which has units of $\frac{[M]\cdot[L]^2}{[T]^2}$. When energy of the fluid is described in units of length, the term used is called **head** and referred to as h.

There are two important distinctions to keep in mind when using head to talk about a fluids energy. First is that head is dependent on the density of the fluid under consideration. Take mercury, for example, which is around 13.6 times more dense than water. 1 meter of mercury head is therefore equivalent to around 13.6 meters of water head. Second is that head is independent of the amount of fluid being considered, *as long as all the fluid is the same density*. Thus, raising 1 liter of water up by one meter and raising 100 liters of water up by one meter are both equivalent to giving the water 1 meter of water head, even though it requires 100 times more energy to raise the hundred liters than to raise the single liter. Since we are concerned mainly with water in this class, we will refer to water head simply as head.

Going back to the Bernoulli equation, the $\frac{p}{\rho g}$ term is called the pressure head, z is called the elevation head, and $\frac{v^2}{2g}$ is the velocity head. The following diagram shows these various forms of head via a 1 meter deep bucket (left) and a jet of water shooting out of the ground (right).

Assumption in using the Bernoulli equation

Though there are many assumptions needed to confirm that the Bernoulli equation can be used, the main one for the purpose of this class is that energy is not gained or lost throughout the streamline being considered. If we consider more precise fluid mechanics terminology, then friction by viscous forces must be negligible. What this means is that the fluid along the streamline being considered is not losing energy to viscosity. As a result, using the Bernoulli equation implies that energy cant be gained or lost. It can only be transferred between its three forms.

Example problems

Here is a simple worksheet with very straightforward example problems using the Bernoulli equation. Note that the solutions use the pressure-form of the Bernoulli equation. This just means that every term in the equation is multiplied by ρg, so the pressure term is just P. The form of the equation does not affect the solution to the problem it helps solved.

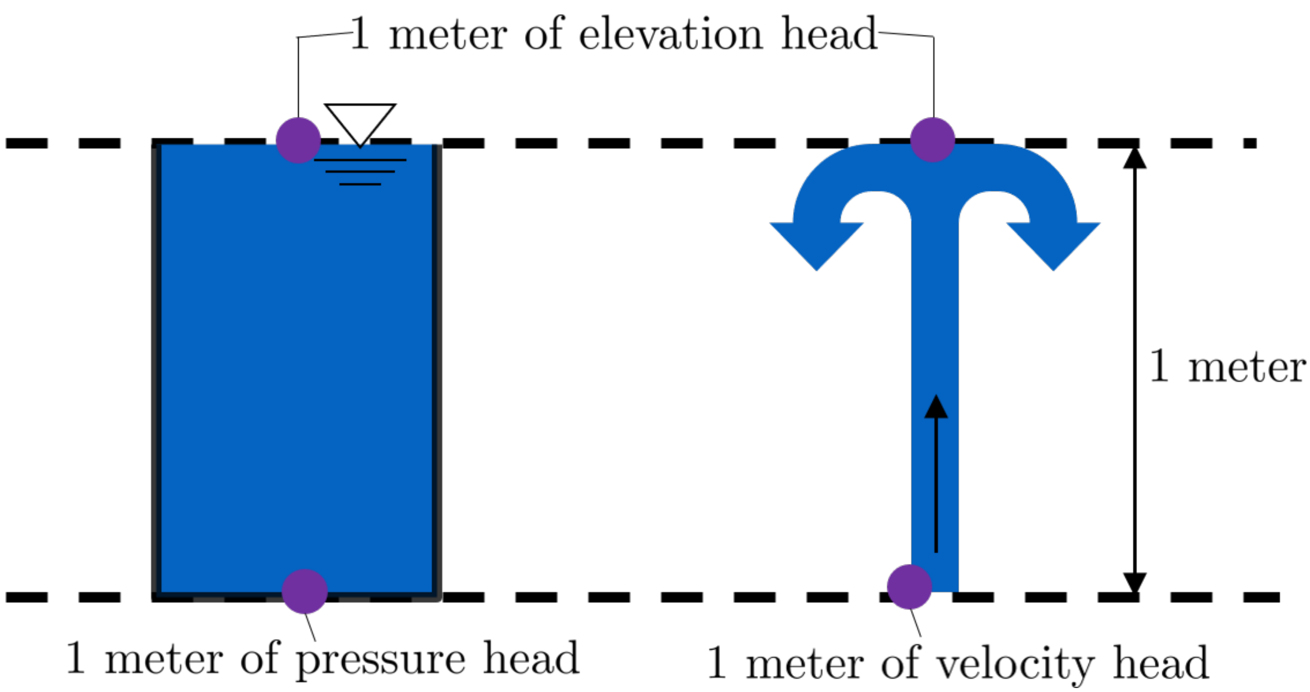


Fig. 6.5: The three forms of hydraulic head.

6.3.2 The Energy Equation

The assumption necessary to use the Bernoulli equation, which is stated above, represents the key difference between the Bernoulli equation and the energy equation for the purpose of this class. The energy equation accounts for the potential addition or loss of fluid energy within the control volume. (L)oss of energy is usually due to viscous friction resisting fluid flow, h_L, or the charging of a (T)urbine, h_T. The most common input of fluid energy into a system is usually caused by a (P)ump within the control volume, h_P.

$$\frac{p_1}{\rho g} + z_1 + \alpha_1 \frac{\bar{v}_1^2}{2g} + h_P = \frac{p_2}{\rho g} + z_2 + \alpha_2 \frac{\bar{v}_2^2}{2g} + h_T + h_L \tag{6.8}$$

Youll also notice the α term attached to the velocity head. This is a correction factor for kinetic energy, and will be neglected in this class; we assume that its value is 1. In the Bernoulli equation, the velocity of a streamline of the fluid is considered, v. The energy equation, however compares control surfaces instead of streamlines, and the velocities across a control surface many not all be the same. Hence, $\bar{v}$ is used to represent the average velocity. Since AguaClara does not use pumps nor turbines, $h_P = h_T = 0$. With these simplifications, the energy equation can be written as follows:

$$\frac{p_1}{\rho g} + z_1 + \frac{\bar{v}_1^2}{2g} = \frac{p_2}{\rho g} + z_2 + \frac{\bar{v}_2^2}{2g} + h_L \tag{6.9}$$

This is the form of the energy equation that you will see over and over again in this book. To summarize, the main difference between the Bernoulli equation and the energy equation for the purposes of this class is energy loss. The energy equation accounts for the fluids loss of energy over time while the Bernoulli equation does not. So how can the fluid lose energy?

6.4 Headloss

Head(L)oss, h_L is a term that is ubiquitous in both this class and fluid mechanics in general. Its definition is exactly as it sounds: it refers to the loss of energy of a fluid as it flows through space. There are two components to head loss:

major losses caused by (f)riction between the fluid the surface its flowing over, $h_{\rm f}$, and minor losses caused by fluid-fluid internal friction resulting from flow (e)xpansions, h_e. These two components combine such that $h_L = h_{\rm f} + h_e$.

6.4.1 Major Losses

These losses are the result of friction between the fluid and the surface over which the fluid is flowing. A force acting parallel to a surface is referred to as shear. It can therefore be said that major losses are the result of shear between the fluid and the surface its flowing over. To help in understanding major losses, consider the following example: imagine, as you have so often in physics class, pushing a large box across the ground. Friction is what resists your efforts to push the box. The farther you push the box, the more energy you expend pushing against friction. The same is true for water moving through a pipe, where water is analogous to the box you want to move, the pipe is similar to the floor that provides the friction, and the major losses of the water through the pipe is analogous to the energy **you** expend by pushing the box.

In this class, we will be dealing primarily with major losses in circular pipes, as opposed to channels or pipes with other geometries. Fortunately for us, Henry Darcy and Julius Weisbach came up with a handy equation to determine the major losses in a circular pipe *under both laminar and turbulent flow conditions*. Their equation is logically and unoriginally named the Darcy-Weisbach equation. It is shown below:

$$h_{\rm f} \, = {\rm f} \frac{L}{D} \frac{\bar v^2}{2g} \tag{6.10}$$

Substituting the continuity equation $Q = \bar v A$ in the form of $\bar v^2 = \frac{16Q^2}{\pi^2 D^4}$ gives another, equivalent form of Darcy-Weisbach which uses flow, Q, instead of velocity, $\bar v$:

$$h_{\rm f} \, = {\rm f} \frac{8}{g \pi^2} \frac{LQ^2}{D^5} \tag{6.11}$$

Such that:
$h_{\rm f}$ = major loss
f = Darcy friction factor
L = pipe length
Q = pipe flow rate
D = pipe diameter

See also:

Function in aide_design: `pc.headloss_fric(FlowRate, Diam, Length, Nu, PipeRough)` Returns only major losses. Works for both laminar and turbulent flow. PipeRough describes the pipe roughness ϵ described shortly below.

Darcy-Weisbach is wonderful because it applies to both laminar and turbulent flow regimes and contains relatively easy to measure variables. The one exception is the Darcy friction factor, f. This parameter is an approximation for the magnitude of friction between the pipe walls and the fluid, and its value changes depending on the whether or not the flow is laminar or turbulent, and varies with the Reynolds number in both flow regimes.

For laminar flow, the friction factor can be determined from the following equation:

$${\rm f} = \frac{64}{\rm Re} \tag{6.12}$$

For turbulent flow, the friction factor is more difficult to determine. In this class, we will use the Swamee-Jain equation:

$${\rm f} = \frac{0.25}{\left[\log\left(\frac{\epsilon}{3.7D} + \frac{5.74}{{\rm Re}^{0.9}}\right)\right]^2} \tag{6.13}$$

Such that:
ϵ = pipe roughness, $[L]$
D = pipe diameter, $[L]$

See also:

Function in aide_design: `pc.fric(FlowRate, Diam, Nu, PipeRough)` Returns f for laminar *or* turbulent flow. For laminar flow, use zero for the `PipeRough` input.

The simplicity of the equation for f during laminar flow allows for substitutions to create a very useful, simplified equation for major losses during laminar flow. This simplification combines the Darcy-Weisbach equation, the equation for the Darcy friction factor during laminar flow, and the Reynolds number formula:

$$h_{\rm f} = {\rm f} \frac{8}{g\pi^2} \frac{LQ^2}{D^5} \tag{6.14}$$

$${\rm f} = \frac{64}{\rm Re} \tag{6.15}$$

$$\rm{Re} = \frac{4Q}{\pi D\nu} \tag{6.16}$$

To form the Hagen-Poiseuille equation for major losses during laminar flow, and *only* during laminar flow:

$$h_{\rm f} = \frac{128\mu LQ}{\rho g\pi D^4} \tag{6.17}$$

$$h_{\rm f} = \frac{32\nu L\bar v}{gD^2} \tag{6.18}$$

The significance of this equation lies in its relationship between $h_{\rm f}$ and Q. Hagen-Poiseuille shows that the terms are directly proportional ($h_{\rm f} \propto Q$) during laminar flow, while Darcy-Weisbach shows that $h_{\rm f}$ grows with the square of Q during turbulent flow ($h_{\rm f} \propto Q^2$). As you will soon see, minor losses, h_e, will grow with the square of Q in both laminar and turbulent flow. This has implications that will be discussed in a future chapter: *Flow Control and Measurement Design*.

In 1944, Lewis Ferry Moody plotted a ridiculous amount of experimental data, gathered by many people, on the Darcy-Weisbach friction factor to create what we now call the Moody diagram. This diagram has makes it easy to find the friction factor f. f is plotted on the left-hand y-axis, relative pipe roughness $\frac{\epsilon}{D}$ is on the right-hand y-axis, and Reynolds number $\rm Re$ is on the x-axis. The Moody diagram is an alternative to computational methods for finding f.

6.4.2 Minor Losses

Unfortunately, there is no simple pushing a box across the ground example to explain minor losses. So instead, consider a hydraulic jump. In the video, you can see lots of turbulence and eddies in the transition region between the fast, shallow flow and the slow, deep flow. The high amount of mixing of the water in the transition region of the hydraulic jump results in significant friction *between water and water*. This turbulent, eddy-induced, fluid-fluid friction results in minor losses, much like fluid-pipe friction results in major losses.

As occurs in a hydraulic jump, a flow expansion (from shallow flow to deep flow) creates the turbulent eddies that result in minor losses. This will be a recurring theme in throughout the course: **minor losses are caused by flow expansions**. Imagine a pipe fitting that connects a small diameter pipe to a large diameter one, as shown in Fig. 6.7 below. The flow must expand to fill up the entire large diameter pipe. This expansion creates turbulent eddies near the union between the small and large pipes, and these eddies result in minor losses. You may already know the equation for minor losses, but understanding where it comes from is very important for effective AguaClara plant design. For this reason, you are strongly recommended to read through its full derivation: *Review: Fluid Mechanics Derivations*.

There are three forms of the minor loss equation that you will see in this class:

$$\textbf{First form}: \quad h_e = \frac{\left(\bar v_{in} - \bar v_{out}\right)^2}{2g} \tag{6.19}$$

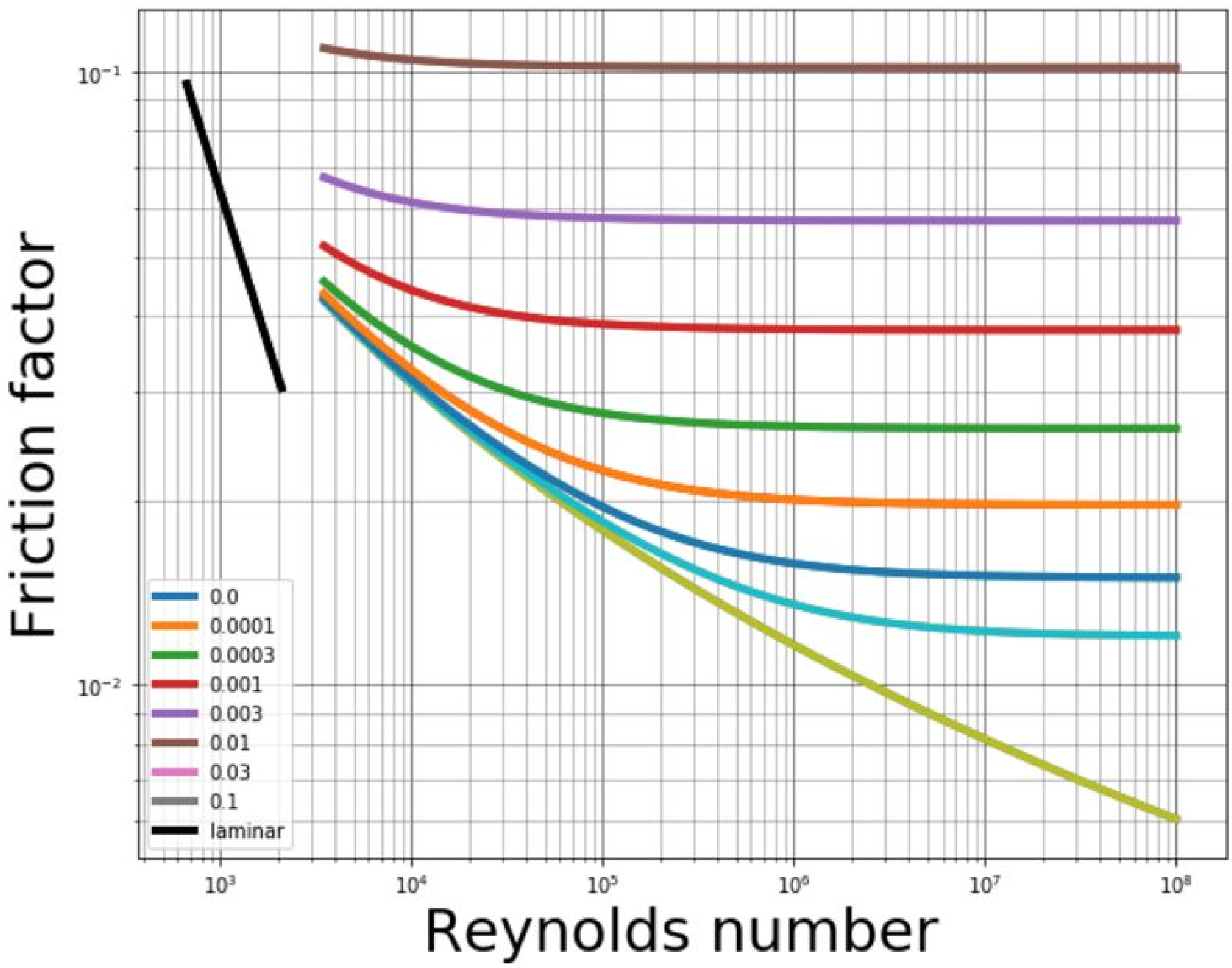


Fig. 6.6: This is the famous and famously useful Moody diagram.

$$\textbf{Second form}: \quad h_e = \left(1 - \frac{A_{in}}{A_{out}}\right)^2 \frac{\bar{v}_{in}^2}{2g} = K_e^{'} \frac{\bar{v}_{in}^2}{2g}, \quad \text{where} \quad K_e^{'} = \left(1 - \frac{A_{in}}{A_{out}}\right)^2 \tag{6.20}$$

$$\textbf{Third form}: \quad h_e = \left(\frac{A_{out}}{A_{in}} - 1\right)^2 \frac{\bar{v}_{out}^2}{2g} = K_e \frac{\bar{v}_{out}^2}{2g}, \quad \text{where} \quad K_e = \left(\frac{A_{out}}{A_{in}} - 1\right)^2 \tag{6.21}$$

Such that:
$K_e^{'}, \ K_e$ = minor loss coefficients, dimensionless

Note: You will most often see $K_e^{'}$ and K_e used without the e subscript, as $K^{'}$ and K.

See also:

Function in aide_design: `pc.headloss_exp_general(Vel, KMinor)` Returns h_e. Can be either the second or third form due to user input of both velocity and minor loss coefficient. It is up to the user to use consistent $\bar{v}$ and K_e.

See also:

Function in aide_design: `pc.headloss_exp(FlowRate, Diam, KMinor)` Returns h_e. Uses third form, K_e.

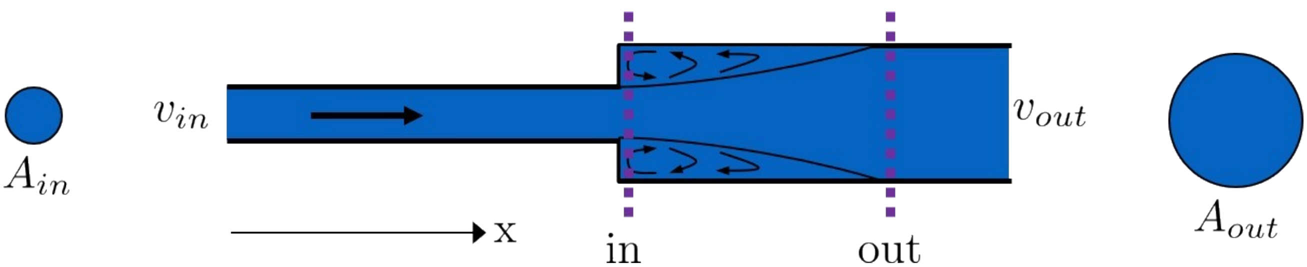


Fig. 6.7: The in and out subscripts in each of the three forms of the minor loss equation refer to this diagram that was used for the derivation.

The second and third forms are the ones which you are probably most familiar with. The distinction between them, however, is critical. First, consider the magnitudes of A_{in} and A_{out}. A_{in} can never be larger than A_{out}, because the flow is expanding. When flow expands, the cross-sectional area it flows through must increase. As a result, both $\frac{A_{out}}{A_{in}} > 1$ and $\frac{A_{in}}{A_{out}} < 1$ must always be true. This means that $K^{'}$ can never be greater than 1, while K technically has no upper limit.

If you have taken CEE 3310, you have seen tables of minor loss coefficients like this one, and they almost all have coefficients greater than 1. This implies that these tables use the third form of the minor loss equation as we have defined it, where the velocity is $\bar{v}_{out}$. There is a good reason for using the third form over the second one: $\bar{v}_{out}$ is far easier to determine than $\bar{v}_{in}$. Consider flow through a pipe elbow, as shown in the image below.

In order to find $\bar{v}_{out}$, we first need to know what (or where) is out and what is in. A simple way to distinguish the two surfaces is that in occurs when the flow is most contracted, and out occurs when the flow has fully expanded after that maximal contraction. Going on these guidelines, Control surface 2 (CS 2) in the figure above above would be in, since it represents the most contracted flow in the elbow-pipe system. Therefore, CS 3 would be out, as it represents the flow having fully expanded after its compression at CS 2.

$\bar{v}_{out}$ is easy to determine because it is the velocity of the fluid as it flows through the entire area of the pipe. Thus, $\bar{v}_{out}$ can be found with the continuity equation, since the flow through the pipe and its diameter are easy to measure, $\bar{v}_{out} = \frac{4Q}{\pi D^2}$. On the other hand, $\bar{v}_{in}$ is difficult to find, as the area of the contracted flow is dependent on the exact geometry of the elbow. This is why the third form of the minor loss equation, as we have defined it, is the most

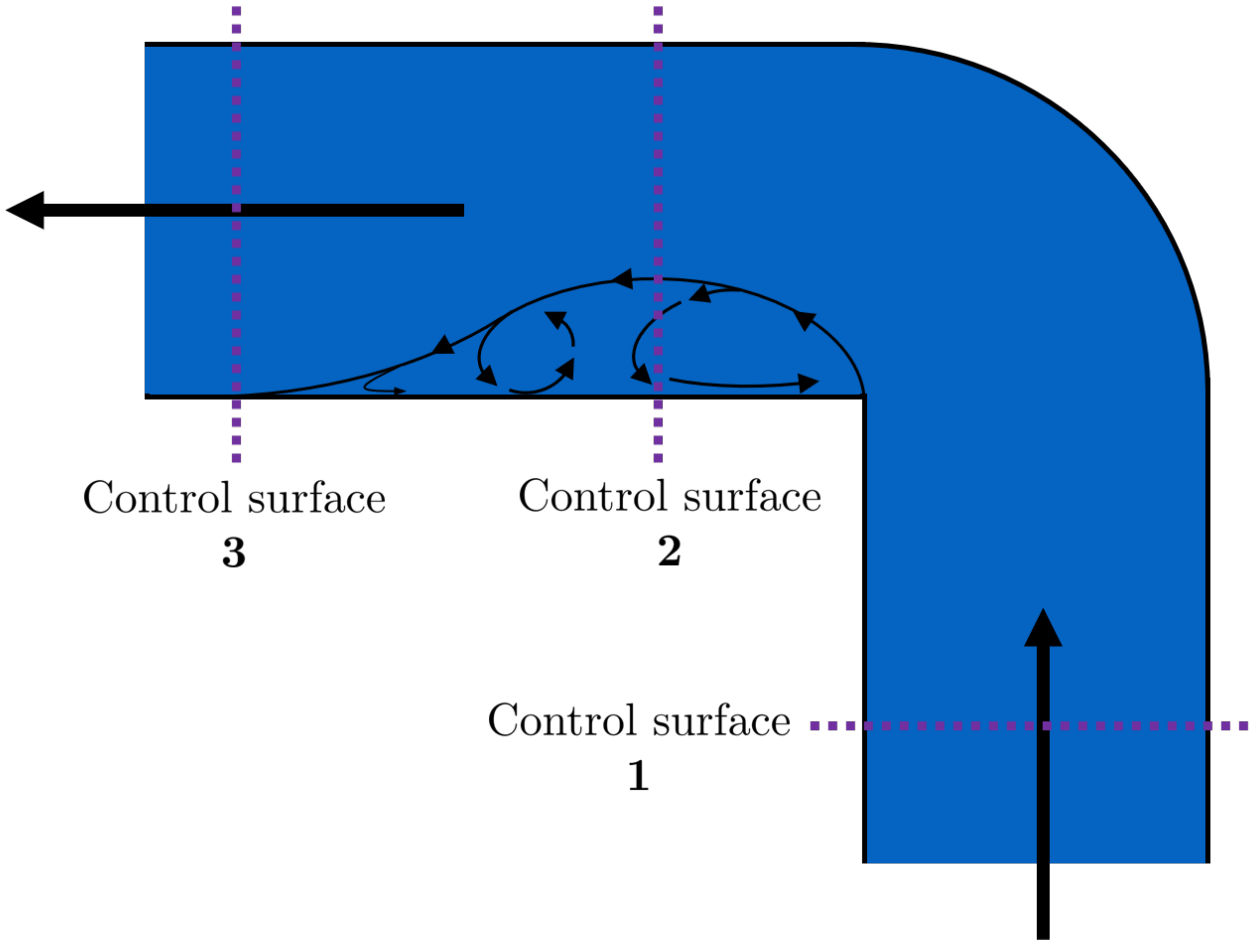


Fig. 6.8: Flow around a pipe elbow results in a minor loss. Control surface 1 can be abbreviated as CS 1

common:

$$h_e = K\frac{\bar{v}_{out}^2}{2g} = \left(\frac{A_{out}}{A_{in}} - 1\right)^2 \frac{\bar{v}_{out}^2}{2g} \tag{6.22}$$

Note: When considering a hydraulic system within a control volume, there can be many sources of minor losses. Instead of saying $h_e = K_1\frac{\bar{v}_{out}^2}{2g} + K_2\frac{\bar{v}_{out}^2}{2g} + ...$ we can simply lump all of the minor loss coefficients into one: $\sum K = K_1 + K_2 + ...$ Thus, it is also common to see this form of the minor loss equation when finding the minor loss across control volumes: $\sum K\frac{v_{out}^2}{2g}$.

6.4.3 Head Loss = Elevation Difference Trick

This trick, also called the control volume trick, or more colloquially, the head loss trick, is incredibly useful for simplifying hydraulic systems and is used all the time in this class.

Consider the following figure:

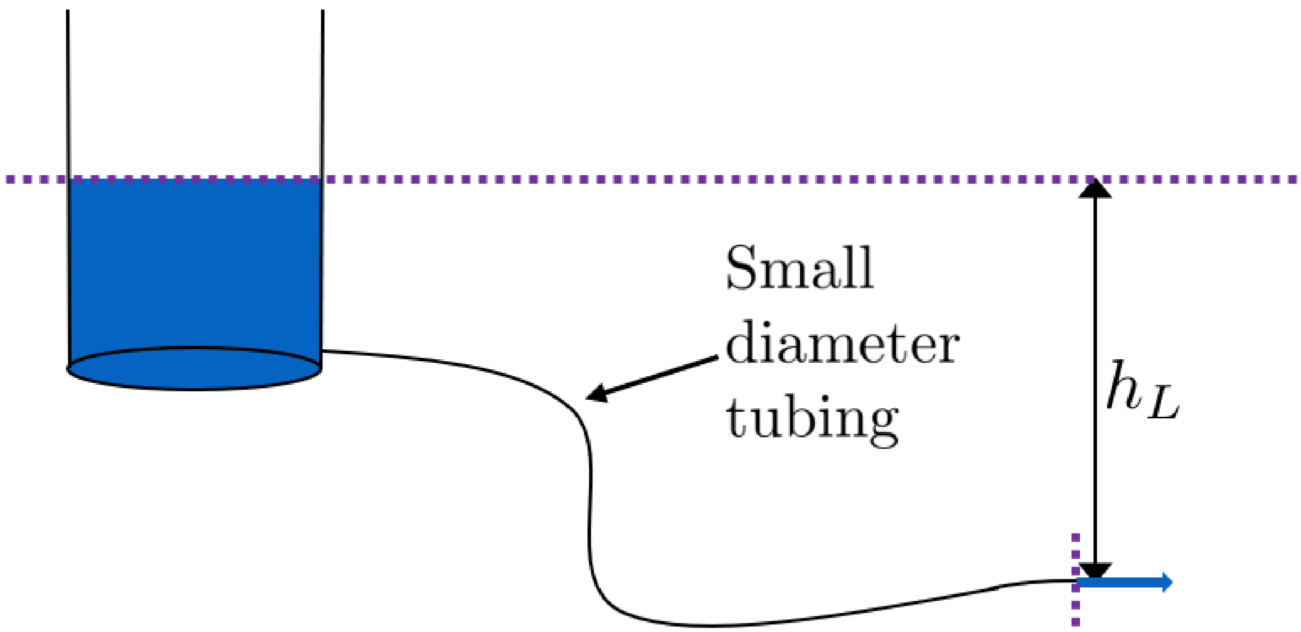


Fig. 6.9: A typical hydraulic system can be used to understand the head loss trick.

In systems like this, where an elevation difference is causing water to flow, the elevation difference is called the **driving head**. In the system above, the driving head is the elevation difference between the water level and the end of the tubing. Usually, driving head is written as Δz or Δh, though above it is labelled as h_L. Doesnt h_L refer to head loss though? Yes it does! Referring to Δh or Δz *IS* the head loss trick, and how it works is explained in the following paragraphs and equations.

The figure is technically violating the energy equation by saying that the elevation difference between the water in the tank and the end of the tube is h_L. It implies that all of the driving head, Δz, is lost to head loss. Since all of the energy is gone, there should not be water flowing out of the tubing. But there is. Lets apply the energy equation across the control surfaces shown in the figure. Pressures at both ends are atmospheric and the velocity of water at the top of tank is negligible.

$$\cancel{\frac{p_1}{\rho g}} + z_1 + \cancel{\frac{\bar{v}_1^2}{2g}} = \cancel{\frac{p_2}{\rho g}} + z_2 + \frac{\bar{v}_2^2}{2g} + h_L \tag{6.23}$$

We now get:

$$\Delta z = \frac{\bar{v}_2^2}{2g} + h_L \tag{6.24}$$

This equation contradicts the figure above, which says that $\Delta z = h_L$ and neglects $\frac{\bar{v}_2^2}{2g}$. The figure above is correct, however, if you apply the head loss trick. The trick incorporates the $\frac{\bar{v}_2^2}{2g}$ term *into* the h_L term as a minor loss. See the math below:

$$\Delta z = \frac{\bar{v}_2^2}{2g} + h_e + h_f \tag{6.25}$$

$$\Delta z = \frac{\bar{v}_2^2}{2g} + \left(\sum K\right)\frac{\bar{v}_2^2}{2g} + h_f \tag{6.26}$$

$$\Delta z = \left(1 + \sum K\right)\frac{\bar{v}_2^2}{2g} + h_f \tag{6.27}$$

$$\Delta z = \left(\sum K\right)\frac{\bar{v}_2^2}{2g} + h_f \tag{6.28}$$

This last step incorporated the kinetic energy term of the energy equation, $\frac{\bar{v}_2^2}{2g}$, into the minor loss equation by saying that its K is 1 and incorporating that 1 into $\sum K$. From here, we reverse our steps to get $\Delta z = h_L$, starting with $h_e = \left(\sum K\right)\frac{\bar{v}_2^2}{2g}$

$$\Delta z = h_e + h_f \tag{6.29}$$

$$\Delta z = h_L \tag{6.30}$$

By applying the head loss trick, you are considering the entire flow of the fluid out of a control volume as energy lost via minor losses. This is just an algebraic trick, the only thing to remember when applying this trick is that $\sum K$ will always be at least 1, even if there are no real minor losses in the system.

6.5 The Orifice Equation

This equation is one that youll see and use again and again throughout this class. Understanding it now will be invaluable, as future concepts will use and build on this equation.

6.5.1 What is a Vena Contracta?

Before describing the equation, we must first understand the concept of a vena contracta. Refer to the figure below.

The flow contracts as the fluid moves past the gate. This happens because the fluid cant make a sharp turn as it tries to go around the gate, as indicated by the streamline in the figure. Instead, the most extreme streamline makes a gradual change in direction. As a result of this gradual turn, the flow contracts and the cross-sectional area the fluid is flowing decreases.

The term vena contracta describes the phenomenon of contracting flow due to streamlines being unable to make sharp turns. Π_{vc} is a dimensionless ratio comparing the flow area at the point of maximal contraction, $A_{downstream}$, and the flow area *before* the contraction, A_{gate}. In the figure above, the equation for the vena contracta coefficient would be:

$$\Pi_{vc} = \frac{A_{downstream}}{A_{gate}} \tag{6.31}$$

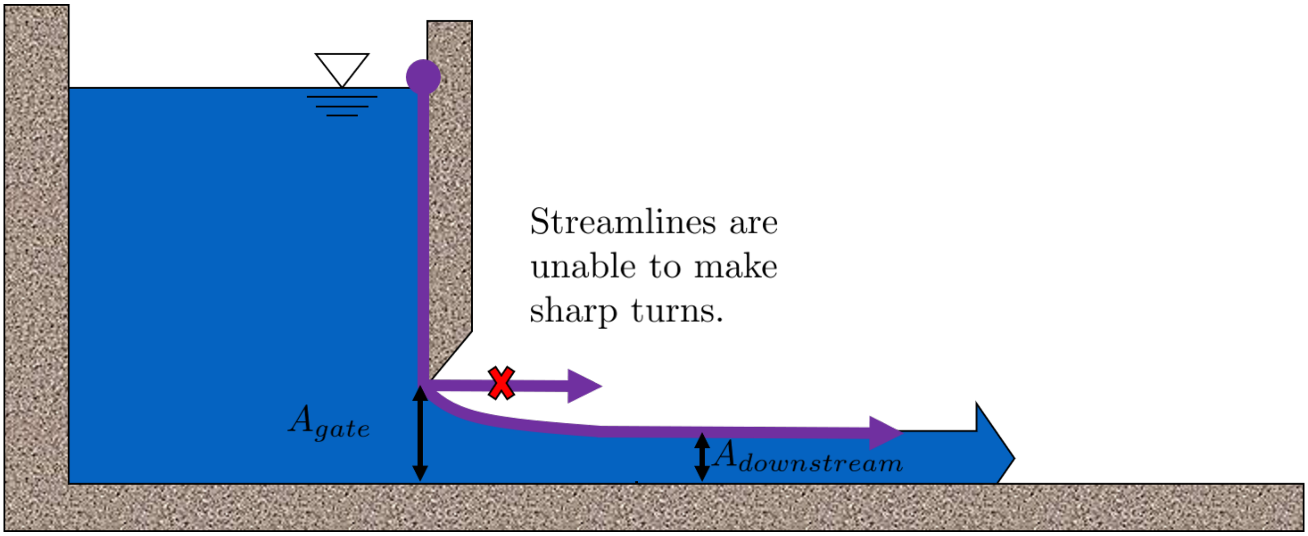


Fig. 6.10: This figure shows flow around a sluice gate. Since streamlines cant make sharp turns, the flow is forced to gradually curve and contract to an area smaller than the area of the gate.

When the most extreme turn a streamline must make is 90ř, the value of the vena contracta coefficient is close to 0.62. This parameter value, 0.62, is in aide_design as `pc.RATIO_VC_ORIFICE`. The vena contracta coefficient value is a function of the flow geometry. Since the ratio always puts the most contracted area over the least contracted area, Π_{vc} is always less than 1.

Important: **A vena contracta coefficient is not a minor loss coefficient.** Though the equations for the two both involve contracted and non-contracted areas, these coefficients are not the same. Minor losses coefficients imply energy loss, and vena contractas do not. Minor losses coefficients deal with flow expansions, and vena contracas deal with flow contractions. Confusing the two coefficients is common mistake that this paragraph will hopefully help you to avoid.

Note: Note that what this class calls Π_{vc} is often referred to as a Coefficient of Contraction, C_c, in other engineering courses and settings.

6.5.2 Origin of the Orifice Equation

The orifice equation is derived from the Bernoulli equation as applied to the purple points in the following image:

At point 1, the pressure is atmospheric and the instantaneous velocity is negligible as the water level in the bucket drops slowly. At point 2, the pressure is also atmospheric. We define the difference in elevations between the two points, $z_1 - z_2$, to be Δh. With these simplifications ($p_1 = \bar{v}_1 = p_2 = 0$) and assumptions ($z_A - z_B = \Delta h$), the Bernoulli equation becomes:

$$\Delta h = \frac{\bar{v}_2^2}{2g} \tag{6.32}$$

Substituting the continuity equation $Q = \bar{v}A$ in the form of $\bar{v}_2^2 = \frac{Q^2}{A_{vc}^2}$, the vena contracta coefficient in the form of $A_{vc} = \Pi_{vc} A_{or}$ yields:

$$\Delta h = \frac{Q^2}{2g\Pi_{vc}^2 A_{or}^2} \tag{6.33}$$

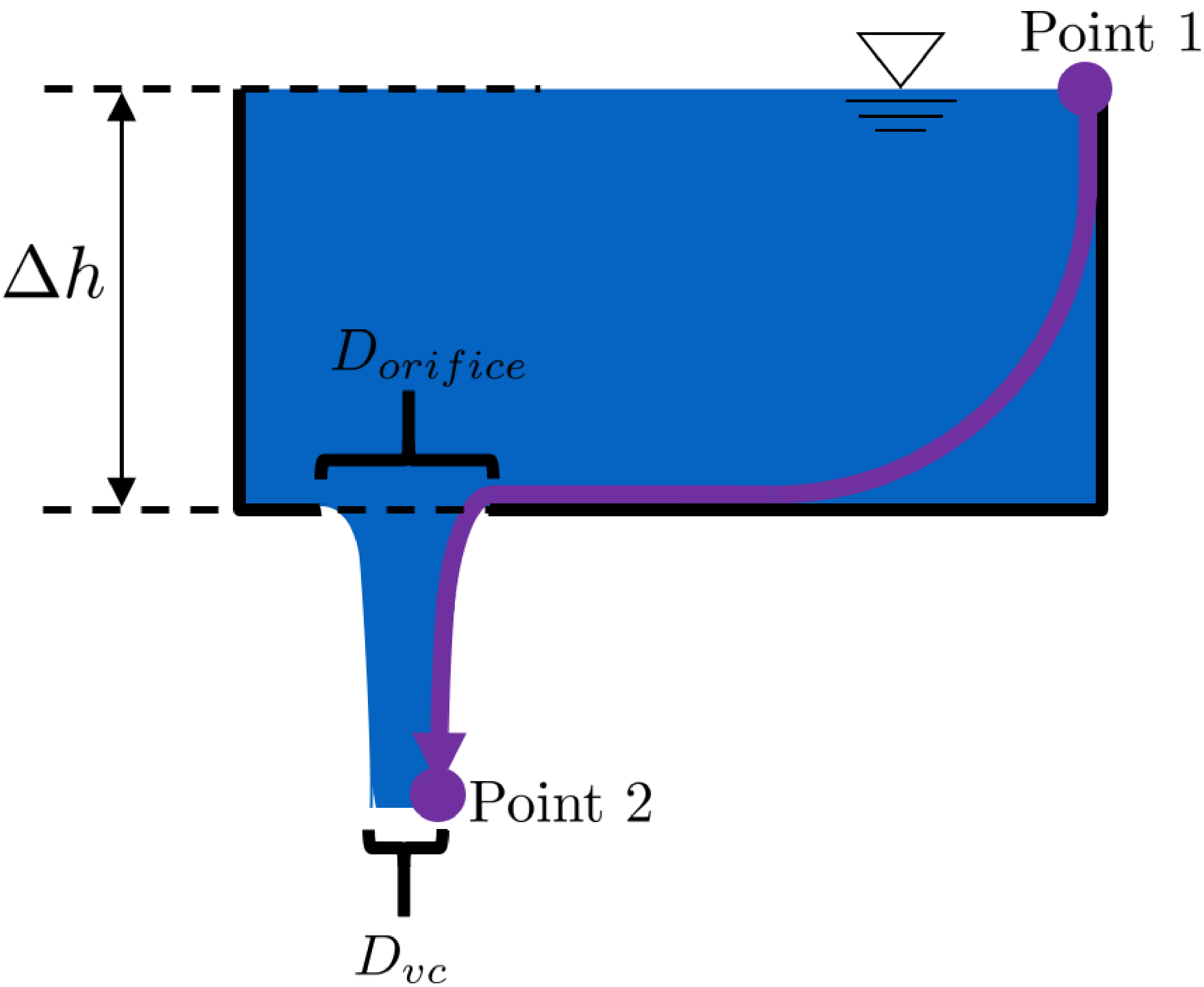


Fig. 6.11: Flow through a hole in the bottom of a bucket is a great example of the orifice equation.

Which, rearranged to solve for Q gives **The Orifice Equation:**

$$Q = \Pi_{vc} A_{or} \sqrt{2g\Delta h} \tag{6.34}$$

Such that:
Π_{vc} = 0.62 = vena contracta coefficient, in aide_design as `pc.RATIO_VC_ORIFICE`
A_{or} = orifice area- NOT contracted flow area
Δh = elevation difference between orifice and water level

See also:

Equation in aide_design: `pc.flow_orifice(Diam, Height, RatioVCOrifice)` Returns flow through a horizontal orifice.

See also:

Equation in aide_design: `pc.flow_orifice_vert(Diam, Height, RatioVCOrifice)` Returns flow through a vertical orifice. The height parameter refers to height above the center of the orifice.

There are two configurations for an orifice in the tank holding a fluid: horizontal and vertical. These are both displayed in the figure below. The orifice equation written is for a horizontal orifice; the equation for flow through vertical orifice equation requires integration or the orifice equation across its height to return the correct flow. This is explored in the Flow Control and Measurement Examples section.

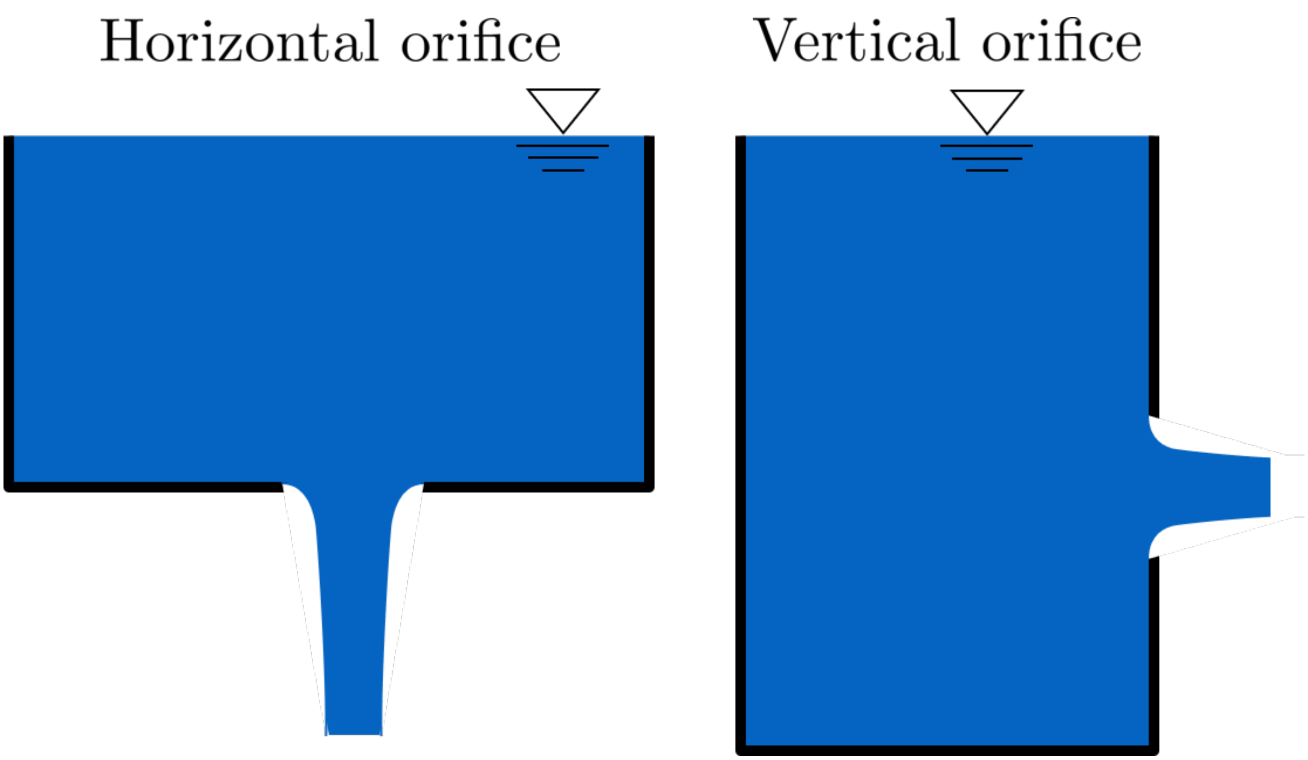


Fig. 6.12: The descriptions vertical and horizontal **apply to the orientation of the orifices,** not to the orientation of the fluid coming out of the orifices.

6.6 Section Summary

1. **Introductory Concepts:**

   - **Continuity** means that the mass of a fluid is conserved as it flows, and implies a constant density. The continuity equation has two purposes:

     (a) Relating the average velocity of a fluid, $\bar{v}$, to its flow rate, Q, via the cross-sectional area, A, that it flows through. When the fluid is flowing in a pipe, we can simply this even further to relate the flow rate and velocity to the pipes diameter, D. The final equation below is only used for circular pipes, as it includes a pipe diameter.

     $$Q = \bar{v}A = \bar{v}\frac{\pi D^2}{4} \tag{6.35}$$

     (a) Finding the average velocity or flow when the geometry of a fluids flow changes, as the mass of the fluid must be conserved when it transitions through flow geometries.

     $$Q_1 = Q_2 \tag{6.36}$$

     $$\bar{v}_1 A_1 = \bar{v}_2 A_2 \tag{6.37}$$

     $$\bar{v}_1 \frac{\pi D_1^2}{4} = \bar{v}_2 \frac{\pi D_2^2}{4} \tag{6.38}$$

   - **Laminar and Turbulent flow** describe the disorder and chaos of fluid flow. The **Reynolds number,** Re is used to distinguish laminar from turbulent flow. For $\mathrm{Re} < 2100$, flow is considered laminar. For $\mathrm{Re} > 2100$, flow is considered turbulent. The equations for the Reynolds number are below:

     $$\mathrm{Re} = \frac{\bar{v}D}{\nu} = \frac{4Q}{\pi D \nu} = \frac{\rho \bar{v} D}{\mu} \tag{6.39}$$

   - **Control volumes vs Streamlines.** This section is quite short, a summary would simply repeat what the sections says. The section is its own summary; read it here: *Streamlines and Control Volumes*

2. **Bernoulli vs Energy equations:** The Bernoulli equation assumes that energy is conserved throughout a streamline or control volume. The Energy equation assumes that there is energy loss, or head loss h_L. This head loss is composed of major losses, h_{f}, and minor losses, h_e.

   Bernoulli equation:

   $$\frac{p_1}{\rho g} + z_1 + \frac{\bar{v}_1^2}{2g} = \frac{p_2}{\rho g} + z_2 + \frac{\bar{v}_2^2}{2g} \tag{6.40}$$

   Energy equation, simplified to remove pumps, turbines, and α factors:

   $$\frac{p_1}{\rho g} + z_1 + \frac{\bar{v}_1^2}{2g} = \frac{p_2}{\rho g} + z_2 + \frac{\bar{v}_2^2}{2g} + h_L \tag{6.41}$$

3. **Major losses:** Defined as the energy loss due to shear between the walls of the pipe/flow conduit and the fluid. The Darcy-Weisbach equation is used to find major losses in both laminar and turbulent flow regimes. The equation for finding the Darcy friction factor, f, changes depending on whether the flow is laminar or turbulent. The Moody diagram is a common graphical method for finding f. During laminar flow, the Hagen-Poiseuille equation, which is just a combination of Darcy-Weisbach, Reynolds number, and $\mathrm{f} = \frac{64}{\mathrm{Re}}$, can be used

Darcy-Weisbach equation:

$$h_{\rm f} = {\rm f} \frac{L}{D} \frac{\bar v^2}{2g} \tag{6.42}$$

For water treatment plant design we tend to use plant flow rate, Q, as our master variable and thus we have.

$$h_{\rm f} = {\rm f} \frac{8}{g \pi^2} \frac{LQ^2}{D^5} \tag{6.43}$$

f for laminar flow:

$${\rm f} = \frac{64}{\rm Re} = \frac{16 \pi D \nu}{Q} = \frac{64 \nu}{\bar v D} \tag{6.44}$$

f for turbulent flow:

$${\rm f} = \frac{0.25}{\left[\log \left(\frac{\epsilon}{3.7D} + \frac{5.74}{{\rm Re}^{0.9}} \right) \right]^2} \tag{6.45}$$

Hagen-Poiseuille equation for laminar flow:

$$h_{\rm f} = \frac{32 \mu L \bar v}{\rho g D^2} = \frac{128 \mu Q}{\rho g \pi D^4} \tag{6.46}$$

4. **Minor losses:** Defined as the energy loss due to the generation of turbulent eddies when flow expands. Once more: minor losses are caused by flow expansions. There are three forms of the minor loss equation, two of which look the same but use different coefficients ($K^{'}$ vs K) and velocities ($\bar v_{in}$ vs $\bar v_{out}$). *Make sure the coefficient you select is consistent with the velocity you use.* The third form, written in purple, is the most commonly used form of the minor loss equation.

$$\textbf{First form}: \quad h_e = \frac{(\bar v_{in} - \bar v_{out})^2}{2g} \tag{6.47}$$

$$\textbf{Second form}: \quad h_e = \left(1 - \frac{A_{in}}{A_{out}}\right)^2 \frac{\bar v_{in}^2}{2g} = K_e^{'} \frac{\bar v_{in}^2}{2g}, \quad \text{where} \quad K_e^{'} = \left(1 - \frac{A_{in}}{A_{out}}\right)^2 \tag{6.48}$$

$$\textbf{Third form}: \quad h_e = \left(\frac{A_{out}}{A_{in}} - 1\right)^2 \frac{\bar v_{out}^2}{2g} = K_e \frac{\bar v_{out}^2}{2g}, \quad \text{where} \quad K_e = \left(\frac{A_{out}}{A_{in}} - 1\right)^2 \tag{6.49}$$

5. **Major and minor losses vary with flow:** While it is generally important to know how increasing or decreasing flow will affect head loss, it is even more important for this class to understand exactly how flow will affect head loss. As the table below shows, head loss will always be proportional to flow squared during turbulent flow. During laminar flow, however, the exponent on Q will be between 1 and 2 depending on the proportion of major to minor losses.

Table 6.1: Proportionality between head loss h_L and flow rate Q for different flow regimes and types of head loss.

$h_L proptoQ^?$	Major Losses	Minor Losses
Laminar	Q	Q^2
Turbulent	Q^2	Q^2

6. The **head loss trick**, also called the control volume trick, can be used to incorporate the kinetic energy out term of the energy equation, $\frac{\bar v_2^2}{2g}$, into head loss as a minor loss with $K = 1$, so the minor loss equation becomes $\left(1 + \sum K\right) \frac{\bar v^2}{2g}$. This is used to be able to say that $\Delta z = h_L$ and makes many equation simplifications possible in the future.

7. **Orifice equation and vena contractas:** The orifice equation is used to determine the flow out of an orifice given the elevation of water above the orifice. This equation introduces the concept of vena contracta, which describes flow contraction due to the inability of streamlines to make sharp turns. The equation shows that the flow out of an orifice is proportional to the square root of the driving head, $Q \propto \sqrt{\Delta h}$. Depending on the orientation of the orifice, vertical (like a hole in the side of a bucket) or horizontal (like a hole in the bottom of a bucket), a different equation in aide_design should be used.

   The Orifice Equation:

$$Q = \Pi_{vc} A_{or} \sqrt{2g\Delta h} \tag{6.50}$$

CHAPTER

SEVEN

REVIEW: FLUID MECHANICS DERIVATIONS

7.1 Minor Loss Equation

This section contains the derivation of the minor loss equation using the following figure as a reference. The derivation begins with a slightly simplified energy equation across the control volume shown. Our energy equation begins with h_P and h_T having been eliminated.

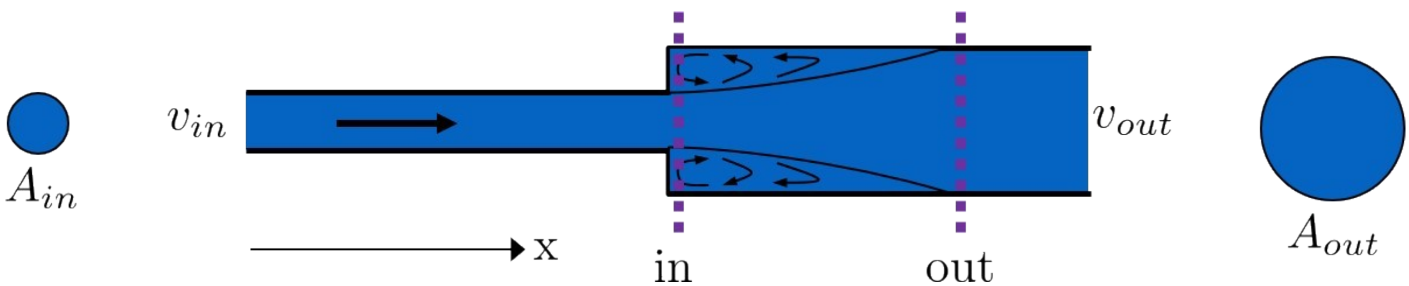


Fig. 7.1: This is the system we will use to derive the minor loss equation.

$$\frac{p_{in}}{\rho g} + z_{in} + \frac{\bar{v}_{in}^2}{2g} = \frac{p_{out}}{\rho g} + z_{out} + \frac{\bar{v}_{out}^2}{2g} + h_L \tag{7.1}$$

Since the elevations at the center of the in and out control surfaces are the same, we can eliminate z_{in} and z_{out}. As we are considering such a small length of pipe, we will neglect the major loss component of head loss. Thus, $h_L = h_e + \cancel{h_f}$. The following three equations are all the same, simply rearranged to solve for h_e.

$$\frac{p_{in}}{\rho g} + \frac{\bar{v}_{in}^2}{2g} = \frac{p_{out}}{\rho g} + \frac{\bar{v}_{out}^2}{2g} + h_e \tag{7.2}$$

$$\frac{p_{in} - p_{out}}{\rho g} = \frac{\bar{v}_{out}^2 - \bar{v}_{in}^2}{2g} + h_e \tag{7.3}$$

$$h_e = \frac{p_{in} - p_{out}}{\rho g} + \frac{\bar{v}_{in}^2 - \bar{v}_{out}^2}{2g} \tag{7.4}$$

This last equation has h_e as a function of four variables (p_{in}, p_{out}, v_{in}, and v_{out}); we would like it to be a function of only one. Thus, we will invoke conservation of momentum in the horizontal direction across our control volume to remove variables. The difference in momentum from the in point to the out point is driven by the pressure difference between each end of the control volume. We will be considering the pressure at the centroid of our control surfaces, and we will neglect shear along the pipe walls. After these assumptions, our momentum equation becomes the following:

$$M_{in,\,x} + M_{out,\,x} = F_{p_{in},\,x} + F_{p_{out},\,x} \tag{7.5}$$

Such that:

M_x = momentum flowing through the control volume in the x-direction

F_{p_x} = force due to pressure acting on the boundaries of the control volume in the x-direction

Recall that momentum is mass times velocity for solids, mv, with units of $\frac{[M][L]}{[T]}$. Since we consider water flowing through a pipe, there is not one singular mass or one singular velocity. Instead, there is a mass flow rate, or a mass per time indicated by $\dot{m} = \rho Q$, which has units of $\frac{[M]}{[T]}$. Therefore, the momentum for a fluid is $\rho Q \bar{v}$. Applying the continuity equation $Q = \bar{v}A$, we get to the following equation for the momentum of a fluid flowing through a pipe which we will use in this derivation, $M = \rho \bar{v}^2 A$. The pressure force is simply the pressure at the centroid of the flow multiplied by the area the pressure is acting upon, pA.

To ensure correct sign convention, we will make each side of the equation negative for reasons discussed shortly. Since $\bar{v}_{in} > \bar{v}_{out}$, the left hand side will be $M_{out} - M_{in}$ in order to be negative. The reduction in velocity from in to out causes an increase in pressure, therefore $p_{in} - p_{out}$ is negative. With these substitutions, the conservation of momentum equation becomes as follows:

$$M_{out} - M_{in} = p_{in} - p_{out} \tag{7.6}$$

$$\rho \bar{v}_{out}^2 A_{out} - \rho \bar{v}_{in}^2 A_{in} = p_{in} A_{out} - p_{out} A_{out} \tag{7.7}$$

Note that the area term attached to p_{in} is actually A_{out} instead of A_{in}, as one might think. This is because $A_{out} = A_{in}$. We chose our control volume to start a few millimeters into the larger pipe, which means that the cross-sectional area does not change over the course of the control volume.

Dividing both sides of the equation by $A_{out} \rho g$, we obtain the following equation, which contains the very same pressure term as our adjusted energy equation above, equation (7.4). This is why we chose a negative sign convention.

$$\frac{p_{in} - p_{out}}{\rho g} = \frac{\bar{v}_{out}^2 - \bar{v}_{in}^2 \frac{A_{in}}{A_{out}}}{g} \tag{7.8}$$

Now, we combine the momentum, continuity, and adjusted energy equations:

$$\text{Energy equation}: \ h_e = \frac{p_{in} - p_{out}}{\rho g} + \frac{\bar{v}_{in}^2 - \bar{v}_{out}^2}{2g} \tag{7.9}$$

$$\text{Momentum equation}: \ \frac{p_{in} - p_{out}}{\rho g} = \frac{\bar{v}_{out}^2 - \bar{v}_{in}^2 \frac{A_{in}}{A_{out}}}{g} \tag{7.10}$$

$$\text{Continuity equation}: \ \frac{A_{in}}{A_{out}} = \frac{\bar{v}_{out}}{\bar{v}_{in}} \tag{7.11}$$

To obtain an equation for minor losses with just two variables, $\bar{v}_{in}$ and $\bar{v}_{out}$.

$$h_e = \frac{\bar{v}_{out}^2 - \bar{v}_{in}^2 \frac{\bar{v}_{out}}{\bar{v}_{in}}}{g} + \frac{\bar{v}_{in}^2 - \bar{v}_{out}^2}{2g} \tag{7.12}$$

Now we will combine the two terms. The numerator and denominator of the first term, $\frac{\bar{v}_{out}^2 - \bar{v}_{in}^2 \frac{\bar{v}_{out}}{\bar{v}_{in}}}{g}$ will be multiplied by 2 to become $\frac{2\bar{v}_{out}^2 - 2\bar{v}_{in}^2 \frac{\bar{v}_{out}}{\bar{v}_{in}}}{2g}$. The equation then looks like:

$$h_e = \frac{\bar{v}_{out}^2 - 2\bar{v}_{in}\bar{v}_{out} + \bar{v}_{in}^2}{2g} \tag{7.13}$$

7.1.1 Final Forms of the Minor Loss Equation

Factoring the numerator yields to the first final form of the minor loss equation:

$$\textbf{First form}: \quad h_e = \frac{\left(\bar{v}_{in} - \bar{v}_{out}\right)^2}{2g} \tag{7.14}$$

From here, the two other forms of the minor loss equation can be derived by solving for either $\bar v_{in}$ or $\bar v_{out}$ using the ubiquitous continuity equation $\bar v_{in} A_{in} = \bar v_{out} A_{out}$:

$$\textbf{Second form}: \quad h_e = \left(1 - \frac{A_{in}}{A_{out}} \right)^2 \frac{\bar v_{in}^2}{2g} \;=\; K_e^{'} \frac{\bar v_{in}^2}{2g}, \quad \text{where} \quad K_e^{'} = \left(1 - \frac{A_{in}}{A_{out}} \right)^2 \tag{7.15}$$

$$\textbf{Third form}: \quad h_e = \left(\frac{A_{out}}{A_{in}} - 1 \right)^2 \frac{\bar v_{out}^2}{2g} \;=\; K_e \frac{\bar v_{out}^2}{2g}, \quad \text{where} \quad K_e = \left(\frac{A_{out}}{A_{in}} - 1 \right)^2 \tag{7.16}$$

Note: You will often see $K_e^{'}$ and K_e used without the e subscript, they will appear as $K^{'}$ and K.

Being familiar with these three forms and how they are used will be of great help throughout the class. The third form is the one that is most commonly used.

CHAPTER

EIGHT

FLOW CONTROL AND MEASUREMENT INTRODUCTION

8.1 Tank with a Valve

8.1.1 Flow Q and Water Level h as a Function of Time

Our first step is to see if we can get constant head out of a simple system. The most simple flow control system is a bucket or tank with a hole in it. This system is too coarse to provide constant head. One step above that is a bucket or tank with a valve. This is where we begin our search for constant head.

Using the setup of in the image below, we derive the following equation for flow Q through the valve as a function of time t. The derivation is found here: *for a Tank with a Valve*. You are advised to read through it if you are at all confused about this equation.

$$\frac{Q}{Q_0} = 1 - \frac{1}{2}\frac{t}{t_{Design}}\frac{h_{Tank}}{h_0} \tag{8.1}$$

Such that:
Q = $Q(t)$ = flow of hypochlorite through valve at time t
Q_0 = flow of hypochlorite through valve at time $t = 0$
t = elapsed time
t_{Design} = time it *would* take for tank to empty if flow stayed constant at Q_0, which it does not
h_{Tank} = elevation of water level with reference to tank bottom at time t = 0
h_0 = elevation of water level with reference to the valve at time $t = 0$

This equation has historically give students some trouble, and while its nuances are explained in the derivation, they will be quickly summarized here:

- t_{Design} is **NOT** the time it takes to drain the tank. It is the time that it *would* take to drain the tank *if* the flow rate at time $t = 0$, Q_0, were the flow rate forever, which it is not. t_{Design} was used in the derivation to simplify the equation, which is why this potentially-confusing parameter exists. The actual time it takes to drain the tank lies somewhere between t_{Design} and $2\,t_{Design}$ and depends on the ratio $\frac{h_{Tank}}{h_0}$.
- h_{Tank} is not the same as h_0. h_{Tank} is the height of water level in the tank with reference to the tank bottom. h_0 is the water level in the tank with reference to the valve. Neither change with time, they both refer to the water level at one instance in time, $t = 0$. Therefore, $h_0 \geq h_{Tank}$ is always true. If the tank is elevated far above the valve, then the $h_0 >> h_{Tank}$. If the valve is at the same elevation as the bottom of the tank, then $h_0 = h_{Tank}$. Please refer to the figure above to clarify h_0 and h_{Tank}.

We can use the proportionality $Q \propto \sqrt{h}$, which applies to both minor losses and orifices to form a relationship between water level in the tank h and time t. This proportionality comes from rearranging the minor loss equation $h = K\frac{Q^2}{2gA^2}$ for Q instead of h. A table of proportionality between Q and math:*h* can be found in Table 6.1

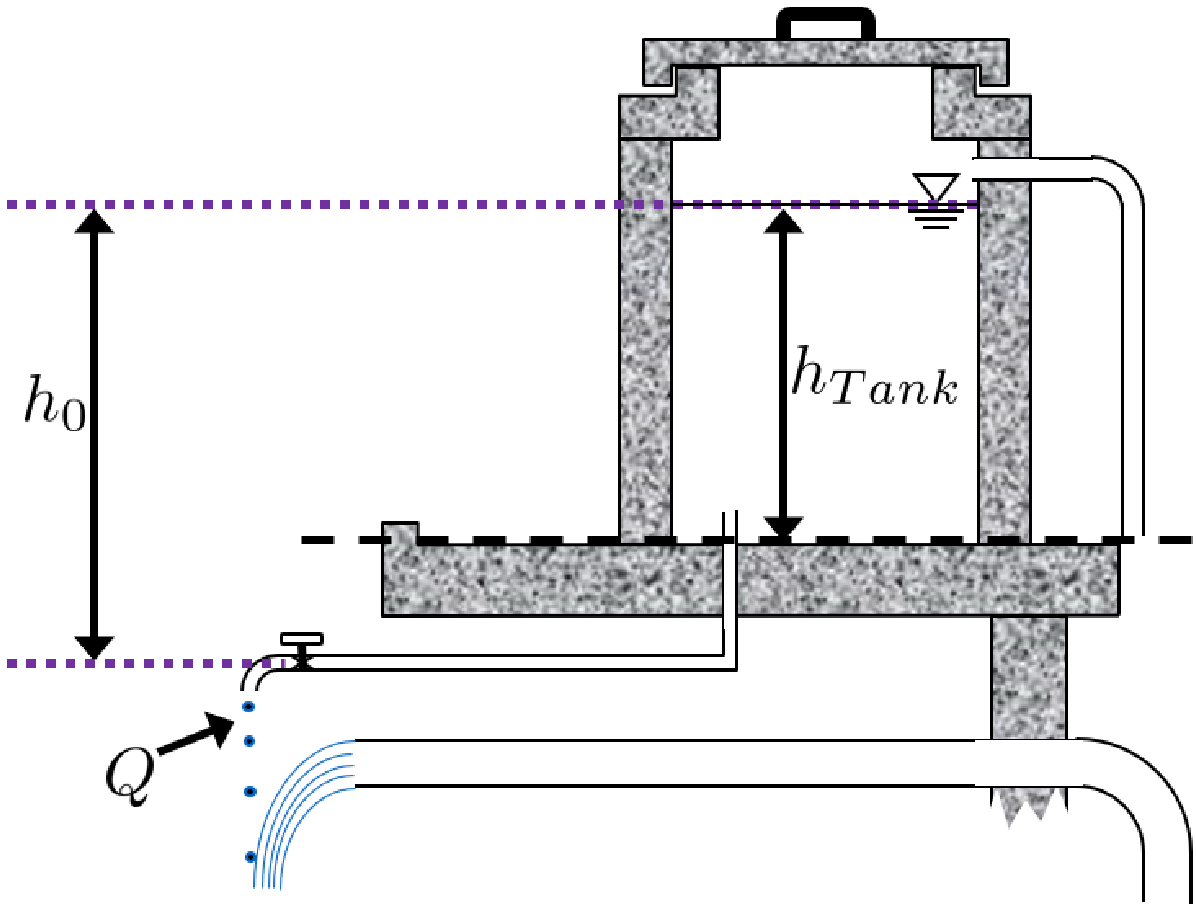


Fig. 8.1: This figure shows the variables that are defined in the equation above.

Using equation (8.1) and this proportionality relationship, we make the following plots. On the left, the valve is at the same elevation as the bottom of the tank, or $h_{Tank} = h_0$. Our attempt to get a continuous flow rate out of this system is to make $\frac{h_{Tank}}{h_0}$ very small by elevating the tank far above the valve. On the right, $\frac{h_{Tank}}{h_0} = \frac{1}{50}$. While the plot looks great and provides essentially constant head, elevating the tank by 50 times its height is not realistic. The tank with a valve is not a solution to the constant head problem.

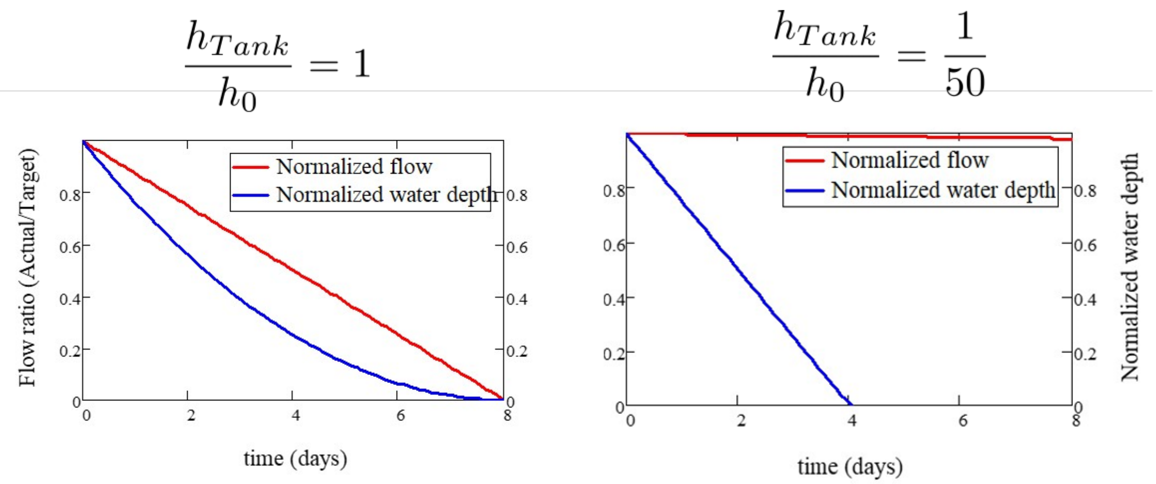


Fig. 8.2: These graphs show how manipulation of the variables in the $Q(t)$ expression can result in effectively constant head.

8.1.2 Drain System for a Tank

While the tank with a valve scenario is not a good constant head solution, we can use our understanding of the system to properly design drain systems for AguaClara reactors like flocculators and sedimentation tanks, since they are just tanks with valves. The derivation for the following equation is here, along with more details on AguaClaras pipe stub method for draining tanks: *and for Tank Drain Equation*. The derived Tank Drain equation is as follows:

$$D_{Pipe} = \sqrt{\frac{8L_{Tank}W_{Tank}}{\pi t_{Drain}}} \left(\frac{H_{Tank}\sum K}{2g} \right)^{\frac{1}{4}} \tag{8.2}$$

The equation can also be rearranged to solve for the time it would take to drain a tank given its dimensions and a certain drain pipe size:

$$t_{Drain} = \frac{8L_{Tank}W_{Tank}}{\pi D_{Pipe}^2} \left(\frac{H_{Tank}\sum K}{2g} \right)^{\frac{1}{2}} \tag{8.3}$$

Such that:
D_{Pipe} = Diameter of the drain piping
$L_{Tank}, W_{Tank}, H_{Tank}$ = Tank dimensions
t_{Drain} = Time it takes to drain the tank
$\sum K$ = Sum of all the minor loss coefficients in the system

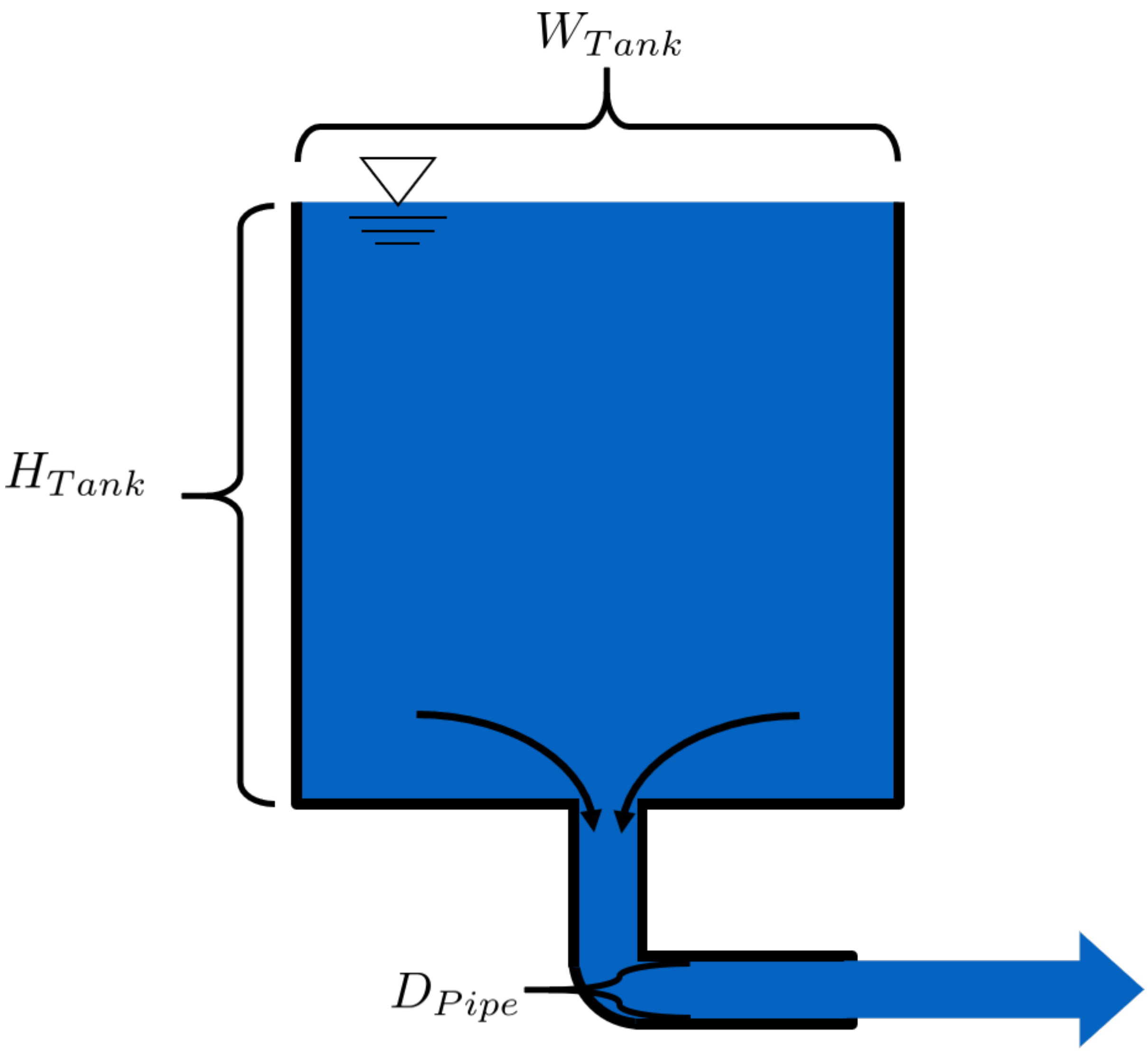


Fig. 8.3: Variables for draining a tank

CHAPTER

NINE

FLOW CONTROL AND MEASUREMENT DESIGN

This section explores AguaClaras search for constant head in checmial dosing. The term **constant head** means that the driving head of a system, Δz or Δh, does not change over time, even as water flows through or out of the system. Constant head implies constant flow, since the energy driving the flow does not change.

The challenge of constant head in chemical dosing for water treatment plants is not *just* providing one continuous flow of chemicals; it is also varying that flow of chemicals as the flow rate through the plant changes, so that the concentration of chemicals in the raw water stays the same.

9.1 Important Terms and Equations

Terms:

1. Dose
2. Coagulant
3. Chlorination
4. Turbidity
5. Organic Matter
6. Constant Head Tank
7. Sutro weir

Equations:

1. Hagen-Poiseuille equation

9.2 AguaClara Flow Control and Measurement Technologies

Each technology or component for this section will have five subsections:

- **What it is**
- **What it does and why**
- **How it works**
- **Notes**

Before diving into the technologies, recall the purpose of the chemicals that we are seeking to constantly **dose**, and why it is important to keep a constant, specific dose. Also recall that dose means concentration of chemical *in the water we are trying to treat*, not in the stock tanks of the chemicals. Coagulant like alum, PAC, and some iron-based chemicals are used to turn small particles into bigger particles, allowing them to be captured more easily. Waters with high turbidity, indicative of a lot of particles like clay and bacteria, require more coagulant to treat effectively. Additionally, waters with a lot of organic matter require significantly more coagulant to treat. Chlorine is used to disinfect water that has already been fully treated. A proper and consistent chlorine dose is required, as too low of a dose creates a risk of reintroduction of pathogens in the distribution system and too high of a dose increases the risk of carcinogenic disinfection byproduct formation.

Important: This section will often refer to the proportionality between flow Q and head Δh (recall that $\Delta h = h_L$ after applying the head loss trick) by using the proportional to symbol, $\propto$. It is important to remember that it doesnt necessarily matter whether Q or h_L goes first, $Q \propto \sqrt{h_L}$ is equivalent to saying that $h_L \propto Q^2$.

9.2.1 Almost Linear Flow Controller

What it is

This device consists of a bottle of chemical solution, called the **Constant Head Tank** (CHT), a float valve to keep a solution in the CHT at a constant water level, a flexible tube starting at the bottom of the CHT, and many precisely placed and equally spaced holes in a pipe, as the image below shows. The holes in the pipe hold the other end of the tube that starts at the CHT.

Chemical solution, either coagulant or chlorine, is stored in a stock tank somewhere above the CHT. A different tube connects the stock tank to the float valve within the CHT.

What it does and why

This flow controller provides a constant flow of chemical solution to the water in the plant. When the end of the flexible tube is placed in a hole, the elevation difference between the water level in the bottle and the hole is set and does not change unless the tube is then placed in another hole. Thus, a constant flow is provided while the end of the tube is not moved.

As has been mentioned previously, the amount of chlorine and coagulant that must be added to the raw water changes depending on the flow rate of the plant; the change is necessary to keep the dose constant. More water flowing through the plant means more chlorine is necessary to maintain the dose of chlorine in the treated water. For coagulant, there are also other factors aside from plant flow rate that impact the required dose, including the turbidity and amount of organic matter in the water. The operator must be able to change the dose of both coagulant and chlorine quickly and easily, and they must be able to know the value of the new dose they set. The Almost Linear Flow Controller accomplishes this by having a large number of holes in the flow control pipe next to the CHT. This large number of holes gives the operator many options for adjusting the dose, and let them quickly change the flow of chemicals into the raw water by moving the end of the flexible tube from one hole to another.

How it works

The idea behind this flow controller is to have a linear relationship between Q and h_L (remember that h_L is equal to Δh when we apply the head loss trick), which can be written as $Q \propto h_L$. Here, Q is the flow of chemicals out of the flexible tube, and h_L is the elevation difference between the water level in the CHT and the end of the flexible tube.

As you remember from section 1.5, the summary of Fluids Review, $Q \propto \Delta h$, or $\Delta h \propto Q$ as it was written in the section summary, is only true for the combination of major losses and laminar flow, which makes applicable the Hagen-Poiseuille equation. Therefore, the flow must always be laminar in the flexible tube that goes between the CHT and the holes, and major losses must far exceed minor losses.

It is easy to design for laminar flow, but the Almost Linear Flow Controller was unable to make major losses far exceed minor losses. The bending in the flexible tube caused a lot of minor losses which changed in magnitude depending on exactly how the tube was bent. This made the flow controller almost linear, but that wasnt good enough.

Notes

- This flow controller is **no longer used by AguaClara.**
- The tube connecting the CHT to the outlet of chemicals must really belong and, more importantly, **straight** to form a linear relationship between driving head and flow. This was not true for the Almost Linear Flow Controller. When you read about the Linear Chemical Flow Controller (CDC), you will be learning about the replacement to the Almost Linear Flow Controllers replacement.

9.2.2 Linear Flow Orifice Meter (LFOM)

What it is

The LFOM is a weir shape cut into a pipe. It was meant to imitate the Sutro Weir while being far easier to build. The LFOM is a pipe with rows of holes, or orifices, drilled into it. There are progressively fewer holes per row as you move up the LFOM, as the shape is meant to resemble half a parabola on each side. The size of all holes is the same, and the amount of holes per row are precisely calculated. Water in the entrance tank flows into and down the LFOM, towards the rapid mix and flocculator.

What it does and why

The LFOM does one thing and serves two purposes.

What it does:

The LFOM creates a linear relationship between water level in the entrance tank and the flow out of the entrance tank. *It does not control the flow through the plant.* If the LFOM were replaced with a hole in the bottom of the entrance tank, the same flow rate would go through the plant, the only difference being that the water level in the entrance tank would scale with flow squared $h \propto Q^2$ instead of $h \propto Q$. For example, if an LFOM has 10 rows of holes and has been designed for a plant whose maximum flow rate is 10 L/s, then the operator knows that the number of rows submerged in water is equal to the flow rate of the plant in L/s. So if the water were up to the third row of holes, there would be 3 L/s of water flowing through the plant.

Why it is useful:

1. Allows the operator to measure the flow through the plant quickly and easily, explained above.
2. Allows for the Linear Chemical Dose Controller, which will be explained next, to automatically adjust the flow of coagulant/chlorine into the plant as the plant flow rate changes. This means the operator would only need to adjust the flow of coagulant when there is a change in turbidity or organic matter.

How it works

This is best understood with examples. By shaping a weir differently, different relationships between Q and h are formed: * In the case of a rectangular weir, $Q \propto h^{\frac{3}{2}}$ * In the case of a v-notch weir, $Q \propto h^{\frac{5}{2}}$ * In the case of a Sutro weir and thus LFOM, $Q \propto h$.

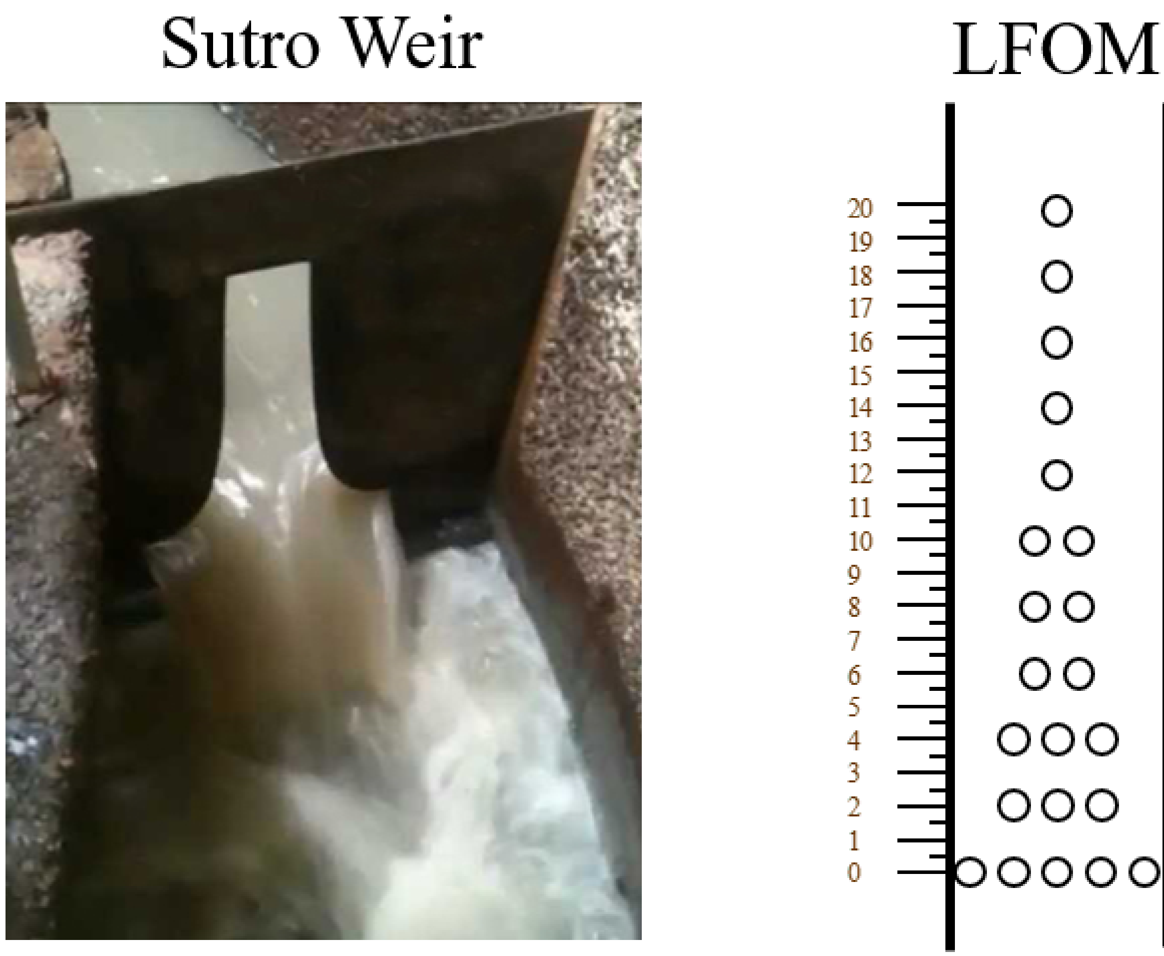


Fig. 9.1: On the left is a sutro weir. On the right is AguaClaras approximation of the sutro weirs geometery. This elegant innovation is called a linear flow orifice meter, or LFOM for short.

Notes

- The LFOM is not perfect. Before the water level reaches the second row of holes, the LFOM is simulating a rectangular weir, and thus $h \not\propto Q$. The Sutro weir also experiences this problem.
- If the water level exceeds the topmost row of the LFOMs orifices, the linearity also breaks down. The entire LFOM begins to act like an orifice, the exponent of Q in $h \propto Q$ becomes greater than 1. This is because the LFOM approaches orifice behavior, and for orifices, $h \propto Q^2$.

9.2.3 Linear Chemical Dose Controller (CDC)

Since the Linear Chemical Dose Controller has become the standard in AguaClara, it is often simply called the Chemical Dose Controller, **or CDC for short**. It can be confusing to describe with words, so be sure to flip through the slides in the Flow Control and Measurement powerpoint, as they contain very, very, helpful diagrams of the CDC.

What it is

The CDC brings together the LFOM and many improvements to the Almost Linear Flow Controller. Lets break it down, with the image below as a guide.

1. Start at the Constant Head Tank (CHT). This is the same set up as the Almost Linear Flow Controller. The stock tank feeds into the CHT, and the float valve makes sure that the water level in the constant head tank is always the same.

2. Now the tubes. These fix the linearity problems that were the main problem in the Almost Linear Flow Controller. * The tube connected to the bottom of the CHT is large diameter to minimize any head loss through it.

- The three thin, straight tubes are designed to generate a lot of major losses and to minimize any minor losses. This is to make sure that major losses far exceed any minor losses, which will ensure that the Hagen-Poiseuille equation is applicable and that flow will be directly proportional to the head, $Q \propto \Delta h$. Why are there 3 tubes?
    1. **3 short instead of 1 short** Removing 2 of the 3 tubes would mean 3 times the flow through the remaining tube. This means the velocity in the tube would be 3 times as fast. Since minor losses scale with v^2 and major losses only scale with v, this would increase the ratio of $\frac{\text{minor losses}}{\text{major losses}}$, which would break the linearity were trying to achieve. It would also increase the total head loss through the system, resulting in a lower maximum flow rate than before.
    2. **1 long instead of 3 short** One tube whose length is equal to the three combined would be inconveniently long, and would suffer from the same problems as above. There would be even more head loss through the tube, since its length would be longer.
- The large-diameter tube on the right of the three thin, straight tubes is where the chemicals flow out. The end of the tube is connected to both a slider and a drop tube. The drop tube allows for supercritical flow of the chemical leaving the dosing tubes; once the chemical enters the drop tube it falls freely and no longer affects the CDC system.

3. The slider rests on a lever. This lever is the critical part of the CDC, it connects the water level in the entrance tank, which is adjusted by the LFOM, to the difference in head between the CHT and the end of the dosing tube. This allows the flow of chemicals to automatically adjust to a change in the plant flow rate, maintaining a constant dose in the plant water. One end of the lever tracks the water level in the entrance tank by using a float. The counterweight on the other side of the lever is to make sure the float floats, since this float is usually made of PVC, which is more dense than water.
4. The slider itself controls the dose of chemicals. For any given plant flow rate, the slider can be adjusted to increase or decrease the amount of chemical flowing through the plant.

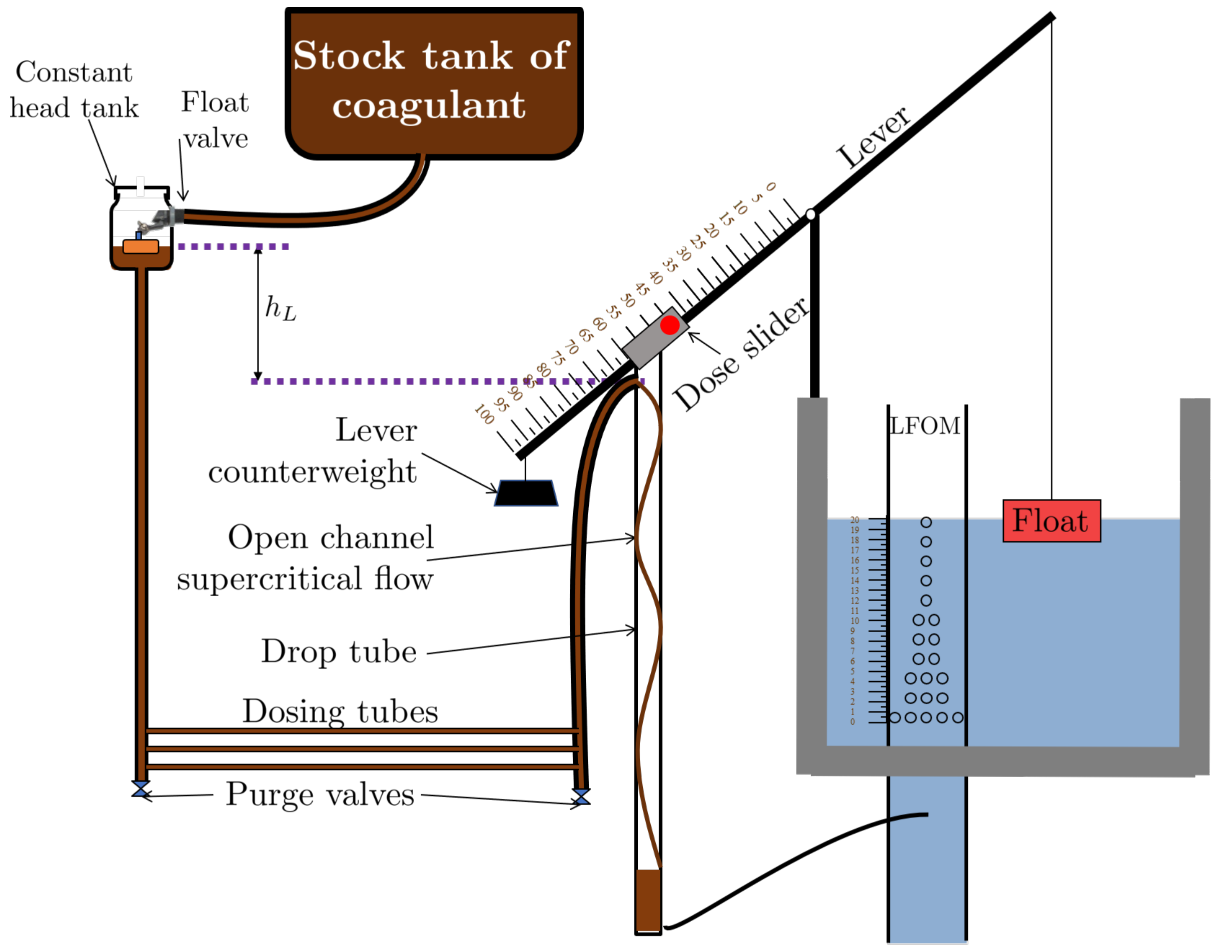


Fig. 9.2: This is the setup of the chemical dose controller.

What it does and why

The CDC makes it easy and accurate to dose chemicals. The flow of chemicals automatically adjusts to changes in the plant flow rate to keep a constant dose, set by the operator. When a turbidity event occurs, the operator can change the dose of coagulant by moving the coagulant slider *lower* on the lever to increase the dose. The slider has labelled marks so the operator can record the dose accurately.

How it works

A lot of design has gone into the CDC. The design equations and their derivations that the following steps are based on can be found here: *Design Equations for the Linear Chemical Dose Controller (CDC)*, and you are very, very strongly encouraged to read them.

The CDC can be designed manually using the equations from the derivation linked above or via aide_design, using the equations found in cdc_functions.py. Either way, the design algorithm is roughly the same:

1. Calculate the maximum flow rate, $Q_{Max,\,Tube}$, through each available dosing tube diameter D that keeps error due to minor losses below 10% of total head loss. Recall that tubing diameter is an array, as there are many diameters available at hardware stores and suppliers. This means that for each step, there will be as many solutions as there are reasonable diameters available.

$$Q_{Max,\,Tube} = \frac{\pi D^2}{4}\sqrt{\frac{2h_L g \Pi_{Error}}{\sum K}} \tag{9.1}$$

2. Calculate how much flow of chemical needs to pass through the CDC at maximum plant flow and maximum chemical dose. This depends on the concentration of chemicals in the stock tank.

$$Q_{Max,\,CDC} = \frac{Q_{Plant} \cdot C_{Dose,\,Max}}{C_{StockTank}} \tag{9.2}$$

3. Calculate the number of dosing tubes required if the tubes flow at maximum capacity (round up)

$$n_{Tubes} = \text{ceil}\left(\frac{Q_{Max,\,CDC}}{Q_{Max,\,Tube}}\right) \tag{9.3}$$

4. Calculate the length of dosing tube(s) that correspond to each available tube diameter.

$$L_{Min} = \left(\frac{g h_L \pi D^4}{128\nu Q_{Max}} - \frac{Q_{Max}}{16\pi\nu}\sum K\right) \tag{9.4}$$

5. Select a tube length from your array of solutions. Pick the longest dosing tube that you can, keeping in mind that the tube(s) must be able to fit in the plant and cant be longer than the length of the plant wall it will be placed along.

6. Finally, select the dosing tube diameter and flow rate corresponding to the selected tube length.

Notes

Nothing in life is perfect, and the CDC is no exception. It has a few causes of inaccuracy which go beyond non-zero minor losses: * Float valves are not perfect. There will still be minor fluctuations of the fluid level in the CHT which will result in imperfect dosing. * Surface tension may resist the flow of chemicals from the dosing tube into the drop tube during low flows. Since the CDC design does not consider surface tension, this is a potential source of error. * The lever and everything attached to it are not weightless. Changing the dose of coagulant or chlorine means moving the slider along the lever. Since the slider and tubes attached to it (drop tube, dosing tube) have mass, moving the slider means that the torque of the lever is altered. This means that the depth that the float is submerged is changed, which affects Δh of the system. This can be remedied by making the floats diameter as large as possible, which makes these fluctuations small. This problem can not be avoided entirely.

9.3 Section Summary

1. **Tank with a valve:** .. math:

```
\frac{Q}{Q_0} = 1 - \frac{1}{2} \frac{t}{t_{Design}} \frac{h_{Tank}}{h_0}
```

This equation describes flow Q as a function of time t of a fluid leaving a tank through a valve. Attempting to get this tank with a valve system to yield constant head means raising the tank far, far above the valve that controls the flow. This is unreasonable when designing a flow control system for constant dosing, but can be used to design systems to drain a tank. See the section above for a description of the variables in the equation.

2. **LFOM:** The LFOM makes the water level in the entrance tank linear with respect to the flow out of the entrance tank. This is useful in measuring the flow and is a critical component in AguaClaras chemical dosing system. The LFOM *measures* the flow through the plant, it does not *control* the flow through the plant.

3. **The Linear Chemical Dose Controller (CDC)** combines the: * linear relationship between water level and flow in the entrance tank caused by the LFOM, * linear relationship between elevation difference and flow caused by the Hagen-Poiseuille equation, which is only valid for major losses under laminar flow, and * a lever to link the two linear relationships

To keep the chemical dose constant by automatically adjusting the addition of coagulant and chlorine as the plant flow rate varies. Two sliders on the lever allows the operator to change the dose of coagulant and chlorine independently of the plant flow rate.

CHAPTER

TEN

FLOW CONTROL AND MEASUREMENT DERIVATIONS

10.1 $Q(t)$ for a Tank with a Valve

This document contains the derivation of the flow through a tank-with-a-valve over time, $Q(t)$. Our reference will be a simple hypochlorinator, shown in the following image. In the image, a hypochlorite solution is slowly dripping and mixing with piped source water, thereby disinfecting it. The valve is almost closed to make sure that the hypochlorite solution drips instead of flows. At the end of this document is an image which shows the variables in the final equation.

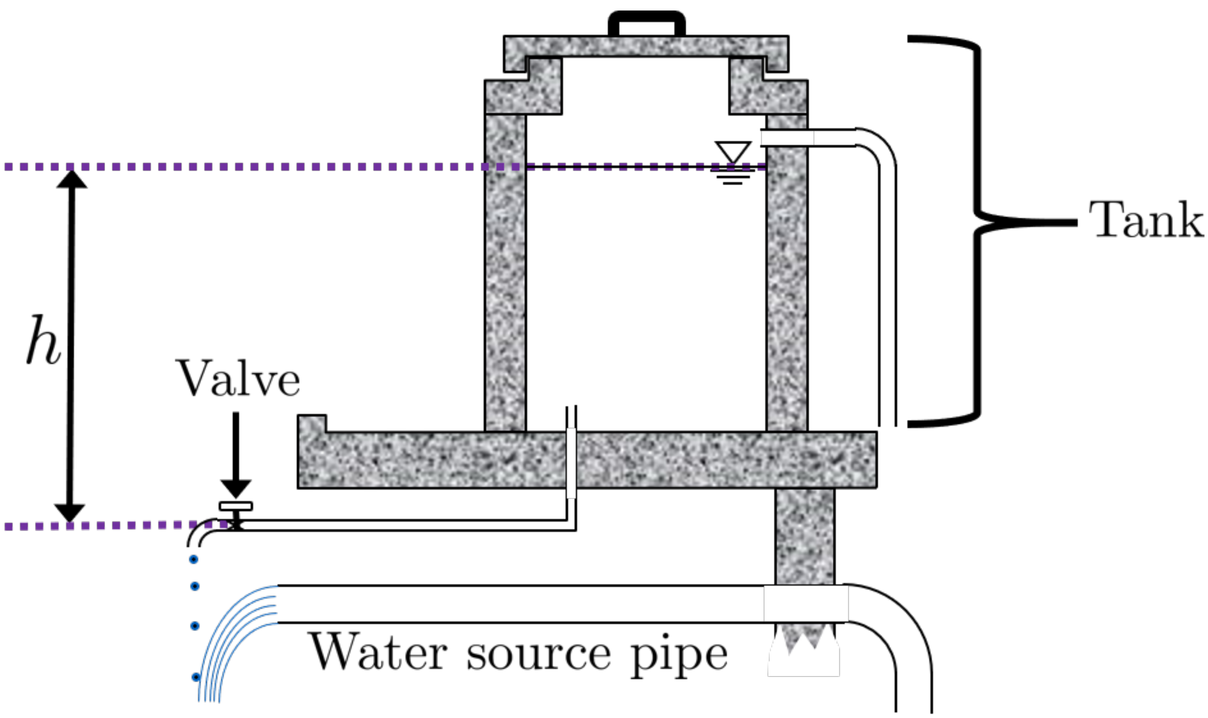


Fig. 10.1: This is a common setup for chlorinating water before distributing it to a nearby community.

This derivation begins by finding two equations for flow, Q, through the hypochlorinator and setting them equal to each other. First, the rate of change of the volume of hypochlorite solution in the tank is equivalent to the flow out of the hypochlorinator. Since the volume of hypochlorite solution in the tank is equal to the tanks cross-sectional area times it height, we get the following equation:

$$Q = -\frac{d\forall}{dt} = -\frac{A_{Tank} dh}{dt} \quad (10.1)$$

Such that:
$\frac{d\forall}{dt}$ = rate of change in volume of solution in the tank
$\frac{dh}{dt}$ = rate of change in height of water (hypochlorite solution) level with time

Our other equation for flow is the head loss equation. Since major losses are negligible for a short pipe-low flow rate system, we only need to consider minor losses. The only real minor loss in this system occurs in the almost-closed valve that is dripping the hypochlorite solution. However, we will also use the head loss trick. Therefore, the total driving head of the system h is equal to the minor losses:

$$h = h_e = \left(\sum K\right) \frac{Q^2}{2gA_{Valve}^2} \tag{10.2}$$

Bear in mind that this is the second form of the minor loss equation as described in *this derivation*. Rearranging the minor loss equation to solve for Q, it looks like this:

$$Q = A_{Valve}\sqrt{\frac{2h_e g}{\sum K}} \tag{10.3}$$

Now we can set both equations for Q equal to each other and move them both to one side:

$$A_{Tank}\frac{dh}{dt} + A_{Valve}\sqrt{\frac{2gh}{\sum K}} = 0 \tag{10.4}$$

From here, calculus and equation substitution dominate the derivation. Separating the variables of the equation immediately above, we get the following integral:

$$\frac{-A_{Tank}}{A_{Valve}\sqrt{\frac{2g}{\sum K}}} \int\limits_{h_0}^{h} \frac{dh}{\sqrt{h}} = \int\limits_{0}^{t} dt \tag{10.5}$$

Which, when integrated, yields:

$$\frac{-A_{Tank}}{A_{Valve}\sqrt{\frac{2g}{\sum K}}} \cdot 2\left(\sqrt{h} - \sqrt{h_0}\right) = t \tag{10.6}$$

And solved for $\sqrt{h}$ returns:

$$\sqrt{h} = \sqrt{h_0} - t\frac{A_{Valve}}{2A_{tank}}\sqrt{\frac{2g}{\sum K}} \tag{10.7}$$

At this point, the steps and equation substitutions may begin to seem unintuitive. Do not worry if you do not understand why *exactly* a particular substitution is occurring. Since we determined above that $h_e = h$, our equation above for $\sqrt{h}$ is also an equation for $\sqrt{h_e}$. As such, we will plug the equation above back into the minor loss equation solved for Q from above, $Q = A_{Valve}\sqrt{\frac{2h_e g}{\sum K}}$, to produce:

$$Q = A_{Valve}\sqrt{\frac{2g}{\sum K}}\left(\sqrt{h_0} - t\frac{A_{Valve}}{2A_{tank}}\sqrt{\frac{2g}{\sum K}}\right) \tag{10.8}$$

Now we can focus on getting rid of the variables A_{Valve}, $\sum K$, and A_{tank}. By using the minor loss equation once more, we can remove both A_{Valve} and $\sum K$. Consider the initial state of the system, when the hypochlorinator is set up and starts dropping its first few drops of hypochlorite solution into the water. The initial flow rate, Q_0, and elevation difference between the water level and the valve, h_0, can be input into the minor loss equation, which can then be solved for A_{Valve}:

$$A_{Valve} = \frac{Q_0}{\sqrt{\frac{2h_0 g}{\sum K}}} \tag{10.9}$$

Plugging this equation for A_{Valve} into the equation for Q just above, we get the following two equations, in which the second equation is a simplified version of the first:

$$Q = Q_0 \frac{1}{\sqrt{h_0}} \left(\sqrt{h_0} - \frac{Q_0 t}{2 A_{Tank} \sqrt{h_0}} \right) \tag{10.10}$$

$$\frac{Q}{Q_0} = 1 - \frac{t Q_0}{2 A_{Tank} h_0} \tag{10.11}$$

This next step will eliminate A_{Tank}. However, it requires some clever manipulation that has a tendency to cause some confusion. We will define a new parameter, t_{Design}, which represents the time it would take to empty the tank **if the initial flow rate through the valve, :math:'Q_0', stays constant in time**. Of course, the flow Q through the valve does not stay constant in time, which is why this derivation document exists. But imagining this hypothetical t_{Design} parameter allows us to form the following equation:

$$Q_0 t_{Design} = A_{Tank} h_{Tank} \tag{10.12}$$

This equation describes draining all the hypochlorite solution from the tank. The volume of the solution, $A_{Tank} h_{Tank}$, is drained in t_{Design}. Rearranged, the equation becomes:

$$\frac{Q_0}{A_{Tank}} = \frac{h_{Tank}}{t_{Design}} \tag{10.13}$$

Such that:

h_{Tank} = elevation of water level in the tank with reference to tank bottom at the initial state, $t = 0$

Here lies another common source of confusion. h_{Tank} is not the same as h_0. h_{Tank} is the height of water level in the tank with reference to the tank bottom. h_0 is the water level in the tank with reference to the valve. Therefore, $h_0 \geq h_{Tank}$ is true if the valve is located at or below the bottom of the tank. If the tank is elevated far above the valve, then the $h_0 >> h_{Tank}$. If the valve is at the same elevation as the bottom of the tank, then $h_0 = h_{Tank}$. Please refer to the following image to clarify h_0 and h_{Tank}. Also note that both h_{Tank} and h_0 are not variables, they are constants which are defined by the initial state of the hypochlorinator, when the solution just begins to flow.

Finally, our fabricated equivalence, $\frac{Q_0}{A_{Tank}} = \frac{h_{Tank}}{t_{Design}}$ can be plugged into $\frac{Q}{Q_0} = 1 - \frac{tQ_0}{2A_{Tank}h_0}$ to create the highly useful equation for flow rate as a function of time for a drip hypochlorinator:

$$\frac{Q}{Q_0} = 1 - \frac{1}{2} \frac{t}{t_{Design}} \frac{h_{Tank}}{h_0} \tag{10.14}$$

Which can be slightly rearranged to yield:

$$Q(t) = Q_0 \left(1 - \frac{1}{2} \frac{t}{t_{Design}} \frac{h_{Tank}}{h_0} \right) \tag{10.15}$$

Such that:

$Q = Q(t)$ = flow of hypochlorite through valve at time t

t = elapsed time

t_{Design} = time it would take for tank to empty *if* flow stayed constant at Q_0, which it does not

h_{Tank} = elevation of water level with reference to tank bottom

h_0 = elevation of water level with reference to the valve

How does this tank with a valve scenario differ from the hole in a bucket scenario? some might ask. If you are interested, you may go through the derivation on your own using the orifice equation instead of the minor loss equation for the first step. If you do so youll find that the equation remains almost the same, the only difference being that the $\frac{h_{Tank}}{h_0}$ term drops out for an orifice, as $h_{Tank} = h_0$. The big difference in the systems lies with the flexibility of having a valve. It can be tightened or loosened to change the flow rate, whereas changing the size of an orifice multiple times in a row is not recommended and is usually irreversible.

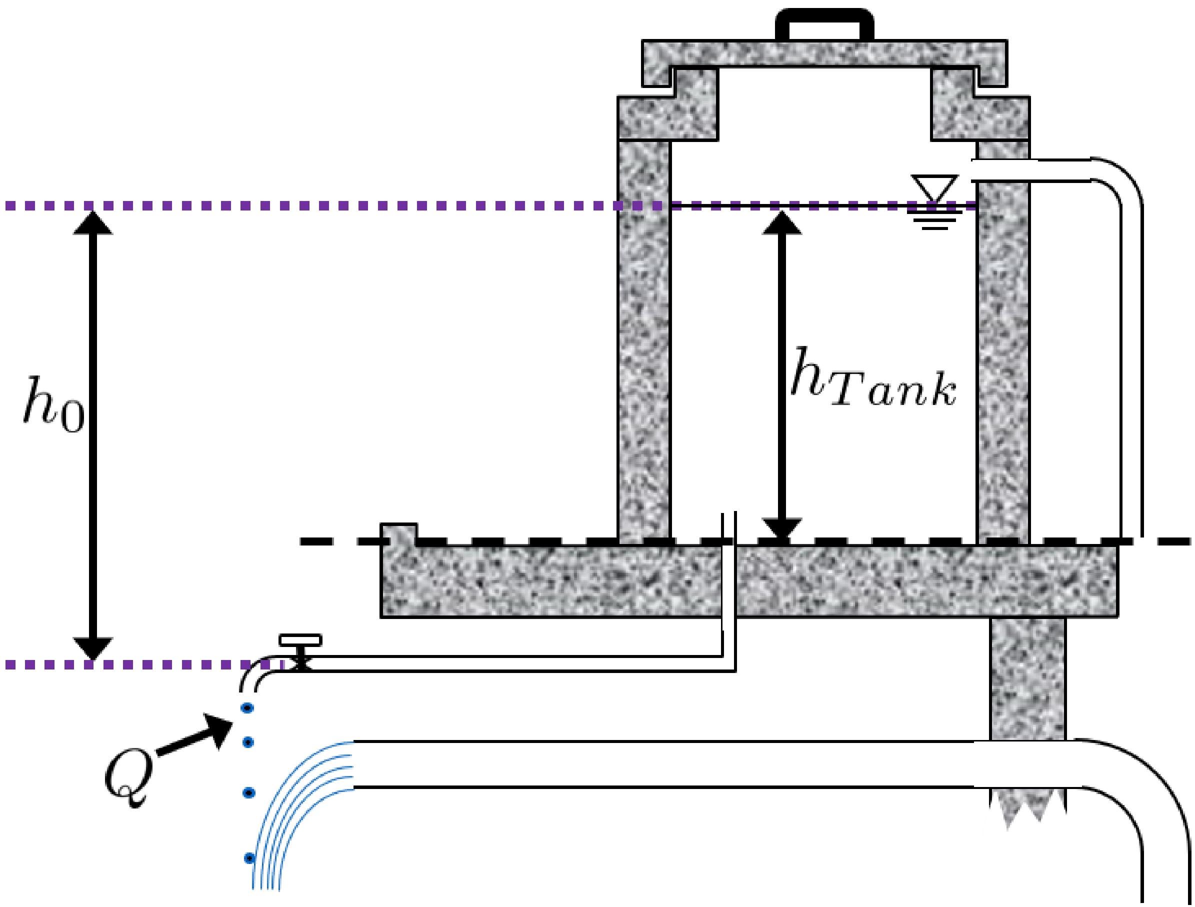


Fig. 10.2: Q_0 = initial flow rate of hypochlorite solution at time $t = 0$, t_{Design} = time it would take to drain the tank if flow was held constant at Q_0

10.2 $D(t)$ and $t(D)$ for Tank Drain Equation

This document contains the derivation of D_{Pipe}, which is the pipe diameter necessary to install in a drain system to entirely drain a tank in time t_{Drain}.

First, it is necessary to understand how AguaClara tank drains work and what they look like. Many tanks, including the flocculator and entrance tank, have a hole in their bottoms which are fitted with pipe couplings. During normal operation, these couplings have pipe stubs in them, and the pipe stubs are tall enough to go above the water level in the tank and not allow water to flow into the drain. When the pipe stub is removed, the water begins to flow out of the drain, as the image below indicates. The drain pipe consists of pipe and one elbow, shown in the image.

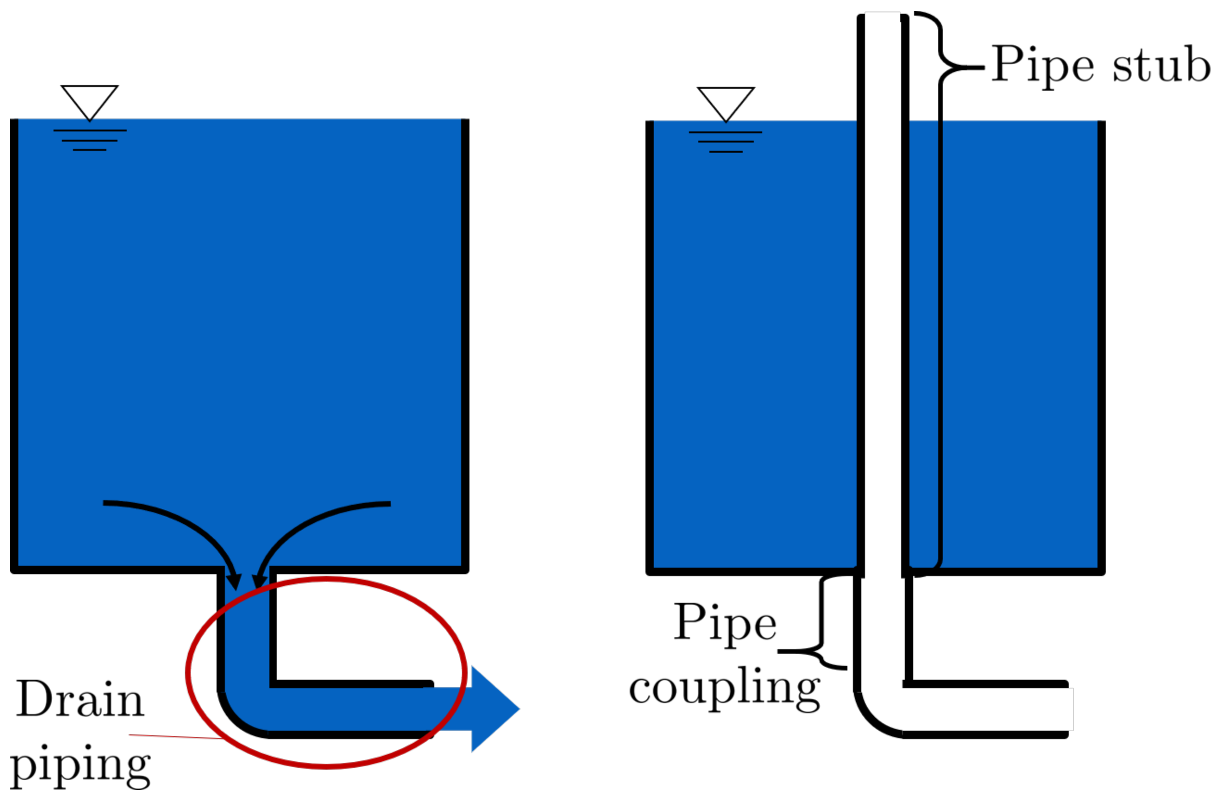


Fig. 10.3: This is AguaClaras alternatives to having valves.

While AguaClara sedimentation tanks use valves instead of pipe to begin the process of draining, the actual drain piping system is the same, pipe and an elbow. The equation that will soon be derived applies to both pipe stub and valve drains.

We will start the derivation from the following equation, which is found in an intermediate step from the $Q(t)$ *for a Tank with a Valve*. While this system does not have a valve, it has other sources of minor loss and therefore the equation is still valid.

$$\sqrt{h} = \sqrt{h_0} - t\frac{A_{Valve}}{2A_{Tank}}\sqrt{\frac{2g}{K}} \tag{10.16}$$

We need to make some adjustments to this equation before proceeding, to make it applicable for this new drain-system scenario. First, we want to assume that the tank has fully drained. Thus, $t = t_{Drain}$ and $h = 0$. Next, we recall that the tank drain is not actually a valve, but just pipe and an elbow, so $A_{Valve} = A_{Pipe}$. Additionally, there can be multiple points of minor loss in the drain system: the entrance from the tank into the drain pipe, the elbow, and potentially the exit of the water out of the drain pipe. When considering a sedimentation tank, the open valve required to begin drainage also has a minor loss associated with it. Therefore, it is necessary to substitute $\sum K$ for K With these substitutions, the equation becomes:

$$\sqrt{h_0} = t_{Drain} \frac{A_{Pipe}}{2A_{Tank}} \sqrt{\frac{2g}{\sum K}} \tag{10.17}$$

Now, with the knowledge that $A_{Pipe} = \frac{\pi D_{Pipe}^2}{4}$ and rearranging to solve for D_{Pipe}, we obtain the following equation:

$$D_{Pipe} = \sqrt{\frac{8A_{Tank}}{\pi t_{Drain}} \sqrt{\frac{h_0 \sum K}{2g}}} \tag{10.18}$$

To get the equation in terms of easily measureable tank parameters, we substitute $L_{Tank}W_{Tank}$ for A_{Tank}. To maintain consistency in variable names, we substitute H_{Tank} for h_0.

Note: By saying that $h_0 = H_{Tank}$, we are making the assumption that the pipe drain is at the same elevation as the bottom of the tank. The pipe drain is actually a little lower than the bottom of the tank, but that would make the tank drain faster than t_{Drain}, which is preferred. Therefore, we are designing a slight safety factor when we say that $h_0 = H_{Tank}$.

Finally, we arrive at the equation for drain pipe sizing:

$$D_{Pipe} = \sqrt{\frac{8L_{Tank}W_{Tank}}{\pi t_{Drain}}} \left(\frac{H_{Tank} \sum K}{2g} \right)^{\frac{1}{4}} \tag{10.19}$$

We can also easily rearrange to find the time required to drain a tank given a drain diameter:

$$t_{Drain} = \frac{8L_{Tank}W_{Tank}}{\pi D_{Pipe}^2} \sqrt{\frac{H_{Tank} \sum K}{2g}} \tag{10.20}$$

Such that the variables are as the appear in the image below.

10.3 Design Equations for the Linear Chemical Dose Controller (CDC)

This document will include the equation derivations required to design a CDC system. The most important restriction in this design process is maintaining linearity between head h and flow Q, which is the entire purpose of the CDC. Recall that major losses under laminar flow scale with Q and minor losses scale with Q^2 Since it is impossible to remove minor losses from the system entirely, we will simply try to make minor losses very small compared to major losses. The CDC does this by including dosing tube(s), which are long, straight tubes designed to generate a lot of major losses. There can be one tube or multiple, depending on the design conditions.

We will use the head loss trick that was introduced in the Fluids Review section. Therefore, the elevation difference between the water level in the constant head tank (CHT) and the end of the tube connected to the slider, Δh, is equal to the head loss between the two points, h_L. Thus, $\Delta h = h_L = h_e + h_f$.

Note: There are a lot of equations in this section, and they may quickly get confusing. They are color coded in an attempt to make them easier to follow. There are two final design equations: $\bar{v}_{Max}$ and math:*color{purple}{L_{Min}}*, and they will be written in purple text coloring to make them noticeable.

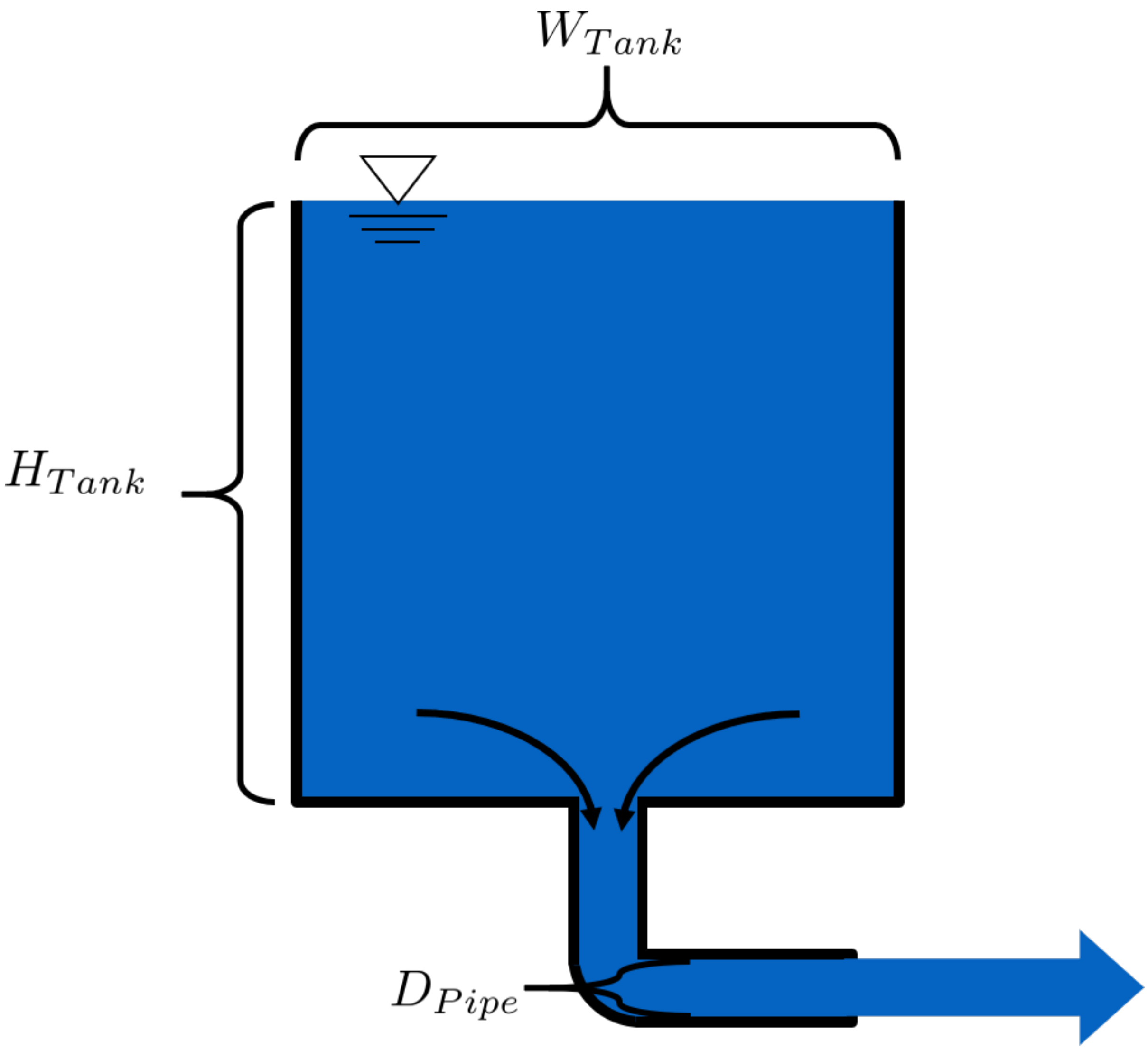


Fig. 10.4: L_{Tank} is the length of the tank which goes the page. K is the aggregate minor loss coefficient of the drain system.

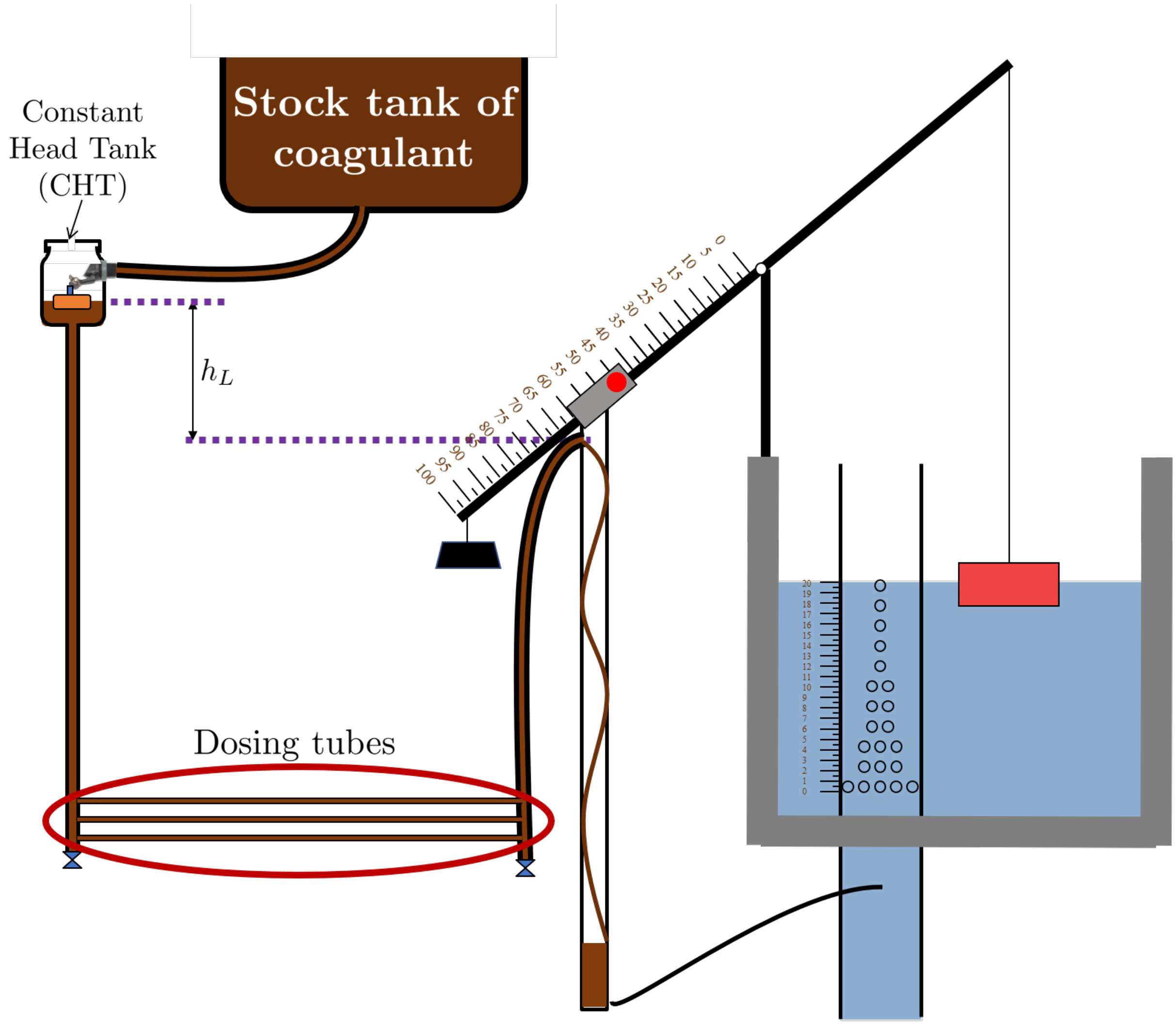


Fig. 10.5: Visual representation of CDC.

10.3.1 CDC Design Equation Derivation

Important: When designing the CDC, there are a few parameters which are picked and set initially, before applying any equations. These parameters are:

1. D = tube diameter. only certain tubing diameters are manufactured (like $\frac{x}{16}$ inch), so an array of available tube diameters is set initially.
2. $\sum K$ = sum of minor loss coefficients for the whole system. This is also set initially, it is usually 2.
3. $h_{L_{Max}}$ = maximum elevation difference between CHT water level and outlet of solution. This parameter is usually 20 cm.

We begin by defining the head loss through the system h_L, which is equivalent to defining the driving head Δh. Major losses will be coded as red.

$$h_{\rm f} = \frac{128\nu LQ}{g\pi D^4} \tag{10.21}$$

Such that:
ν = kinematic viscosity *of the solution going through the dosing tube(s)*. This is either coagulant or chlorine
Q = flow rate through the dosing tube(s)
L = length of the dosing tube(s)

Note: Tube(s) is used because there may be 1 or more dosing tubes depending on the particular design.

Minor losses are equal to:

$$h_e = \frac{8Q^2}{g\pi^2 D^4} \sum K \tag{10.22}$$

Therefore, the total head loss is a function of flow, and is shown in the following equation.

$$h_L(Q) = \frac{128\nu LQ}{g\pi D^4} + \frac{8Q^2}{g\pi^2 D^4} \sum K \tag{10.23}$$

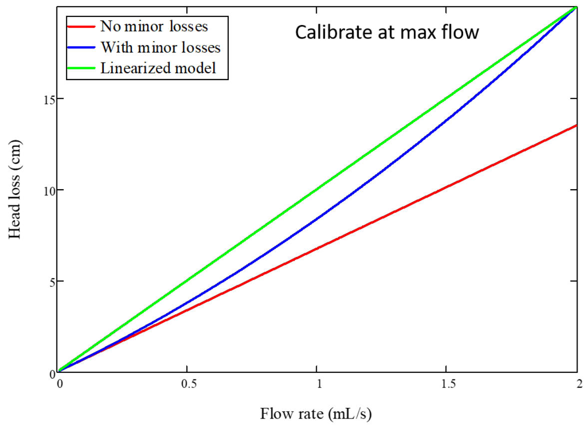
Blue will be used to reference *actual* head loss from now on. This is the same equation as above.

$$h_L(Q) = \left(\frac{128\nu L}{g\pi D^4} + \frac{8Q}{g\pi^2 D^4} \sum K \right) Q \tag{10.24}$$

This equation is not linear with respect to flow. We can make it linear by turning the variable Q in the $\frac{8Q}{g\pi^2 D^4} \sum K$ term into a constant. To do this, we pick a maximum flow rate of coagulant/chlorine through the dose controller, Q_{Max}, and put that into the term in place of Q. The term becomes $\frac{8Q_{Max}}{g\pi^2 D^4} \sum K$, and our linearized model of head loss, coded as green, becomes:

$$h_{L_{linear}}(Q) = \left(\frac{128\nu L}{g\pi D^4} + \frac{8Q_{Max}}{g\pi^2 D^4} \sum K \right) Q \tag{10.25}$$

Here is a plot of the three colored equations above. Our goal is to minimize the minor losses in the system; to bring the red and blue curves as close as possible to the green one.



$$h_f = \frac{128\nu LQ}{g\pi D^4}$$

$$h_L(Q) = \left(\frac{128\nu L}{g\pi D^4} + \frac{8Q}{g\pi^2 D^4}\sum K_e\right) Q$$

$$h_{L_{linear}}(Q) = \left(\frac{128\nu L}{g\pi D^4} + \frac{8Q_{max}}{g\pi^2 D^4}\sum K_e\right) Q$$

Fig. 10.6: MathCAD generated graph for linearity error analysis. TODO: make this in python

Designing for the error constraint, Π_{Error}

Important: The first step in the design is to make sure that major losses far exceed minor losses. This will result in an equation for the maximum velocity that can go through the dosing tube(s), $\bar{v}_{Max}$.

Minor losses will never be 0, so how much error in our linearity are we willing to accept? Lets define a new parameter, Π_{Error}, as the maximum amount of error we are willing to accept. We are ok with 10% error or less, so $\Pi_{Error} = 0.1$.

$$\Pi_{Error} = \frac{h_{L_{linear}} - h_L}{h_{L_{linear}}} = 1 - \frac{h_L}{h_{L_{linear}}} \tag{10.26}$$

$$1 - \Pi_{Error} = \frac{h_L}{h_{L_{linear}}} \tag{10.27}$$

Now we plug $h_L(Q)$ and $h_{L_{linear}}$ back into the equation for $1 - \Pi_{Error}$ and take the limit as $Q \to 0$, as that is when the relative difference between actual head loss and our linear model for head loss is the greatest.

$$1 - \Pi_{Error} = \frac{\left(\frac{128\nu L}{g\pi D^4} + \cancel{\frac{8Q}{g\pi^2 D^4}\sum K}\right) Q}{\left(\frac{128\nu L}{g\pi D^4} + \frac{8Q_{Max}}{g\pi^2 D^4}\sum K\right) Q} = \frac{\left(\frac{128\nu L}{g\pi D^4}\right)}{\left(\frac{128\nu L}{g\pi D^4} + \frac{8Q_{Max}}{g\pi^2 D^4}\sum K\right)} \tag{10.28}$$

The next steps are algebraic rearrangements to solve for L. This L describes the *minimum* length of dosing tube necessary to meet our error constraint at *maximum* flow. Thus, we will refer to it as $L_{Min,\Pi_{Error}}$.

$$(1 - \Pi_{Error})\frac{128\nu L}{g\pi D^4} + (1 - \Pi_{Error})\frac{8Q_{Max}}{g\pi^2 D^4}\sum K = \frac{128\nu L}{g\pi D^4} \tag{10.29}$$

$$-\Pi_{Error}\frac{128\nu L}{g\pi D^4} + (1 - \Pi_{Error})\frac{8Q_{Max}}{g\pi^2 D^4}\sum K = 0 \tag{10.30}$$

$$L = \left(\frac{1 - \Pi_{Error}}{\Pi_{Error}}\right)\frac{Q_{Max}}{16\nu\pi}\sum K \tag{10.31}$$

$$L_{Min,\Pi_{Error}} = L = \left(\frac{1 - \Pi_{Error}}{\Pi_{Error}}\right)\frac{Q_{Max}}{16\nu\pi}\sum K \tag{10.32}$$

Note that this equation is independent of head loss.

Unfortunately, both $L_{Min,\,\Pi_{Error}}$ and Q_{Max} are unknowns. We can plug this equation for $L_{Min,\,\Pi_{Error}}$ back into the head loss equation at maximum flow, which is $h_{L_{Max}} = \left(\frac{128\nu L Q_{Max}}{g\pi D^4} + \frac{8Q_{Max}^2}{g\pi^2 D^4}\sum K\right)$ and rearrange for Q_{Max} to get:

$$Q_{Max} = \frac{\pi D^2}{4}\sqrt{\frac{2h_{L_{Max}} g\Pi_{Error}}{\sum K}} \tag{10.33}$$

See also:

Function in aide_design `cdc.max_linear_flow(Diam, HeadlossCDC, Ratio_Error, KMinor)` Returns the maximum flow Q_{Max} that can go through a dosing tube will making sure that linearity between head loss and flow is conserved.

From this equation for Q_{Max}, we can get to our first design equation, $\bar{v}_{Max}$ by using the continuity equation $\bar{v}_{Max} = \frac{Q_{Max}}{\frac{\pi D^2}{4}}$

$$\bar{v}_{Max} = \sqrt{\frac{2h_L g\Pi_{Error}}{\sum K}} \tag{10.34}$$

Designing for the proper amount of head loss, $h_{L_{Max}}$

Important: The second step in the design is to make sure that the maximum head loss corresponds to the maximum flow of chemicals. This will result in an equation for the length of the dosing tube(s), L_{Min}.

We previously derived an equation for the minimum length of the dosing tube(s), $L_{Min,\,\Pi_{Error}}$, which was the minimum length needed to ensure that our linearity constraint was met. This equation is shown again below, in red:

$$L_{Min,\,\Pi_{Error}} = \left(\frac{1-\Pi_{Error}}{\Pi_{Error}}\right)\frac{Q_{Max}}{16\nu\pi}\sum K \tag{10.35}$$

This equation does not, however, account for getting to the proper amount of head loss. If we were to use this equation to design the dosing tubes, we might not end up with the proper amount of flow Q_{Max} at the maximum head loss h_{LMax}. So we need to double check to make sure that we get our desired head loss.

First, consider the head loss at maximum flow that was used to get the equation for Q_{Max}:

$$h_{L_{Max}} = \left(\frac{128\nu L Q_{Max}}{g\pi D^4} + \frac{8Q_{Max}^2}{g\pi^2 D^4}\sum K\right) \tag{10.36}$$

Now that we know all of the parameters in this equation except for L, we can solve the equation for L. This the *shortest* tube that generates our required head loss, $h_{L_{Max}}$.

$$L_{Min,\,headloss} = L = \left(\frac{g h_{L_{Max}}\pi D^4}{128\nu Q_{Max}} - \frac{Q_{Max}}{16\pi\nu}\sum K\right) \tag{10.37}$$

See also:

Function in aide_design: `cdc._length_cdc_tube_array(FlowPlant, ConcDoseMax, ConcStock, DiamTubeAvail, HeadlossCDC, temp, en_chem, KMinor)` Returns L_{Min}, takes in the flow rate input of *plant design flow rate*.

See also:

Function in aide_design: `cdc._len_tube(Flow, Diam, HeadLoss, conc_chem, temp, en_chem, KMinor)` Returns L_{Min}, takes in the flow rate input of *max flow rate through the dosing tube(s)*.

If you decrease the max flow Q_{Max} and hold $h_{L_{Max}}$ constant, $L_{Min,\,headloss}$ becomes larger. This means that a CDC system for a plant of 40 $\frac{L}{s}$ must be different than one for a plant of 20 $\frac{L}{s}$. If we want to maintain the same head loss at maximum flow in both plants, then the dosing tube(s) will need to be a lot longer for the 20 $\frac{L}{s}$ plant.

To visualize the distinction between $L_{Min,\,\Pi_{Error}}$ and math:*color{green}{ L_{Min, , head loss}}*, see the following plot. $L_{Min,\,headloss}$ is discontinuous because it takes in the smallest allowable tube diameter as an input. As the chemical flow rate through the dosing tube(s) decreases, the dosing tube diameter does as well. Whenever you see a jump in the green points, that means the tubing diameter has changed.

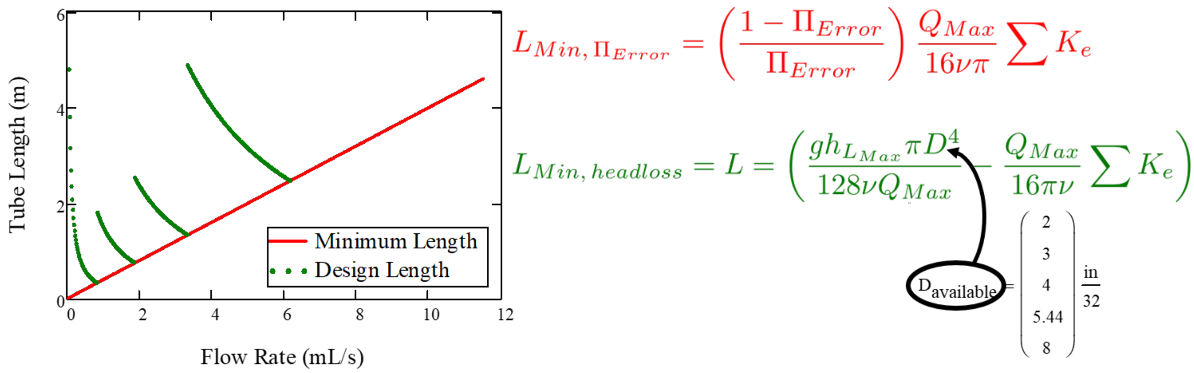


Fig. 10.7: CDC length modeling in MathCAD.

As you can see, the head loss constraint is more limiting than the linearity constraint when designing for tube length. Therefore, the design equation for tube length is the one which accounts for head loss. This is the second and final design equation for designing the CDC:

$$L_{Min} = L_{Min,\,headloss} = \left(\frac{g h_{L_{Max}} \pi D^4}{128 \nu Q_{Max}} - \frac{Q_{Max}}{16 \pi \nu} \sum K\right) \tag{10.38}$$

The equations for $\bar{v}_{Max}$ and L_{Min} are the only ones you **need** to manually design a CDC.

CDC Dosing Tube(s) Diameter D_{Min} Plots

Below are equations which also govern the CDC and greatly aid in understanding the physics behind it, but are not strictly necessary in design.

By rearranging $Q_{Max} = \frac{\pi D^2}{4}\sqrt{\frac{2 h_L g \Pi_{Error}}{\sum K}}$, we can solve for D to get the *minimum* diameter we can use assuming the shortest tube possible that meets the error constraint, $L_{Min,\,\Pi_{Error}}$. If we use a diameter smaller than $D_{Min,\,\Pi_{Error}}$, we will not be able to simultaneously reach Q_{Max} and meet the error constraint Π_{Error}.

$$D_{Min,\,\Pi_{Error}} = \left[\frac{8 Q_{Max}^2 \sum K}{\Pi_{Error} h_l g \pi^2}\right]^{\frac{1}{4}} \tag{10.39}$$

We can also find the minimum diameter needed to guarantee laminar flow, which is another critical condition in the CDC design. We can do this by combining the equation for Reynolds number at the maximum Re for laminar flow, $\mathrm{Re}_{Max} = 2100$ with the continuity equation at maximum flow:

$$\mathrm{Re}_{Max} = \frac{\bar{v}_{Max} D}{\nu} \tag{10.40}$$

$$\bar{v}_{Max} = \frac{4 Q_{Max}}{\pi D^2} \tag{10.41}$$

To get:

$$D_{Min,\,Laminar} = \frac{4Q_{Max}}{\pi\nu\mathrm{Re}_{Max}} \tag{10.42}$$

Combined with the discrete amount of tubing sizes (shown in dark green), we can create a graph of the three diameter constraints:

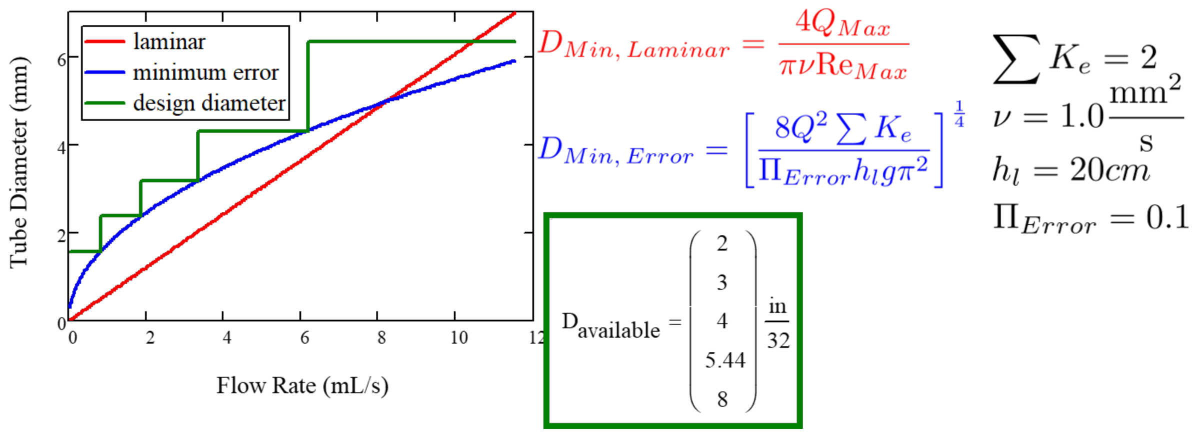


Fig. 10.8: CDC diameter modeling in MathCAD.

CHAPTER ELEVEN

RAPID MIX INTRODUCTION

Before we introduce rapid mix, we need to first discuss our strategy for taking a water source that is contaminated with dissolved or suspended contaminants and

Rapid mix is the term commonly used to describe the processes that occur between the coagulant addition to the raw water and the flocculation process. The processes that occur are not well understood and thus design guidelines have been empirical.

In summary, little is known about rapid mix, much less any sensitivity to scale. However, the models and data reviewed suggest the need to be on the lookout for certain effects. From what is presently known, it can be speculated that since coagulant precipitation is sensitive to both micro- and macro-mixing, scale-up must consider not only energy dissipation rate, but also the reaction injection point and the contacting method. - Mixing in Coagulation and Flocculation 1991 page 292.

Although the processes have not been well characterized, the energy that is invested for rapid miz processes is significant. In many cases the amount of energy used isnt practical for gravity powered water treatment plants. The high energy consumption of rapid mix units has led some municipal water treatment plant operators to experiment with turning off rapid mix units. They have found that at least under some conditions there is no indication that the energy used in rapid mix improved plant performance. Thus there is a need to understand the physical and chemical processes that occur when a concentrated liquid coagulant is added to water.

Rapid mix sets the stage for aggregation of both suspended particles and dissolved substances. Particle and dissolve substance aggregation is mediated by coagulant nanoparticles. The nanoparticles attach to raw water particles as well as to some dissolved species. After the nanoparticles have been mixed with the raw water and have attached to raw water particles the next process, flocculation, can begin. Flocculation is the process of producing collisions between particles to create flocs (aggregates of particles).

Nanoparticle application includes multiple steps that must occur before the raw water particles can begin to aggregate. The sticky nanoparticles can be aluminum (Al^{+3}) or iron (Fe^{+3}) based and in either case the nanoparticles are formed from precipitated hydroxide species $(Al(OH)_3)$ or $(Fe(OH)_3)$. The series of events that are contained in the broad designation of rapid mix are:

1. Liquid coagulant stock solution with a low pH is injected into the raw water
2. Fluid Mixing: Turbulent eddies randomize the fluids (but dont blend them)
   (a) Large scale eddies mix the coagulant with the raw water by creating large fluid deformations. This stretching and turning of the raw water and coagulant is analogous to shuffling a deck of cards. The cards are randomized, but the cards maintain their identity. The original liquids retain their chemical composition. This step must be completed before any flow splitting for parallel treatment trains.
   (b) Turbulent eddies disintegrate into smaller and smaller eddies.
   (c) At a very small scale (Inner viscous length scale) viscosity becomes significant and the kinetic energy of the eddies begins to be converted to heat by viscosity.
3. The coagulant is blended with the raw water by molecular diffusion

4. The higher pH of the raw water causes the coagulant to begin to precipitate as $Al_{12}AlO_4(OH)_{24}(H_2O)_{12}^{7+}$, an aluminum, Al, nanoparticle.
5. The precipitating Al_{13} molecules aggregates with other nearby Al_{13} molecules to form aluminum hydroxide nanoparticles. It is also possible that the nanoparticles are already formed in the coagulant stock suspension. Polyaluminum chloride stock solutions turn white in about a year at room temperature and this suggests that nanoparticles form in the stock solution.
6. The Al nanoparticles attach to other dissolved species and suspended particles.
7. Molecular diffusion causes some dissolved species and Al nanoparticles to aggregate.
8. Fluid shear and molecular diffusion cause Al nanoparticles with attached formerly dissolved species to collide with inorganic particles (such as clay) and organic particles (such as viruses, bacteria, and protozoans).

These multiple steps cover a wide range of length scales and it is not clear at the onset which processes might be the rate limiting steps. We will develop time scale estimates for several of these steps to help identify which processes will likely require the most attention to design. Many of these transport processes are presumed to occur in parallel. Fig. 11.1 shows the range of length scales

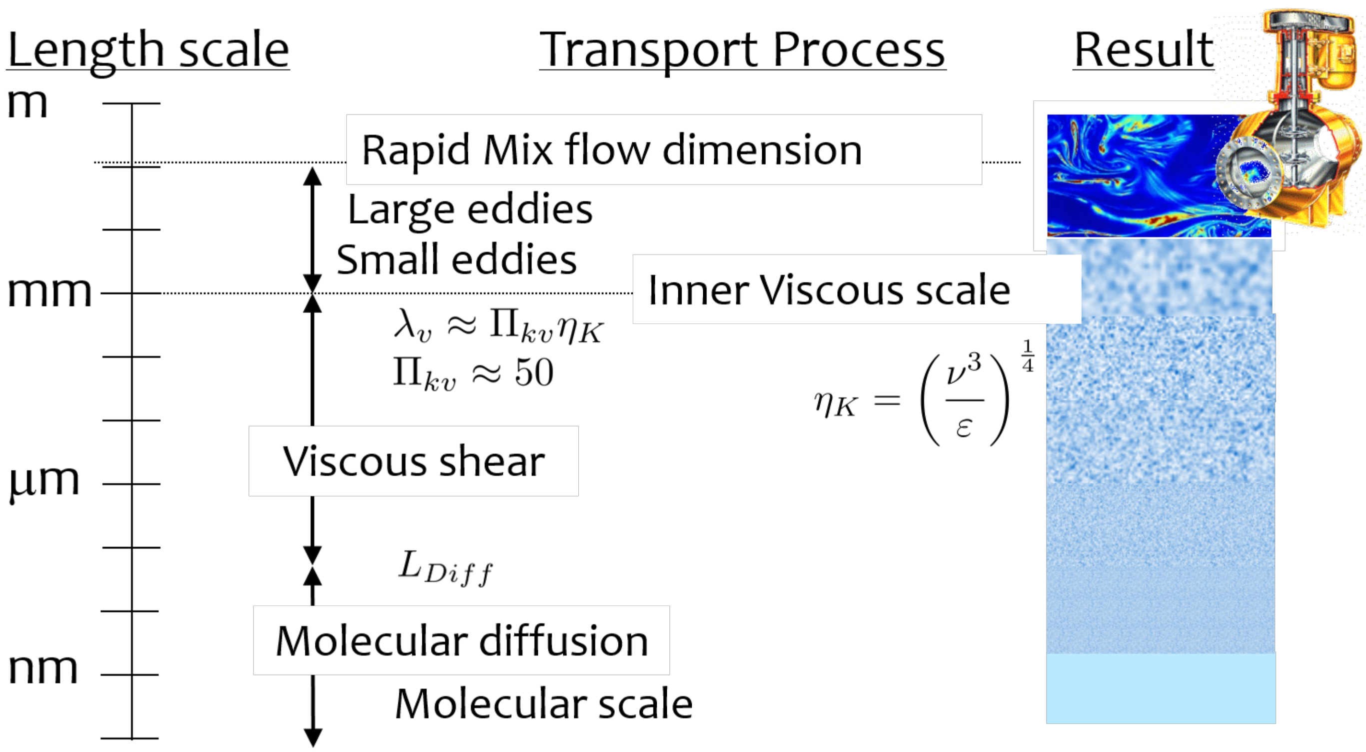


Fig. 11.1: Transport of coagulant nanoparticles occurs over length scales ranging from meter to a fraction of a nanometer.

11.1 Fluid mixing

Fluid mixing is the process by which large scale eddies distribute packets of the coagulant stock throughout the raw water. The term Rapid mix is probably best used to describe this process. Traditional methods of achieving this fluid mixing include

Fig. 11.2: Backmix: a mechanical rapid mixer that has a relatively long residence time in a completely mixed flow reactor.

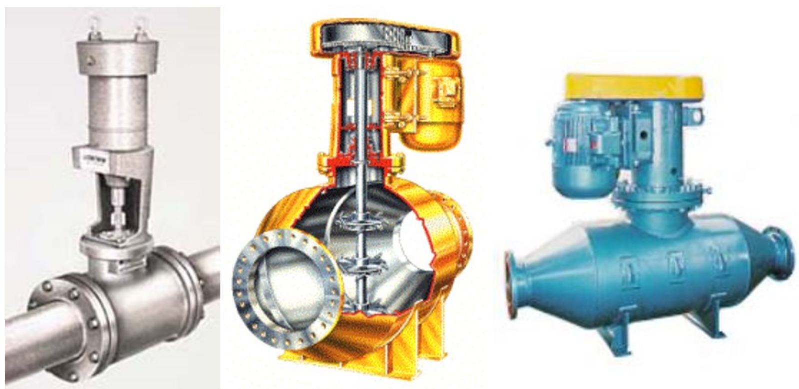


Fig. 11.3: Inline: a mechanical rapid mixer that has a short residence time in a completely mixed flow reactor that is often built into a pipe.

Fig. 11.4: Hydraulic jump: a hydraulic rapid mixer uses the flow expansion downstream from supercritical open channel flow.

The hydraulic jump in Fig. 11.4 uses a flow expanion to generate mixing in an open channel and that suggests that a flow expansion could also be used to generate mixing in a closed conduit. AguaClara rapid mix units consiste of an orifice in the bottom of the Linear Flow Orifice Meter where the water enters the flocculator.

11.2 Chemistry of coagulant nanoparticles

11.2.1 pH Effects of Adding Coagulant

11.2.2 Buffering Capacity of Natural Waters

11.2.3 pH Range for Precipitation of Coagulant Nanoparticles

11.2.4 pH Adjustment in Water Treatment Plants

11.3 Coagulant Nanoparticle Interactions

11.3.1 Dissolved Organic Matter

11.3.2 Suspended Solids

11.3.3 Rate Estimates for Coagulant Nanoparticle Transport to Suspended Solids

11.4 Energy Dissipation Rate, Velocity Gradient, and Mixing

In addition to the general fluids review in the previous chapter, there are a few extra fluid dynamics concepts that are important to know in order to understand drinking water treatment and AguaClaras approach to it. These concepts are primarily focused on the relationships between: * Turbulence * Viscosity * Shear * Velocity Gradients (G), which serve as a measure of fluid deformation * Energy Dissipation Rate (EDR, ε)

Knowledge of these concepts and how they interact is critical to understand rapid mix, flocculation, and disinfection. These concepts and their interactions first become relevant in rapid mix, the step in which the coagulant gets added to the raw water.

The two concepts that were not covered in the previous chapter, *Review: Fluid Mechanics*, are velocity gradient G and energy dissipation rate ε. While these will be very thoroughly described over the course of this introduction, a brief and simple explanation is included to help get the ball rolling.

11.4.1 Understanding G and ε

G, or velocity gradient, is a measure of fluid deformation. It is defined by how quickly one point of water along one streamline moves in comparison to another point on another streamline (v_A compared to v_B, for example), taking into account the distance between the streamlines, Δh. A visual example of a velocity gradient is shown in the image below:

Note on terminology: Fluid deformation is equivalent to velocity gradient, and the two terms can be used interchangeably. They are different ways of thinking about the same concept. Thus, G is the measure of both terms.

ε, or energy dissipation rate, is the rate that the kinetic energy of the fluid is being converted to heat. EDR is a very useful concept because the last step of converting kinetic energy into heat is accomplished by viscosity (ν). This

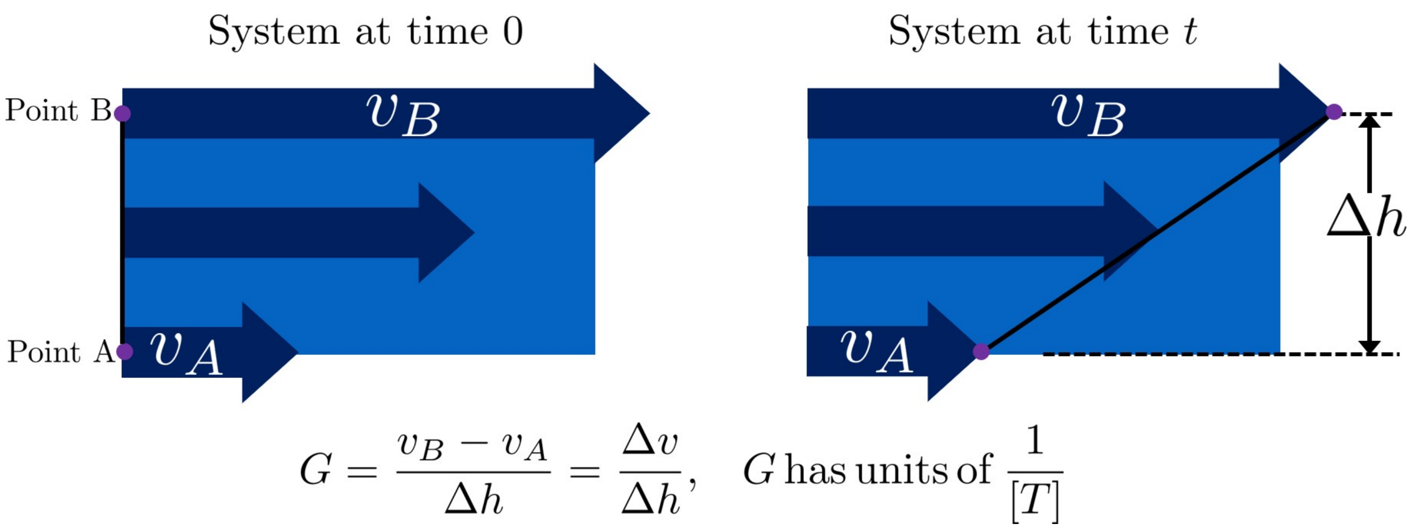


Fig. 11.5: Velocity gradients cause relative velocities of fluid elements. Those relative velocities form the basis of particle collisions that are essential for the flocculation process.

kinetic energy being dissipated by viscosity is the energy associated with velocity gradients (G). Thus, through EDR there is a direct connection between ν and G. This connection will be further covered later on in this introduction.

As mentioned above, EDR and velocity gradients play an important role in mixing and in causing suspended particles to collide with each other, both of which are important topics in flocculation. Their use is not limited to flocculation, they are also helpful in understanding failure modes of plate settlers and terminal head loss of sand filters

We will begin by defining the concept of energy dissipation rate for a control volume. In a control volume that does not include pumps, turbines or other external energy sources or sinks, the mechanical energy lost is indicated by a change in elevation and quantified as gh_L. That mechanical energy is lost in the time that the fluid is in the control volume, θ.

$$\bar{\varepsilon}\theta = gh_L \tag{11.1}$$

This equation simply states that the average rate of energy dissipation times the time over which that dissipation occurs is equal to the total lost mechanical energy. The dimensions of ε are:

$$\varepsilon = \frac{[m^3]}{[s^3]} = \frac{\text{W}}{\text{kg}} \tag{11.2}$$

These dimensions can be understood as a velocity squared per time, otherwise known as a rate of kinetic energy loss (recall that kinetic energy is $\text{Ke} = \frac{\bar{v}^2}{2g}$, or $\text{Ke} \propto \bar{v}^2$), or as power per unit mass, which would be $\frac{\text{W}}{\text{kg}}$.

Velocity gradients are central to flocculation because they cause the deformation of the fluid, and this results in particle collisions. Consider a real-world example via the image below: if everyone on a sidewalk is walking in the same direction at exactly the same velocity, then there will never be any collisions between people (top). If, however, people at one side of the sidewalk stand still and people walk progressively faster as a function of how far they are away from the zero velocity side of the sidewalk, then there will be many collisions between the pedestrians. Indeed, the rate of collisions is proportional to the velocity gradient.

11.5 Common Flow Geometries that Dissipate Energy

Water treatment plants at research and municipal scales deploy a wide range of flow geometries. The following list includes the flow geometries that are commonly used for mixing processes.

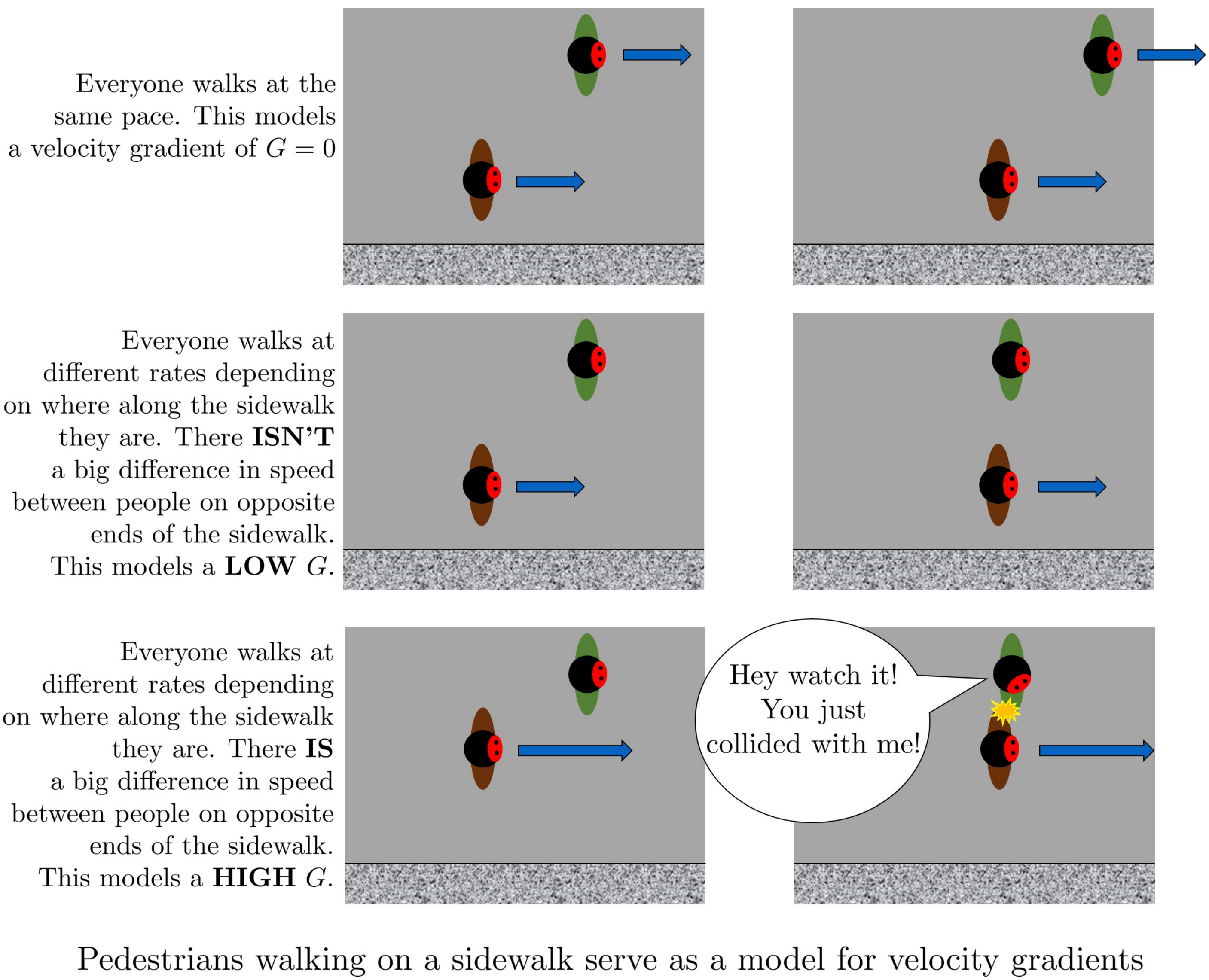


Fig. 11.6: Pedestrians walking on a sidewalk serve as a model for velocity gradients.

- Straight pipe (wall shear) - [uncommon, but included for completeness]
- Coiled tube (wall shear and expansions) - [research scale mixing]
- Series of expansions (expansions) - [hydraulic flocculators]
- Mechanical mixing - [mechanical rapid mix and flocculators]
- Between flat plates (wall shear) - [plate settlers]
- Round jet - (expansion) - [hydraulic rapid mix]
- Plane jet - (expansion) - [inlet into sedimentation tank]
- Behind a flat plate - (expansion) - [mechanical mixers]

The following tables can serve as a convenient reference to the equations describing head loss, energy dissipation rates, and velocity gradients in various flow geometries that are commonly encountered in water treatment plants. The *Rapid Mix Derivations* are available as a reference.

Table 11.1: Table of equations for control volume averaged values of head loss, energy dissipation rate, and the Camp-Stein velocity gradient.

Geometry	h_L	Energy dissipation rate	$G_{CS}(barv)$	$G_{CS}(Q)$
Straight pipe	$h_{\rm f} = {\rm f}\frac{L}{D}\frac{\bar v^2}{2g}$	$\bar\varepsilon = \frac{\rm f}{2}\frac{\bar v^3}{D}$	$G_{CS} = \left(\frac{\rm f}{2\nu}\frac{\bar v^3}{D}\right)^{\frac{1}{2}}$	$G_{CS} = \left(\frac{32{\rm f}}{\pi^3\nu}\frac{Q^3}{D^7}\right)^{\frac{1}{2}}$
Straight pipe laminar	$h_{\rm f} = \frac{32\nu L\bar v}{gD^2}$	$\bar\varepsilon = 32\nu\left(\frac{\bar v}{D}\right)^2$	$G_{CS} = 4\sqrt{2}\frac{\bar v}{D}$	$G_{CS} = \frac{16\sqrt{2}}{\pi}\frac{Q}{D^3}$
Parallel plates laminar	$h_{\rm f} = 12\frac{\nu L\bar v}{gS^2}$	$\bar\varepsilon = 12\nu\left(\frac{\bar v}{S}\right)^2$	$G_{CS} = 2\sqrt{3}\frac{\bar v}{S}$	•
Coiled tube laminar	$h_{L_{coil}} = \frac{32\nu L\bar v}{gD^2}\left[1+0.033\left(log_{10}De\right)^4\right]$	$\bar\varepsilon = 32\nu\left(\frac{\bar v}{D}\right)^2\left[1+0.033\left(log_{10}De\right)^4\right]$	$G_{CS_{coil}} = 4\sqrt{2}\frac{\bar v}{D}\left[1+0.033\left(log_{10}De\right)^4\right]^{\frac{1}{2}}$	•
Expansions	$h_e = K\frac{\bar v_{out}^2}{2g}$	$\bar\varepsilon = K\frac{\bar v_{out}^3}{2H}$	$G_{CS} = \bar v_{out}\sqrt{\frac{K\bar v_{out}}{2H\nu}}$	•

The equations used to convert between columns in the table above are:

$$\bar\varepsilon = \frac{gh_{\rm L}}{\theta} \qquad G_{CS} = \sqrt{\frac{\bar\varepsilon}{\nu}} \qquad \bar v = \frac{4Q}{\pi D} \tag{11.3}$$

Note that the velocity gradient is independent of viscosity (and hence temperature) for laminar flow. This is because the total amount of fluid deformation is simply based on geometry. The no slip condition, the diameter, and the length of the flow passage set the total fluid deformation. Of course, if temperature decreases and viscosity increases the amount of energy required to push the fluid through the flow passage will increase (head loss is proportional to viscosity for laminar flow).

For turbulent flow and for flow expansions the amount of fluid deformation decreases as the viscosity increases and the total energy required to send the flow through the reactor is almost independent of the viscosity. The almost is because for wall shear under turbulent conditions there is a small effect of viscosity that is buried inside the friction factor.

Table 11.2: Table of equations for maximum (wall) energy dissipation rates and wall velocity gradients.

Geometry	Energy dissipation rate at the wall	Velocity gradient at the wall
Straight pipe	$\varepsilon_{wall} = \frac{1}{\nu}\left(\mathrm{f}\frac{\bar v^2}{8}\right)^2$	$G_{wall} = \mathrm{f}\frac{\bar v^2}{8\nu}$
Straight pipe laminar	$\varepsilon_{wall} = \left(\frac{8\bar v}{D}\right)^2\nu$	$G_{wall} = \frac{8\bar v}{D}$
parallel plates	$\varepsilon_{wall} = 36\left(\frac{\bar v}{S}\right)^2\nu$	$G_{wall} = \frac{6\bar v}{S}$
Coiled pipe	•	$G_{CS_{wall_{coil}}} = \mathrm{f}\left[1 + 0.033\left(log_{10}De\right)^4\right]\frac{\bar v^2}{8\nu}$

Table 11.3: Equations for maximum energy dissipation rates and velocity gradients for flow expansions.

Geometry	Pi_{Jet}	Maximum energy dissipation rate	Maximum velocity gradient
Round jet	0.08	$\varepsilon_{Max} = \Pi_{JetRound}\frac{\bar v_{Jet}^3}{D_{Jet}}$	$G_{Max} = \bar v_{Jet}\sqrt{\frac{\Pi_{RoundJet}\bar v_{Jet}}{\nu D_{Jet}}}$
Plane jet	0.0124	$\varepsilon_{Max} = \Pi_{JetPlane}\frac{\bar v_{Jet}^3}{S_{Jet}}$	$G_{Max} = \bar v_{Jet}\sqrt{\frac{\Pi_{JetPlane}\bar v_{Jet}}{\nu S_{Jet}}}$
Behind a flat plate	0.04	$\varepsilon_{Max} = \Pi_{Plate}\frac{\bar v^3}{W_{Plate}}$	$G_{Max} = \bar v\sqrt{\frac{\Pi_{Plate}\bar v}{\nu W_{Plate}}}$

For mechanical mixing where an impeller or other stirring device is adding shaft work to a control volume we have

$$\bar\varepsilon = \frac{P}{m} = \frac{P}{\rho \rlap{-}V} \tag{11.4}$$

where
P = power input into the control volume
m = mass of fluid in the control volume
$\rlap{-}V$ = volume of the control volume
ρ = density of the fluid

The Camp-Stein velocity gradient for a mechanically mixed reactor is

$$G_{CS} = \sqrt{\frac{P}{\rho\nu\rlap{-}V}} \tag{11.5}$$

CHAPTER

TWELVE

RAPID MIX DESIGN

12.1 Length and time scales for each process

Lets begin by describing the coagulant injection for a 60 L/s plant. We will use a *linear flow orifice meter* with 20 cm of head loss.

```
""" importing """
from aide_design.play import*
from aguaclara_research.play import*
import aguaclara_research.floc_model as fm
import matplotlib.pyplot as plt
from matplotlib.ticker import FormatStrFormatter
imagepath = 'AguaClara Water Treatment Plant Design/Rapid Mix/Images/'

Q_plant = 60 * u.L/u.s
HL_LFOM = 20 * u.cm
Pi_LFOM_safety = 1.2
SDR_LFOM = 26
from aide_design.unit_process_design import lfom as lfom
ND_LFOM = lfom.nom_diam_lfom_pipe(Q_plant,HL_LFOM)
print(ND_LFOM, '(',ND_LFOM.to(u.cm), ')')

L_flow = pipe.ID_SDR(ND_LFOM,SDR_LFOM)
L_flow
v_lfom = (Q_plant/pc.area_circle(pipe.ID_SDR(ND_LFOM,SDR_LFOM))).to_base_units()
print(v_lfom)
```

The LFOM requires a 16 inch diameter pipe.

10% velocity rule

```
v_lfom = (Q_plant/pc.area_circle(pipe.ID_SDR(ND_LFOM,SDR_LFOM))).to_base_units()
print(v_lfom)
t_large_scale_mix = (pipe.ID_SDR(ND_LFOM,SDR_LFOM)/(0.1*v_lfom)).to_base_units()
print(t_large_scale_mix)
```

12.1.1 Example problem: Energy dissipation rate in a straight pipe

A water treatment plant that is treating 120 L/s of water injects the coagulant into the middle of the pipe that delivers the raw water to the plant and then splits the flow into 2 parallel treatment trains for subsequent flocculation. The pipe is PVC 16 inch nominal pipe diameter SDR 26. The water temperature is $0°C$. Estimate the minimum distance between the injection point and the flow split.

Solution scheme 1) Calculate the friction factor

```
T_water=0*u.degC
Pipe_roughness = mat.PIPE_ROUGH_PVC
Pipe_roughness
Nu_water = pc.viscosity_kinematic(T_water)
Q_pipe = 120 * u.L/u.s
ND_pipe = 16*u.inch
SDR_pipe = 26
ID_pipe = pipe.ID_SDR(ND_pipe,SDR_pipe)
f_pipe = pc.fric(Q_pipe,ID_pipe,Nu_water,Pipe_roughness)
N_pipe_diameters = (2/f_pipe)**(1/3)
N_pipe_diameters
"""The minimum length for mixing is thus"""
L_mixing = ID_pipe*N_pipe_diameters
print('The minimum distance required for mixing across the diameter of the pipe is ',
↪L_mixing.to_base_units())
```

The previous analysis provides a minimum distance for sufficient mixing so that equal mass flux of coagulant will end up in both treatment trains. This assumes that the coagulant was injected in the pipe centerline. Injection at the wall of the pipe is a poor practice and would require many more pipe diameters because it takes significant time for the coagulant to be mixed out of the slower fluid at the wall. The time required for mixing at the scale of the flow in the plant is thus accomplished in a few seconds. This ends up being the fastest part of the transport of the coagulant nanoparticles on their way to attachment to the clay particles. Next we will determine a typical flow rate of coagulant. **Aluminum** concentrations for polyaluminum chloride (PACl) typically range from 1 to 10 mg/L. The maximum PACl stock solution concentration is about 70 g/L as **Al**.

```
C_PACl_stock = 70 *u.g/u.L
C_PACl_dose_max = 10 * u.mg/u.L
Q_PACl_max = (Q_plant*C_PACl_dose_max/C_PACl_stock).to(u.mL/u.s)
print(Q_PACl_max)
```

We can estimate the diameter of the injection port by setting the kinetic energy loss where the coagulant is injected into the main flow to be 10 cm. The amount of energy we invest in injecting the coagulant into the raw water is a compromise between having to raise the entire chemical feed system including the stock tanks to increase the potential energy and a goal of not having pressure fluctuations inside the LFOM pipe cause flow oscillations in the chemical dosing tube. Thus our goal is to have the kinetic energy at the injection point be large compared with the expected pressure fluctuations in the LFOM.

```
HL_Coag_injection = 10 * u.cm
v_Coag_injection = ((2 * u.gravity * HL_Coag_injection)**0.5).to(u.m/u.s)
print(v_Coag_injection)
D_Coag_injection_min = pc.diam_circle(Q_PACl_max/v_Coag_injection)
print(D_Coag_injection_min.to(u.mm))
```

Orifice Diameter to obtain Target Mixing

$$A_{Orifice}\Pi_{vc} = A_{Jet} \tag{12.1}$$

$$D_{Orifice}\sqrt{\Pi_{vc}} = D_{Jet} \tag{12.2}$$

$$\varepsilon_{Max} \cong \frac{\left(\Pi_{JetRound}\frac{4Q}{\pi D_{Jet}^2}\right)^3}{D_{Jet}} \tag{12.3}$$

$$D_{Orifice} \cong \left(\frac{4Q\Pi_{JetRound}}{\varepsilon_{Max}^{\frac{1}{3}}\pi}\right)^{\frac{3}{7}} \frac{1}{\sqrt{\Pi_{vc}}} \tag{12.4}$$

Off-slide

$$\varepsilon_{Max} \cong \frac{\left(\Pi_{Jet}\frac{4Q_{Jet}}{\pi}\right)^3}{D_{Orifice}^7\sqrt{\Pi_{vc}^7}} \tag{12.5}$$

Rapid Mix Head Loss

$$D_{Orifice} \cong \left(\frac{4Q\Pi_{JetRound}}{\varepsilon_{Max}^{\frac{1}{3}}\pi}\right)^{\frac{3}{7}} \tag{12.6}$$

$$\bar{v}_{Jet} \cong \frac{\left(D_{Jet}\,\varepsilon_{Max}\right)^{\frac{1}{3}}}{\Pi_{JetRound}} \tag{12.7}$$

$$h_e = \frac{\left(D_{Jet}\,\varepsilon_{Max}\right)^{\frac{2}{3}}}{2g\Pi_{JetRound}^2} \tag{12.8}$$

$$h_e = \frac{\left(\frac{4\Pi_{JetRound}Q\varepsilon_{Max}^2}{\pi}\right)^{\frac{2}{7}}}{2g\Pi_{JetRound}^2} \tag{12.9}$$

Off-slide

$$Q = \frac{D_{Jet}^{\frac{7}{3}}\pi\varepsilon_{Max}^{\frac{1}{3}}}{4\Pi_{Jet}} \tag{12.10}$$

CHAPTER

THIRTEEN

RAPID MIX DERIVATIONS

13.1 Eddy turnover time

Eddy turnover time, t_{eddy}, is the time it takes for the eddy to travel a distance equal to its length-scale. We assume that the energy of the large eddy is dissipated into smaller length scales in the time t_{eddy}:

$$t_{eddy} \approx \frac{L_{eddy}}{v_{eddy}} \tag{13.1}$$

The rate of energy loss to smaller scales is

$$\bar\varepsilon \approx \frac{v_{eddy}^2}{t_{eddy}} \tag{13.2}$$

Combining the two equations

$$\bar\varepsilon \approx \frac{v_{eddy}^3}{L_{eddy}} \tag{13.3}$$

We can use this equation to estimate the eddy velocity given an energy dissipation rate.

$$v_{eddy} \approx \left(\bar\varepsilon\, L_{eddy}\right)^{\frac{1}{3}} \tag{13.4}$$

Now we can solve for the eddy turnover time which is a measure of the mixing time at the eddy scale.

$$t_{eddy} \approx \frac{L_{eddy}}{\left(\bar\varepsilon\, L_{eddy}\right)^{\frac{1}{3}}} \approx \left(\frac{L_{eddy}^2}{\bar\varepsilon}\right)^{\frac{1}{3}} \tag{13.5}$$

$t_{coagulant,\, application}$

13.2 Collision Rates

$$\rlap{-}V_{\text{Cleared}} \approx \pi d_{Clay} L_{Diff_{NC}} v_r t_c \tag{13.6}$$

Where $\rlap{-}V_{Occupied} = \Lambda_{Clay}^3$. Solve for t_c:

$$t_c = \frac{\Lambda_{NC}^3}{\pi d_{Clay} L_{Diff_{NC}} v_r} \tag{13.7}$$

This is the average time for a clay particle to have the entire volume of water that it occupies sweep past the clay particle. $v_r \approx \Lambda_{Clay} G$

$$t_c = \frac{\Lambda_{Clay}^3}{\pi d_{Clay} L_{Diff_{NC}} \Lambda_{Clay} G} \tag{13.8}$$

Where $t_c = \frac{dN_c}{dt}$:

$$dN_c = \pi d_{Clay} L_{Diff_{NC}} \Lambda_{Clay}^{-2} G dt \tag{13.9}$$

13.2.1 Collision Rate and Particle Removal

A fraction of the remaining coagulant nanoparticles are removed during the time required for one sweep past the clay particle.

$$\frac{dn_{NC}}{-k\, n_{NC}} = dN_c \tag{13.10}$$

$$\frac{dn_{NC}}{-k\, n_{NC}} = \pi d_{Clay} L_{Diff_{NC}} \Lambda_{Clay}^{-2} G dt \tag{13.11}$$

13.2.2 Integrate the coagulant transport model

Integrate from the initial coagulant nanoparticle concentration to the concentration at time t.

$$\int_{n_{NC_0}}^{n_{NC}} n_{NC}^{-1}\, dn_{NC} = -\pi d_{Clay} L_{Diff_{NC}} \Lambda_{Clay}^{-2} G\, k \int_0^t dt \tag{13.12}$$

Use pC notation to be consistent with how we describe removal efficiency of other contaminants.

$$2.3 pC_{NC} = \pi d_{Clay}\, L_{Diff_{NC}}\, \Lambda_{Clay}^{-2}\, Gkt \tag{13.13}$$

Solve for the time required to reach a target efficiency of application of coagulant nanoparticles to clay.

$$t_{coagulant,\, application} = \frac{2.3 pC_{NC}\, \Lambda_{Clay}^2}{\pi G k\, d_{Clay}\, L_{Diff_{NC}}} \tag{13.14}$$

$G_{coagulant,\, application}$

$$\Delta h = \frac{G^2 \nu \theta}{g} \tag{13.15}$$

Replace θ with t.

$$\Delta h = \frac{G^2 \nu}{g} \frac{2.3 pC_{NC}\, \Lambda_{Clay}^2}{\pi G k\, d_{Clay}\, L_{Diff_{NC}}} \tag{13.16}$$

$$L_{Diff} \approx \left(\frac{2 k_B T d_{Clay}}{3\pi\, \mu\, d_{NC} G} \right)^{\frac{1}{3}} \tag{13.17}$$

$$\Delta h = \frac{G^2 \nu}{g} \frac{2.3 pC_{NC}\, \Lambda_{Clay}^2}{\pi G k\, d_{Clay}} \left(\frac{3\pi\, \mu\, d_{NC} G}{2 k_B T d_{Clay}} \right)^{\frac{1}{3}} \tag{13.18}$$

Solve for the velocity gradient.

$$\Delta h = \frac{G^{\frac{4}{3}} \nu}{g} \frac{2.3 pC_{NC}\, \Lambda_{Clay}^2}{\pi k\, d_{Clay}} \left(\frac{3\pi\, \mu\, d_{NC}}{2 k_B T d_{Clay}} \right)^{\frac{1}{3}} \tag{13.19}$$

$$G_{coagulant,\,application} = d_{Clay}\left(\frac{\pi k\, g\Delta h}{2.3pC_{NC}\,\Lambda_{Clay}^2\nu}\right)^{\frac{3}{4}}\left(\frac{2k_BT}{3\pi\,\mu\, d_{NC}}\right)^{\frac{1}{4}} \tag{13.20}$$

Table of G, ε, and h_L for Different Geometries Derivations

These are the derivations for the equations that *appear in the table containing equations for :math:'G*, ε, and h_L. ### Straight pipe (wall shear) The average energy dissipation rate, $\bar{\varepsilon}$, in a control volume with residence time θ is

$$\bar{\varepsilon} = \frac{gh_{\rm L}}{\theta} \tag{13.21}$$

The residence time can be expressed as a function of length and average velocity.

$$\theta = \frac{L}{\bar{v}} \tag{13.22}$$

For straight pipe flow the only head loss is due to wall shear and thus we have the Darcy Weisbach equation.

$$h_{\rm f} = {\rm f}\frac{L}{D}\frac{\bar{v}^2}{2g} \tag{13.23}$$

Combining the 3 previous equations we obtain the energy dissipation rate for pipe flow

$$\bar{\varepsilon} = \frac{\rm f}{2}\frac{\bar{v}^3}{D} \tag{13.24}$$

The average velocity gradient was defined by Camp and Stein as

$$G_{CS} = \sqrt{\frac{\bar{\varepsilon}}{\nu}} \tag{13.25}$$

where this approximation neglects the fact that square root of an average is not the same as the average of the square roots.

$$G_{CS} = \left(\frac{\rm f}{2\nu}\frac{\bar{v}^3}{D}\right)^{\frac{1}{2}} \tag{13.26}$$

or in terms of flow rate, we have:

$$G_{CS} = \left(\frac{32{\rm f}}{\pi^3\nu}\frac{Q^3}{D^7}\right)^{\frac{1}{2}} \tag{13.27}$$

13.2.3 Straight Pipe Laminar

Laboratory scale apparatus is often limited to laminar flow where viscosity effects dominate. The equations describing laminar flow conditions always include viscosity. For the case of laminar flow in a straight pipe, we have:

$${\rm f} = \frac{64}{Re} \tag{13.28}$$

Reynolds number is defined as

$$Re = \frac{\bar{v}D}{\nu} \tag{13.29}$$

The Darcy Weisbach head loss equation simplifies to the Hagen–Poiseuille equation for the case of laminar flow.

$$h_{\rm f} = \frac{32\nu L\bar{v}}{gD^2} \tag{13.30}$$

and thus the energy dissipation rate in a straight pipe under conditions of laminar flow is

$$\bar{\varepsilon} = 32\nu \left(\frac{\bar{v}}{D}\right)^2 \tag{13.31}$$

The Camp-Stein velocity gradient in a long straight laminar flow tube is thus

$$G_{CS}^2 = 32 \left(\frac{\bar{v}}{D}\right)^2 \tag{13.32}$$

$$G_{CS} = 4\sqrt{2}\frac{\bar{v}}{D} \tag{13.33}$$

Our estimate of G_{CS} based on $\bar{\varepsilon}$ is an overestimate because it assumes that the energy dissipation is completely uniform through the control volume. The true spatial average velocity gradient, $\bar{G}$, for laminar flow in a pipe is (Gregory, 1981),

$$\bar{G} = \frac{8}{3}\frac{\bar{v}}{D} \tag{13.34}$$

Our estimate of G_{CS} for the case of laminar flow in a pipe is too high by a factor of $\frac{3}{\sqrt{2}}$.

As a function of flow rate we have

$$\bar{v} = \frac{Q}{A} = \frac{4Q}{\pi D^2} \tag{13.35}$$

$$G_{CS} = \frac{16\sqrt{2}}{\pi}\frac{Q}{D^3} \tag{13.36}$$

13.2.4 Parallel Plates Laminar

Flow between parallel plates occurs in plate settlers in the sedimentation tank. We will derive the velocity gradient at the wall using the Navier Stokes equation.

We start with the Navier-Stokes equation written for flow in the x direction.

$$\frac{y^2}{2}\frac{dp}{dx} + Ay + B = \mu u \tag{13.37}$$

where u is the velocity in the x direction.

Apply the no slip condition at bottom plate.

$$u = 0 \quad at \quad y = 0 \tag{13.38}$$

Thus the constant $B = 0$.

Apply the no slip condition at top plate.

$$u = 0 \quad at \quad y = S \tag{13.39}$$

Thus the constant $A = \frac{-S}{2}\frac{dp}{dx}$

Substitute the values for constants A and B into the original equation.

$$\frac{y^2}{2}\frac{dp}{dx} - \frac{S}{2}\frac{dp}{dx}y = \mu u \tag{13.40}$$

Simply the equation to obtain

$$u = \frac{y\,(y - S)}{2\mu}\frac{dp}{dx} \tag{13.41}$$

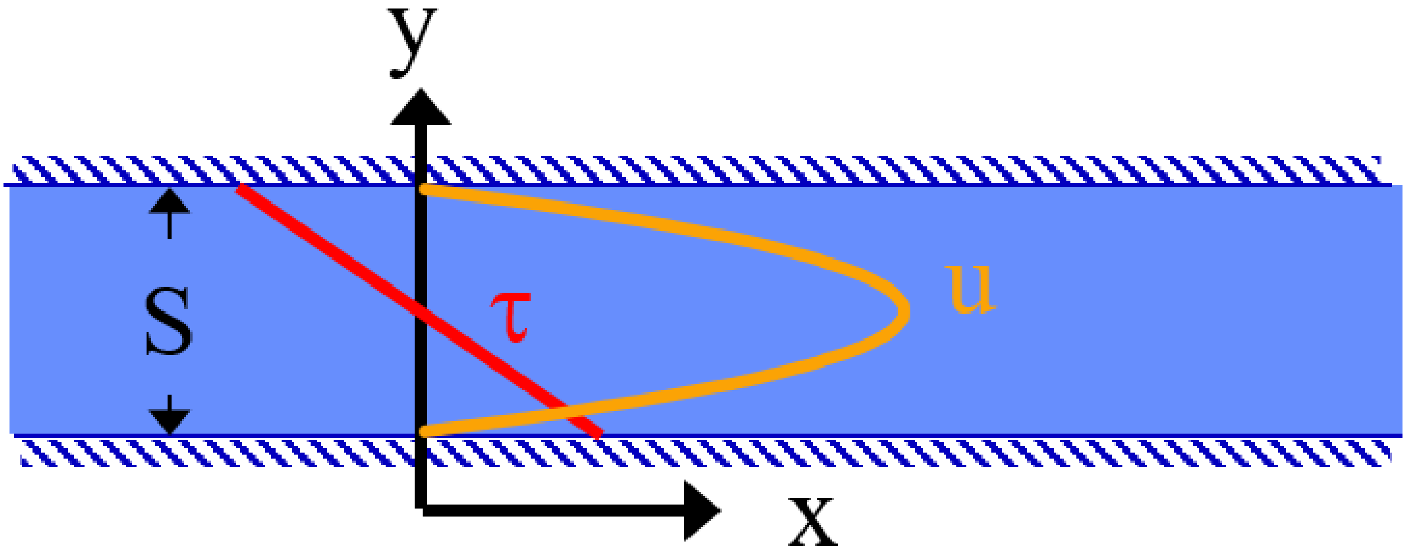


Fig. 13.1: A fluid flowing from left to right due to a pressure gradient results in wall shear on the parallel plates. This flow profile is for the case when $\frac{dp}{dx}$ is negative.

We need a relationship between average velocity and $\frac{dp}{dx}$. We can obtain this by integrating from 0 to S.

$$\bar{v} = \frac{q}{S} = \frac{1}{S}\int_0^S u dy = \frac{1}{S}\int_0^S \left(\frac{y^2 - Sy}{2\mu}\left(\frac{dp}{dx}\right)\right) dy \tag{13.42}$$

$$\bar{v} = -\frac{S^2}{12\mu}\frac{dp}{dx} \tag{13.43}$$

Solving for $\frac{dp}{dx}$

$$\frac{dp}{dx} = -\frac{12\mu\bar{v}}{S^2} \tag{13.44}$$

From the Navier Stokes equation after integrating once we get

$$\mu\left(\frac{du}{dy}\right) = y\frac{dp}{dx} + A \tag{13.45}$$

Substituting our boundary condition, $A = \frac{-S}{2}\frac{dp}{dx}$ we obtain

$$\frac{du}{dy}_{y=0} = -\frac{S}{2\mu}\frac{dp}{dx} \tag{13.46}$$

Substituting the result for $\frac{dp}{dx}$ we obtain

$$\frac{du}{dy}_{y=0} = \frac{6\bar{v}}{S} \tag{13.47}$$

Therefore in velocity gradient notation we have

$$G_{wall} = \frac{6\bar{v}}{S} \tag{13.48}$$

The energy dissipation rate at the wall

$$\varepsilon_{wall} = G_{wall}^2 \nu \tag{13.49}$$

$$\varepsilon_{wall} = \left(\frac{6\bar{v}}{S}\right)^2 \nu \tag{13.50}$$

Head loss due to shear on the plates is obtained from a force balance on a control volume between two parallel plates as shown in Fig. 13.1.

A force balance on a control volume gives

$$2\tau LW = -\Delta PWS \tag{13.51}$$

$$\Delta P = -\frac{2\tau L}{S} \tag{13.52}$$

The equation relating shear and velocity gradient is

$$\tau = \nu\rho\frac{du}{dy} = \nu\rho G \tag{13.53}$$

The velocity gradient at the wall is

$$G_{wall} = \frac{6\bar{v}}{S} \tag{13.54}$$

$$\tau = \nu\rho\frac{6\bar{v}}{S} \tag{13.55}$$

Substituting into the force balance equation

$$\Delta P = -\frac{2\nu\rho 6\bar{v}L}{S^2} \tag{13.56}$$

The head loss for horizontal flow at uniform velocity simplifies too

$$h_{\rm f} = \frac{-\Delta P}{\rho g} \tag{13.57}$$

$$h_{\rm f} = 12\frac{\nu\bar{v}L}{gS^2} \tag{13.58}$$

The average energy dissipation rate is

$$\bar{\varepsilon} = \frac{gh_{\rm L}}{\theta} \tag{13.59}$$

$$\bar{\varepsilon} = 12\nu\left(\frac{\bar{v}}{S}\right)^2 \tag{13.60}$$

The Camp-Stein velocity gradient for laminar flow between parallel plates is

$$G_{CS} = 2\sqrt{3}\frac{\bar{v}}{S} \tag{13.61}$$

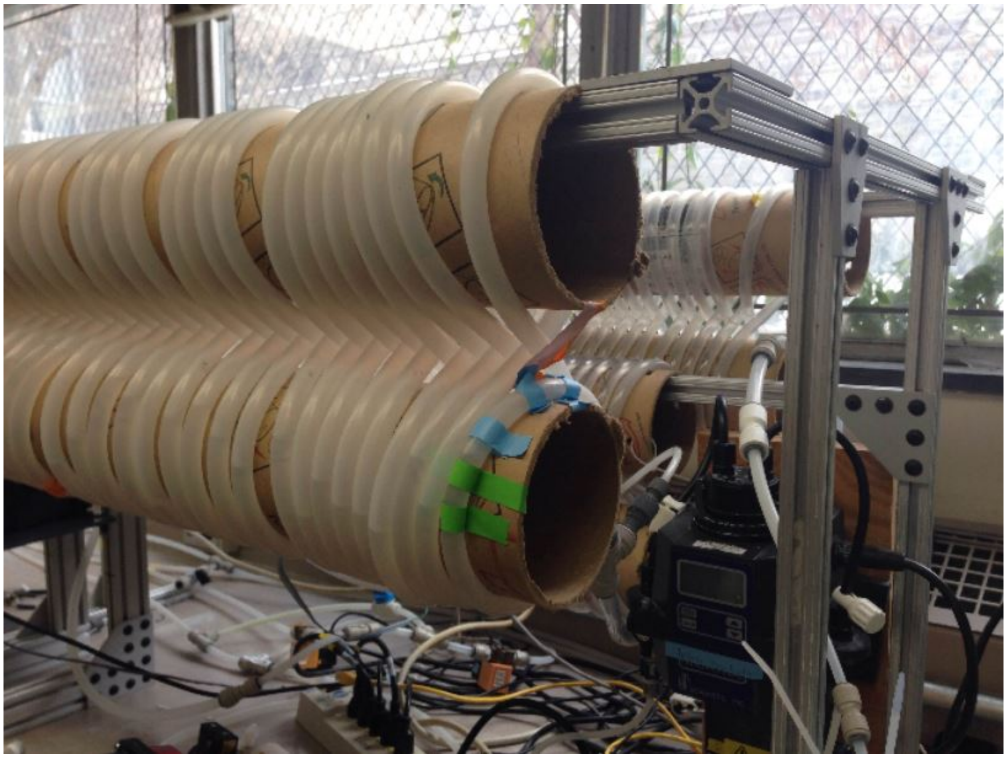


Fig. 13.2: The double coiled laminar flow flocculator creates secondary currents that oscillate in direction. This may be helpful in creating much more mixing than would occur in a straight laminar flow pipe.

13.2.5 Coiled tubes (laminar flow)

Coiled tubes are used as flocculators at laboratory scale. The one shown below is a doubled coil. A single coil would only go around one cylinder

‘ <https://confluence.cornell.edu/display/AGUACLARA/Laminar+Tube+Floc?preview=/10422268/258146480/ReportLaminarTubeFlocSpring2014.pdf>‘__

The ratio of the coiled to straight friction factors is given by Mishra and Gupta

The Dean number is defined as:

$$De = Re\left(\frac{D}{D_c}\right)^{\frac{1}{2}} \tag{13.62}$$

where D is the inner diameter of the tube and D_c is the diameter of the coil. Note that the tubing coils are actually helixes and that for the tubing diameters and coil diameters used for flocculators that the helix doesnt significantly change the radius of curvature.

$$\frac{\mathrm{f}_{coil}}{\mathrm{f}} = 1 + 0.033\left(log_{10}De\right)^4 \tag{13.63}$$

$$h_{L_{coil}} = h_{\mathrm{f}}\left[1 + 0.033\left(log_{10}De\right)^4\right] \tag{13.64}$$

where $h_{\mathrm{f}} = \frac{32\nu L\bar{v}}{gD^2}$. Note that we switch from major losses to total head loss here because the head loss from flowing around the coil is no longer simply due to shear on the wall.

$$h_{L_{coil}} = \frac{32\nu L\bar{v}}{gD^2}\left[1 + 0.033\left(log_{10}De\right)^4\right] \tag{13.65}$$

The average energy dissipation rate is

$$\bar{\varepsilon} = 32\nu\left(\frac{\bar{v}}{D}\right)^2\left[1 + 0.033\left(log_{10}De\right)^4\right] \tag{13.66}$$

The average velocity gradient is proportional to the square root of the head loss and thus we obtain

$$G_{CS_{coil}} = G_{CS}\left[1 + 0.033\left(log_{10}De\right)^4\right]^{\frac{1}{2}} \tag{13.67}$$

where $G_{CS} = 4\sqrt{2}\frac{\bar{v}}{D}$ for laminar flow in a straight pipe.

$$G_{CS_{coil}} = 4\sqrt{2}\frac{\bar{v}}{D}\left[1 + 0.033\left(log_{10}De\right)^4\right]^{\frac{1}{2}} \tag{13.68}$$

13.2.6 Expansions

The average energy dissipation rate for a flow expansion really only has meaning if there is a defined control volume where the mechanical energy is lost. Hydraulic flocculators provide such a case because the same flow expansion is repeated and thus the mechanical energy loss can be assumed to happen in the volume associated with one flow expansion. In this case we have

$$h_e = K\frac{\bar{v}_{out}^2}{2g} \tag{13.69}$$

In this equation K represents the fraction of the kinetic energy that is dissipated.

If we define the length of the control volume (in the direction of flow) as H then the residence time is

$$\theta = \frac{H}{\bar{v}} \tag{13.70}$$

$$\bar{\varepsilon} = \frac{gh_{\rm e}}{\theta} \tag{13.71}$$

Combining the previous equations we obtain

$$\bar{\varepsilon} = K\frac{\bar{v}_{out}^3}{2H} \tag{13.72}$$

$$G_{CS} = \sqrt{\frac{\bar{\varepsilon}}{\nu}} \tag{13.73}$$

$$G_{CS} = \bar{v}_{out}\sqrt{\frac{K\bar{v}_{out}}{2H\nu}} \tag{13.74}$$

13.3 Maximum velocity gradients

13.3.1 Straight pipe (major losses)

The maximum velocity gradient in pipe flow occurs at the wall. This is true for both laminar and turbulent flow. In either case a force balance on a control volume of pipe gives us the wall shear and the wall shear can then be used to estimate the velocity gradient at the wall.

A force balance for the case of steady flow in a round pipe requires that sum of the forces in the x direction must equal zero. Given a pipe with diameter, D, and length, L, we obtain

$$\left(P_{in} - P_{out}\right)\frac{\pi D^2}{4} = \tau_{wall}\pi DL \tag{13.75}$$

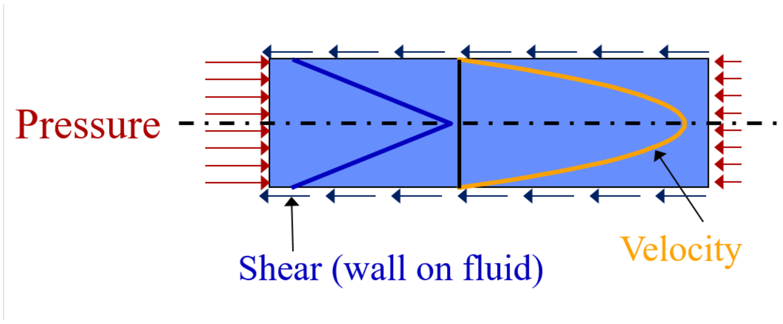


Fig. 13.3: A fluid flowing from left to right due to a pressure gradient results in wall shear.

$$-\Delta P\frac{D}{4} = \tau_{wall}L \tag{13.76}$$

For this control volume the energy equation simplifies to

$$-\Delta P = \rho g h_{\rm f} \tag{13.77}$$

The relationship between shear and velocity gradient is

$$\tau_{wall} = \mu\frac{du}{dy}_{wall} = \nu\rho G_{wall} \tag{13.78}$$

Combining the energy equation, the force balance, and the relationship between shear and velocity gradient we obtain

$$\rho g h_{\rm f}\frac{D}{4} = \nu\rho G_{wall}L \tag{13.79}$$

$$G_{wall} = \frac{g h_{\rm f} D}{4\nu L} \tag{13.80}$$

This equation is valid for both laminar flow. For turbulent flow it is necessary to make the approximation that wall shear perpendicular to the direction of flow is insignificant in increasing the magnitude of the wall shear. We can substitute the Darcy Weisbach equation for head loss to obtain

$$G_{wall} = {\rm f}\frac{\bar{v}^2}{8\nu} \tag{13.81}$$

The energy dissipation rate at the wall is

$$\varepsilon_{wall} = G_{wall}^2\nu \tag{13.82}$$

$$\varepsilon_{wall} = \frac{1}{\nu}\left({\rm f}\frac{\bar{v}^2}{8}\right)^2 \tag{13.83}$$

For laminar flow we can substitute ${\rm f} = \frac{64}{\rm Re}$ and the definition of the Reynolds number to obtain

$$G_{wall} = \frac{8\bar{v}}{D} \tag{13.84}$$

This equation is useful for finding the velocity gradient at the wall of a tube settler.

The energy dissipation rate at the wall is

$$\varepsilon_{wall} = G_{wall}^2\nu \tag{13.85}$$

$$\varepsilon_{wall} = \left(\frac{8\bar{v}}{D}\right)^2\nu \tag{13.86}$$

13.3.2 Coiled tubes (laminar flow)

The shear on the wall of a coiled tube is not uniform. The outside of the curve has a higher velocity gradient than the inside of the curve and there are secondary currents that results in wall shear that is not purely in the locally defined upstream direction. We do not have a precise equation for the wall shear. The best we can do currently is define an average wall shear in the locally defined direction of flow by combining $G_{CS_{wall_{coil}}} = \mathrm{f}_{\mathrm{coil}} \frac{\bar{\mathrm{v}}^2}{8\nu}$ and $\mathrm{f}_{coil} = \mathrm{f}\left[1 + 0.033\left(log_{10}De\right)^4\right]$ to obtain

$$G_{CS_{wall_{coil}}} = \mathrm{f}\left[1 + 0.033\left(log_{10}De\right)^4\right]\frac{\bar{v}^2}{8\nu} \tag{13.87}$$

13.3.3 Expansions

Flow expansions are used intentionally or unavoidable in multiple locations in hydraulically optimized water treatment plants. Rapid mix and hydraulic flocculation use flow expansions to generate fluid mixing and collisions between particles.

13.3.4 Round Jet

Baldyga, et al. 1995

$$\varepsilon_{Centerline} = \frac{50 D_{Jet}^3 \bar{v}_{Jet}^3}{\left(x - 2D_{Jet}\right)^4} \tag{13.88}$$

$$\varepsilon_{Max} = \frac{\left(\frac{50}{(5)^4}\right)\bar{v}_{Jet}^3}{D_{Jet}} \tag{13.89}$$

$$\varepsilon_{Max} = \Pi_{RoundJet}\frac{\bar{v}_{Jet}^3}{D_{Jet}} \tag{13.90}$$

$$\Pi_{RoundJet} = 0.08 \tag{13.91}$$

The maximum velocity gradient in a jet is thus

$$G_{Max} = \bar{v}_{Jet}\sqrt{\frac{\Pi_{RoundJet}\bar{v}_{Jet}}{\nu D_{Jet}}} \tag{13.92}$$

Below we plot the Baldyga et al.ăequation for the energy dissipation rate as a function of distance from the discharge location for the case of a round jet that is discharging into a large tank.

.. plot:: Rapid_Mix/plots/Jet_EDR.py *:include-source:*

13.3.5 Plane Jet

Plane jets occur in hydraulic flocculators and in the sedimentation tank inlet jet system. We havent been able to find a literature estimate of the maximum energy dissipation rate in a plane jet. Original measurements of a plane turbulent jet have been made by Heskestad in 1965 and it may be possible to use that data to get a better estimate of $:raw-latex:*Pi*_{JetPlane} $ from that source.

$$\Pi_{\bar{\epsilon}}^{\epsilon_{Max}} = \frac{\varepsilon_{Max}}{\bar{\varepsilon}} \tag{13.93}$$

$$\varepsilon_{Max} = \Pi_{JetPlane}\frac{\bar{v}_{Jet}^3}{S_{Jet}} \tag{13.94}$$

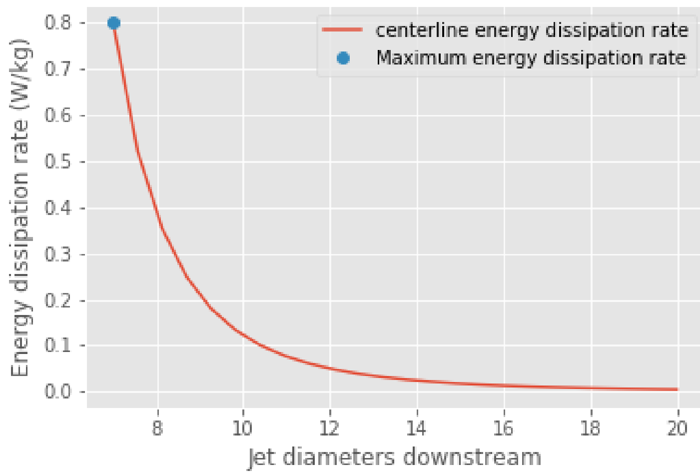


Fig. 13.4: The centerline energy dissipation rate downstream from a round jet. The distance downstream is measured in units of jet diameters. The energy dissipation rate between the jet and 7 jet diameters is developing as the shear between the stationary fluid and the jet propagates toward the center of the jet and turbulence is generated.

The maximum velocity gradient is thus

$$G_{Max} = \bar{v}_{Jet}\sqrt{\frac{\Pi_{JetPlane}\bar{v}_{Jet}}{\nu S_{Jet}}} \tag{13.95}$$

$$\bar{v} = \frac{Q}{SW} \tag{13.96}$$

$$\bar{v}_{Jet} = \frac{\bar{v}}{\Pi_{VCBaffle}} \tag{13.97}$$

$$S_{Jet} = S\Pi_{VCBaffle} \tag{13.98}$$

The average hydraulic residence time for the fluid between two baffles is

$$\theta_B = \frac{H}{\bar{v}} \tag{13.99}$$

where H is the depth of water. Substituting into the equation for ε_{Max} to get the equation in terms of the average velocity $\bar{v}$ and flow dimension S

$$\varepsilon_{Max} = \frac{\Pi_{JetPlane}}{S\Pi_{VCBaffle}}\left(\frac{\bar{v}}{\Pi_{VCBaffle}}\right)^3 \tag{13.100}$$

From the control volume analysis the average energy dissipation rate is

$$\bar{\varepsilon} = K\frac{\bar{v}^2}{2}\frac{1}{\theta_B} = \frac{K}{2}\frac{\bar{v}^3}{H_e} \tag{13.101}$$

where K is the minor loss coefficient for flow around the end of a baffle with a $180°$ turn.

Substitute the values for $\bar{\varepsilon}$ and ε_{Max} to obtain the ratio, $\Pi_{\bar{\epsilon}}^{\epsilon_{Max}}$

$$\Pi_{\bar{\epsilon}}^{\epsilon_{Max}} = \frac{\Pi_{JetPlane}}{\Pi_{VCBaffle}^4}\frac{2H_e}{KS} \tag{13.102}$$

$\Pi_{\bar{\varepsilon}}^{\varepsilon_{Max}}$ has a value of 2 for $H_e/S < 5$ (CFD analysis and Haarhoff, 2001) The transition value for H_e/S is at 5 (from CFD analysis, our weakest assumption).

We also have that $\Pi_{\bar{\varepsilon}}^{\varepsilon_{Max}}$ has a value of $\frac{\Pi_{JetPlane}}{\Pi_{VCBaffle}^4}\frac{2H_e}{KS}$ for $H_e/S > 5$. Thus we can solve for $\Pi_{JetPlane}$ at $H_e/S = 5$

$$\Pi_{JetPlane} = \left(\Pi_{\bar{\epsilon}}^{\epsilon_{Max}}\Pi_{VCBaffle}^4\frac{K}{2}\frac{S}{H_e}\right) \tag{13.103}$$

$$\Pi_{JetPlane} = 0.0124 \tag{13.104}$$

```
x=con.RATIO_VC_ORIFICE**2
Ratio_Jet_Plane = 2*con.RATIO_VC_ORIFICE**8 * con.K_MINOR_FLOC_BAFFLE/2/5
Ratio_Jet_Plane

con.RATIO_VC_ORIFICE**8*con.K_MINOR_FLOC_BAFFLE/Ratio_Jet_Plane
```

13.3.6 Behind a flat plate

A flat plate normal to the direction of flow could be used in a hydraulic flocculator. In vertical flow flocculators it would create a space where flocs can settle and thus it is not a recommended design.

The impellers used in mechanical flocculators could be modeled as a rotating flat plate. The energy dissipation rate in the wake behind the flat plate is often quite high in mechanical flocculators and this may be responsible for breaking previously formed flocs.

Ariane Walker-Horn modeled the flat plate using Fluent in 2015.

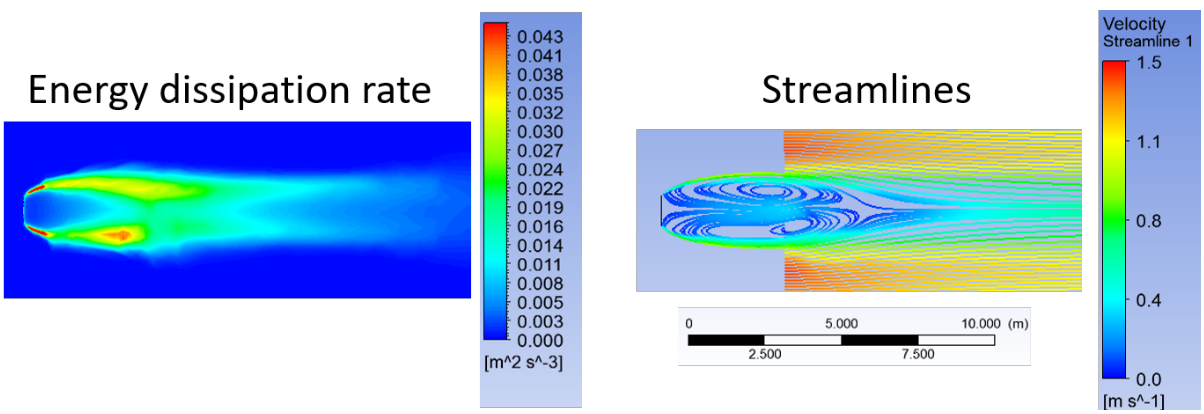


Fig. 13.5: The energy dissipation rate and streamlines for a 1 m wide plate in two dimensional flow with an approach velocity of $1m/s$. The maximum energy dissipation rate was approximately $0.04W/kg$.

$$\varepsilon_{Max} = \Pi_{Plate}\frac{\bar{v}^3}{W_{Plate}} \tag{13.105}$$

The maximum velocity gradient is thus

$$G_{Max} = \bar{v}\sqrt{\frac{\Pi_{Plate}\bar{v}}{\nu W_{Plate}}} \tag{13.106}$$

$$\Pi_{Plate} = \frac{(\varepsilon_{Max}W_{Plate})}{\bar{v}^3} \tag{13.107}$$

```
"""CFD analysis setup used by Ariane Walker-Horn in 2015"""
EDR_Max = 0.04*u.W/u.kg
v = 1*u.m/u.s
W = 1*u.m
Ratio_Jet_Plate = (EDR_Max * W/v**3).to_base_units()
print(Ratio_Jet_Plate)
```

CHAPTER

FOURTEEN

RAPID MIX APPENDIX C: EXAMPLES

14.1 Carbonate reactions, buffering, and pH

The coagulants used for drinking water treatment are acidic and thus result in a lowering of the pH of the treated water. The optimal pH for aluminum coagulant nanoparticle formation is between pH of 6.5 and 8.5. This is also the pH range set by the EPA secondary standards for drinking water. Although many water sources are within this pH range, there are some waters with more extreme values of pH. The aluminum and iron based coagulants are also acidic and in some waters the pH may drop below the ideal range when adding the coagulant. When the pH is outside the acceptable range it is necessary to adjust the pH by adding either a base or an acid.

When acid is added to a water containing bicarbonate, HCO_3^-, one of the potential reactions is for a proton to combine with HCO_3^- to form carbonic acid, H_2CO_3. If a base is added to water the reaction will proceed in the opposite direction. Carbonic acid, H_2CO_3, is chemical indistinguishable from dissolved carbon dioxide, $CO_{2_{aq}}$ and the total of carbonic acid and dissolved carbon dioxide is represented as $H_2CO_3^{\star}$. The reaction of bicarbonate to form carbonic acid removes one proton from solution and thus the concentration of protons doesnt increase as fast as we might have first expected as acid is added to the water.

The reactions of carbonate species with protons provides pH buffering capacity that must be considered when calculating the effect of acid or base addition. Since carbonates are the dominant buffering agents in natural waters it is essential to account for their influence on pH.

14.2 Carbonic Acid and Bicarbonate

$$H_2CO_3^{\star} \xleftrightarrow{K_1} H^+ + HCO_3^- \tag{14.1}$$

Where:

K_1 is the dissociation constant defined below.

$$K_1 = \frac{\left[H^+\right]\left[HCO_3^-\right]}{\left[H_2CO_3^{\star}\right]} \tag{14.2}$$

Where the [ ] indicates concentration in mole/L. We will use the p function, $p(x) = -log_{10}(x)$, to define the dissociation constant.

$$pK_1 = 6.3 \tag{14.3}$$

At the point of equal concentrations of bicarbonate and carbonic acid the dissociation constant, K_1, is equal to the hydrogen ion concentration, H^+. Thus we have equal concentrations at $pK_1 = pH$. This reaction is centered at pH = 6.3 and thus there is maximum buffering due to this reaction at pH = 6.3.

14.3 Bicarbonate and Carbonate

$$HCO_3^- \xleftrightarrow{K_2} H^+ + CO_3^{-2} \tag{14.4}$$

$$K_2 = \frac{[H^+]\left[CO_3^{-2}\right]}{\left[HCO_3^-\right]} \tag{14.5}$$

$$pK_2 = 10.3 \tag{14.6}$$

14.4 Total Concentration of Carbonates

The total concentration of carbonate species is given by

$$C_T = [H_2CO_3^\star] + \left[HCO_3^-\right] + \left[CO_3^{-2}\right] \\ Where :: math : `C_T`isthetotalconcentrationofcarbonates. \tag{14.7}$$

The total concentration of carbonates, C_T, is useful because it is conservative even though the individual species concentrations change as pH changes.

14.5 Alpha Notation

The alpha notation is used to show the concentration dependence on pH and to make the equations simpler.

$$[H_2CO_3^\star] = \alpha_0 C_T \tag{14.8}$$

$$\left[HCO_3^-\right] = \alpha_1 C_T \tag{14.9}$$

$$\left[CO_3^{-2}\right] = \alpha_2 C_T \tag{14.10}$$

14.6 Acid Neutralizing Capacity (ANC) or Alkalinity

Acid neutralizing capacity or alkalinity is the ability of a water sample to react with and neutralize an input of acid. The units of ANC are equivalents (or protons) per liter. Bicarbonate, HCO_3^-, can react with one proton, H^+, and thus each mole of HCO_3^- provides one equivalent per liter of ANC. The other terms in the equation have similar explanations.

$$\text{ANC} = [HCO_3^-] + 2[\text{CO}_3^{-2}] + [\text{OH}^-] - [\text{H}^+] \tag{14.11}$$

Note that carbonic acid and dissolved carbon dioxide are not in the ANC equation because they have no ability to neutralize protons.

We can write the ANC equation using alpha notation

$$ANC = C_T(\alpha_1 + 2\alpha_2) + \frac{K_w}{[H^+]} - \left[H^+\right] \tag{14.12}$$

The alphas are each a function of the proton concentration and the dissociation constants of the carbonate reactions.

$$\alpha_0 = \frac{1}{1 + \frac{K_1}{[H^+]} + \frac{K_1 K_2}{[H^+]^2}} \tag{14.13}$$

$$\alpha_0 = \frac{1}{1 + \frac{K_1}{[H^+]}\left(1 + \frac{K_2}{[H^+]}\right)} \tag{14.14}$$

$$\alpha_1 = \frac{1}{\frac{[\mathrm{H}^+]}{\mathrm{K}_1} + 1 + \frac{\mathrm{K}_2}{[\mathrm{H}^+]}} \tag{14.15}$$

$$\alpha_2 = \frac{1}{\frac{[\mathrm{H}^+]^2}{\mathrm{K}_1\mathrm{K}_2} + \frac{[\mathrm{H}^+]}{\mathrm{K}_2} + 1} \tag{14.16}$$

$$\alpha_2 = \frac{1}{1 + \frac{[\mathrm{H}^+]}{\mathrm{K}_2}\left(1 + \frac{[\mathrm{H}^+]}{\mathrm{K}_1}\right)} \tag{14.17}$$

For completeness we include acid neutralizing capacity for the case where the system is in equilibrium with atmospheric carbon dioxide, CO_2.

$$ANC_{atm\,equilibrium} = \frac{PCO_2K_H}{\alpha_0}(\alpha_1 + 2\alpha_2) + \frac{K_w}{[H^+]} - \left[H^+\right] \tag{14.18}$$

14.6.1 pH Adjustment

In drinking water treatment plant operation it is sometimes necessary to add a base (or acid) to increase (or decrease) the pH of the raw water. The carbonate system is most important in understanding how the base will adjust the pH because the reaction between carbonic acid and bicarbonate occurs around pH 6.3, the pK for that reaction. Carbon dioxide exchange with the atmosphere is insignificant in drinking water treatment unit processes unless there is a aeration stage. Thus we can use the ANC equation for the case with no CO_2 exchange with the atmosphere.

We will evaluate the case where we add a base that will increase the ANC of the raw water and it might also increase the total carbonate concentration. Our goal is to calculate how much of that base to add to reach a target pH. The final ANC after base addition is given by .. math:: ANC_1 = ANC_0 + Pi_{ANC}C_B

where:

ANC_1 is the final acid neutralizing capacity of the mixture after the base is added.

C_B is concentration of base in mole/liter Π_{ANC} is ANC per mole of base

The final carbonate concentration is given by

$$C_{T_1} = C_{T_0} + \Pi_{CO_3^{-2}}C_B \tag{14.19}$$

where:

C_{T_1} is the final total carbonate concentration of the mixture after the base is added.

$\Pi_{CO_3^{-2}}$ is mole of carbonate per mole of base (0 for $NaOH$ and 1 for Na_2CO_3)

Substituting these values into the ANC equation we obtain

$$ANC_0 + \Pi_{ANC}C_B = (C_{T_0} + \Pi_{CO_3^{-2}}C_B)(\alpha_1 + 2\alpha_2) + \frac{K_w}{[H^+]} - \left[H^+\right] \tag{14.20}$$

Now we solve for C_B, the concentration of base that must be added to reach a target pH.

$$(\Pi_{ANC} - \Pi_{CO_3^{-2}}(\alpha_1 + 2\alpha_2))C_B = C_{T_0}(\alpha_1 + 2\alpha_2) + \frac{K_w}{[H^+]} - \left[H^+\right] - ANC_0 \tag{14.21}$$

$$C_B = \frac{C_{T_0}(\alpha_1 + 2\alpha_2) + \frac{K_w}{[H^+]} - [H^+] - ANC_0}{\Pi_{ANC} - \Pi_{CO_3^{-2}}(\alpha_1 + 2\alpha_2)} \tag{14.22}$$

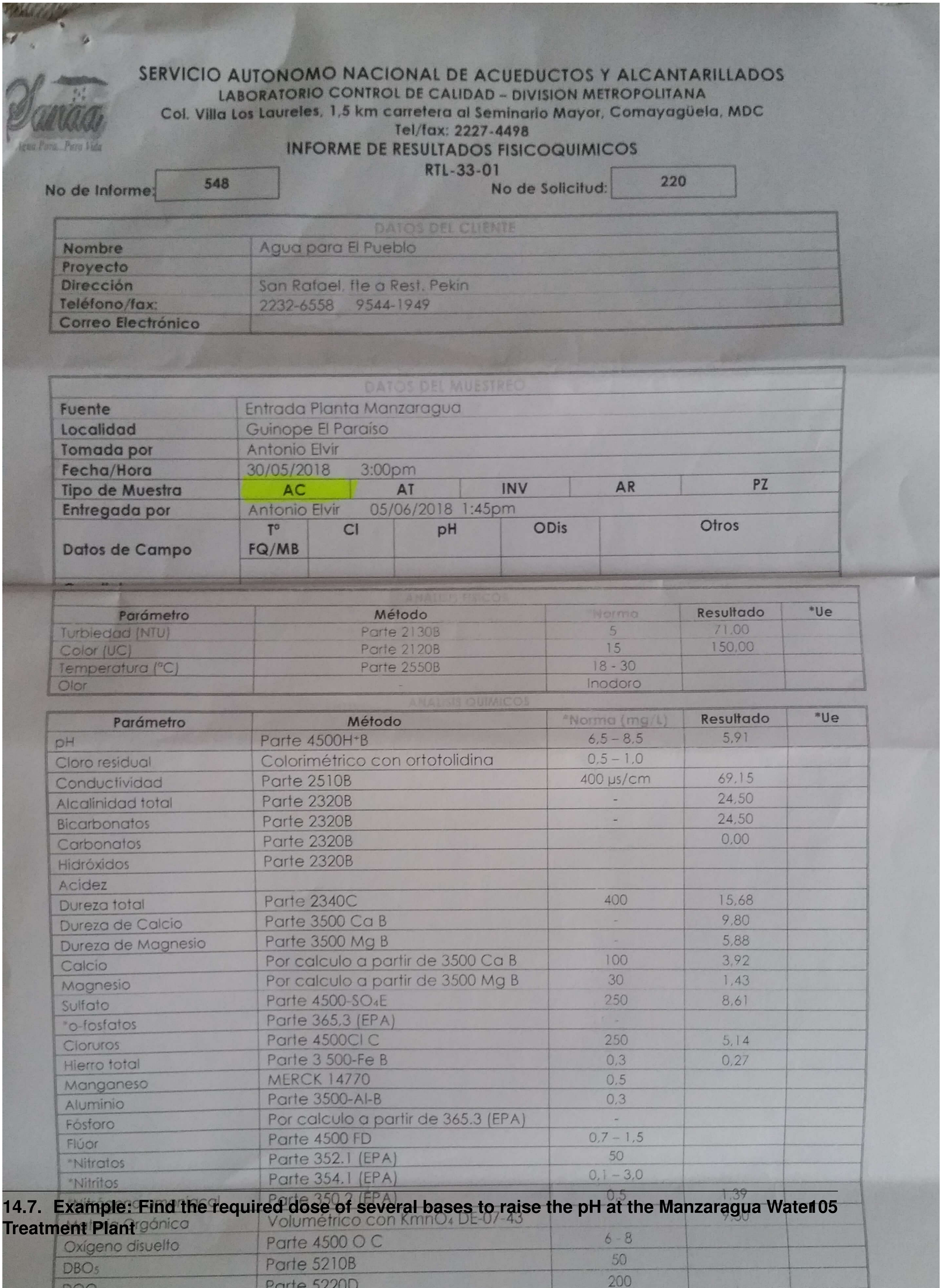
Note that the equations above can also be used for the case where acid is added to reduce the pH. In that case Π_{ANC} will have a negative value.

14.7 Example: Find the required dose of several bases to raise the pH at the Manzaragua Water Treatment Plant

The Mazaragua AguaClara plant consists of two 1 L/s plants operating in parallel. The plant is located in the municipality of Guinope, the department of El Paraiso, Honduras.

Fig. 14.1: Manzaragua water treatment plant using two of the AguaClara 1 L/s plants in parallel.

The plant performed very poorly from the first day of operation. The first attempted fix was to double the flocculator residence time by increasing the number of flocculator pipes (3 inch diameter by 1.5 m long) from 12 to 24. This improved performance, but the plant continued to perform poorly. A raw water sample was analyzed on May 30, 2018 and the following results were obtained.



SERVICIO AUTONOMO NACIONAL DE ACUEDUCTOS Y ALCANTARILLADOS
LABORATORIO CONTROL DE CALIDAD – DIVISION METROPOLITANA
Col. Villa Los Laureles, 1,5 km carretera al Seminario Mayor, Comayagüela, MDC
Tel/fax: 2227-4498
INFORME DE RESULTADOS FISICOQUIMICOS
RTL-33-01

No de Informe: 548   No de Solicitud: 220

DATOS DEL CLIENTE	
Nombre	Agua para El Pueblo
Proyecto	
Dirección	San Rafael, fte a Rest. Pekin
Teléfono/fax:	2232-6558 9544-1949
Correo Electrónico	

DATOS DEL MUESTREO					
Fuente	Entrada Planta Manzaragua				
Localidad	Guinope El Paraíso				
Tomada por	Antonio Elvir				
Fecha/Hora	30/05/2018 3:00pm				
Tipo de Muestra	AC	AT	INV	AR	PZ
Entregada por	Antonio Elvir 05/06/2018 1:45pm				
Datos de Campo	T° FQ/MB	Cl	pH	ODis	Otros

Parámetro	Método	*Norma	Resultado	*Ue
Turbiedad (NTU)	Parte 2130B	5	71,00	
Color (UC)	Parte 2120B	15	150,00	
Temperatura (°C)	Parte 2550B	18 - 30		
Olor	-	Inodoro		

ANALISIS QUIMICOS

Parámetro	Método	*Norma (mg/L)	Resultado	*Ue
pH	Parte 4500H⁺B	6,5 – 8,5	5,91	
Cloro residual	Colorimétrico con ortotolidina	0,5 – 1,0		
Conductividad	Parte 2510B	400 µs/cm	69,15	
Alcalinidad total	Parte 2320B	-	24,50	
Bicarbonatos	Parte 2320B	-	24,50	
Carbonatos	Parte 2320B		0,00	
Hidróxidos	Parte 2320B			
Acidez				
Dureza total	Parte 2340C	400	15,68	
Dureza de Calcio	Parte 3500 Ca B	-	9,80	
Dureza de Magnesio	Parte 3500 Mg B	-	5,88	
Calcio	Por calculo a partir de 3500 Ca B	100	3,92	
Magnesio	Por calculo a partir de 3500 Mg B	30	1,43	
Sulfato	Parte 4500-SO_4E	250	8,61	
*o-fosfatos	Parte 365.3 (EPA)	-		
Cloruros	Parte 4500Cl C	250	5,14	
Hierro total	Parte 3 500-Fe B	0,3	0,27	
Manganeso	MERCK 14770	0,5		
Aluminio	Parte 3500-Al-B	0,3		
Fósforo	Por calculo a partir de 365.3 (EPA)	-		
Flúor	Parte 4500 FD	0,7 – 1,5		
*Nitratos	Parte 352.1 (EPA)	50		
*Nitritos	Parte 354.1 (EPA)	0,1 – 3,0		
Nitrógeno amoniacal	Parte 350.2 (EPA)	0,5	1,39	
Materia Orgánica	Volumétrico con $KmnO_4$ DE-07-43		9,50	
Oxígeno disuelto	Parte 4500 O C	6 - 8		
DBO_5	Parte 5210B	50		
DQO	Parte 5220D	200		

Table 14.1: Manzaragua water quality analysis

Parameter	Units	Standard	Results
Turbidity	NTU	5	71
Color	color units	15	150
pH	pH	6.5 - 8.5	5.91
Conductivity	$\mu s/cm$	400	69.15
Alkalinity	mg/L as $CaCO_3$	•	24.5
Bicarbonates	mg/L as $CaCO_3$	•	24.5
Carbonates	mg/L as $CaCO_3$	•	0
Hardness	mg/L as $CaCO_3$	400	15.68

This water has high color which suggests a high concentration of dissolved organic matter. The pH is a clear problem because the pH is too low for the coagulant nanoparticles to precipitate. As the water sample pH of 5.91 a significant fraction of the coagulant will remain soluble.

Our goal is to determine how much base will need to be added to raise the pH. We do not have data on the *optimal* pH for treating high color water with PACl and so we will use pH 7 as the target. We will need a separate calculation to estimate how much additional Na_2CO_3 will need to be added to balance the PACl acidity.

At circumneutral pH (pH close to 7) the buffering capacity of the water is dominated by carbonate chemistry and specifically by the equilibrium between $H_2CO_3^*$ and $HCO_3^- $. We will use the acid neutralizing capacity (reported as calcium carbonate alkalinity) and the pH from the sample analysis to estimate the total concentration of carbonates. We will not use the sample analysis carbonate concentrations because they can not be precisely correct.

The solution steps are as follows: 1) Find total carbonate concentration, C_{T_0}, of the raw water sample using the ANC equation for the case where the system is not exchanging CO_2 with the atmosphere. 1) Solve for the required concentration of base, C_B.

For step 1 we need to solve the ANC equation for the carbonate concentration.

$$C_{T_0} = \frac{ANC_0 - \frac{K_w}{[H^+]} + [H^+]}{\alpha_1 + 2\alpha_2} \tag{14.23}$$

Note: We eventually should add the effect of the coagulant to this analysis so the required base concentration can be calculated given the raw water alkalinity, raw water pH, and coagulant dose.

Table 14.2: ANC and carbonate values for several bases and acids

Base/Acid	Pi_{ANC}	$Pi_{CO_3^{-2}}$
Na_2CO_3 or $CaCO_3$	2	1
$NaHCO_3$	1	1
$NaOH$	1	0
HCl or HNO_3	-1	0
H_2SO_4	-2	0

For Na_2CO_3 * Π_{ANC} = 2 because we are adding CO_3^{-2} which is multiplied by two in the ANC equation because CO_3^{-2} can react with two protons. * $\Pi_{CO_3^{-2}}$ = 1 because there is one mole of CO_3 per mole of Na_2CO_3

Below is the code used to calculate the required base addition.

```
from aide_design.play import*
from aguaclara_research.play import*
import aguaclara_research.Environmental_Processes_Analysis as epa

"""define molecular weights"""
m_Ca = 40.078*u.g/u.mol
m_C = 12.011*u.g/u.mol
m_O = 15.999*u.g/u.mol
m_Na = 22.99*u.g/u.mol
m_H = 1.008*u.g/u.mol
m_CaCO3 = m_Ca+m_C+3*m_O
m_Na2CO3 = 2*m_Na+m_C+3*m_O
m_NaHCO3 = m_Na+m_H+m_C+3*m_O
m_NaOH = m_Na+m_O+m_H

"""Raw water characteristics"""
pH_0 = 5.91
ANC_0 = (24.5 * u.mg/u.L/m_CaCO3).to(u.mmol/u.L)
ANC_0

def total_carbonates_closed(pH, ANC):
    """This function calculates total carbonates for a closed system given pH and ANC

    Parameters
    ----------
    pH : float
        pH of the sample
    ANC: float
        acid neutralizing capacity of the sample
    Returns
    -------
    The total carbonates of the sample
    Examples
    --------
    >>> total_carbonates_closed(1*u.mmol/u.L,8)
    1.017 mole/liter
    """
    return (ANC - epa.Kw/epa.invpH(pH) + epa.invpH(pH)) / (epa.alpha1_carbonate(pH) +
↪2 * epa.alpha2_carbonate(pH))


CT_0 = total_carbonates_closed(pH_0,ANC_0)


""" calculate the amount of base that must be added to reach a target pH"""
def pH_adjust(pH_0,ANC_0,Pi_ANC,Pi_CO3,pH_target):
  """This function calculates the required base (or acid) to adjust the pH to a
↪target value. The buffering capacity is assumed to be completely due to carbonate
↪species. The initial carbonate concentration is calculated based on the initial pH
↪and the initial ANC.

  Parameters
  ----------
  pH_0: float
      pH of the sample
  ANC_0: float
      acid neutralizing capacity (Alkalinity) of the sample in eq/L.
```

(continues on next page)

(continued from previous page)

```
  Pi_ANC: float
    equivalents of ANC per mole of base (or acid)
  Pi_CO3: float
    mole of carbonate per mole of base (or acid)
  pH_target: float
    pH goal
  Returns
  -------
  The required concentration of base (or acid) in millimoles/L
  Examples
  --------
  >>> pH_adjust(5.91,0.2*u.mmol/u.L,1,1,7)
  2.2892822041250924 millimole/liter
  """
  CT_0 = total_carbonates_closed(pH_0,ANC_0)
  B_num = CT_0 * (epa.alpha1_carbonate(pH_target) + 2 * epa.alpha2_carbonate(pH_
↪target)) + epa.Kw/epa.invpH(pH_target) - epa.invpH(pH_target) - ANC_0
  B_den = Pi_ANC - Pi_CO3*(epa.alpha1_carbonate(pH_target) + 2 * epa.alpha2_
↪carbonate(pH_target))
  return (B_num/B_den).to(u.mmol/u.L)

"""target pH"""
pH_target = 7

Pi_ANC_Na2CO3 = 2
Pi_CO3_Na2CO3 = 1

Pi_ANC_NaHCO3 = 1
Pi_CO3_NaHCO3 = 1

Pi_ANC_NaOH = 1
Pi_CO3_NaOH = 0

C_Na2CO3 = pH_adjust(pH_0,ANC_0,Pi_ANC_Na2CO3,Pi_CO3_Na2CO3,pH_target)

C_NaHCO3 = pH_adjust(pH_0,ANC_0,Pi_ANC_NaHCO3,Pi_CO3_NaHCO3,pH_target)
C_NaOH = pH_adjust(pH_0,ANC_0,Pi_ANC_NaOH,Pi_CO3_NaOH,pH_target)

"""Display results in a pandas table"""
base = ["NaOH","NaHCO3","Na2CO3"]
myindex = ["[mmoles/L]","[mg/L]"]
row1 = [C_Na2CO3.magnitude,C_NaHCO3.magnitude,C_NaOH.magnitude]
row2 = [(C_Na2CO3*m_Na2CO3).to(u.mg/u.L).magnitude,(C_NaHCO3*m_NaHCO3).to(u.mg/u.L).
↪magnitude,(C_NaOH*m_NaOH).to(u.mg/u.L).magnitude]
df = pd.DataFrame([row1,row2],index=myindex,columns=base)
print(df.round(2))

"""Graph the base concentration required as a function of the target pH"""
pH_graph = np.linspace(6,7,50)
C_Na2CO3 = pH_adjust(pH_0,ANC_0,Pi_ANC_Na2CO3,Pi_CO3_Na2CO3,pH_graph)
C_NaHCO3 = pH_adjust(pH_0,ANC_0,Pi_ANC_NaHCO3,Pi_CO3_NaHCO3,pH_graph)
C_NaOH = pH_adjust(pH_0,ANC_0,Pi_ANC_NaOH,Pi_CO3_NaOH,pH_graph)

fig, ax = plt.subplots()

ax.plot(pH_graph,C_NaHCO3)
ax.plot(pH_graph,C_Na2CO3)
```

(continues on next page)

(continued from previous page)

```
ax.plot(pH_graph,C_NaOH)
imagepath = 'Rapid_Mix/Images/'
ax.set(xlabel='pH target', ylabel='Base concentration (mmole/L)')
ax.legend(["sodium bicarbonate","sodium carbonate","sodium hydroxide"])
fig.savefig(imagepath+'mole_base_for_target_pH')
plt.show()

fig, ax = plt.subplots()
ax.plot(pH_graph,(C_Na2CO3*m_Na2CO3).to(u.mg/u.L))
ax.plot(pH_graph,(C_NaOH*m_NaOH).to(u.mg/u.L))
ax.set(xlabel='pH target', ylabel='Base concentration (mg/L)')
ax.legend(["sodium carbonate","sodium hydroxide"])
fig.savefig(imagepath+'mg_base_for_target_pH')
plt.show()
```

The analysis reveals that the choice of base matters. The most efficient (on a mass or mole basis) base is $NaOH$ because it doesnt add any carbonates that dont fully react with the hydrogen ions. The decision about which base to use will be influenced by economics, operator safety, and by whether additional carbonate buffering simplifies plant operation with changing raw water quality.

Table 14.3: Calcium base.

Chemical name	common name	Chemcal formula
calcium carbonate	limestone or chalk	$CaCO_3$
calcium hydroxide	slaked lime or hydrated lime	$Ca(OH)_2$
calcium oxide	quicklime	CaO

The calcium bases are relatively inexpensive and have the disadvantage of lower solubility than sodium bases. Calcium carbonate has a low solubility, carbon dioxide is present in the atmosphere, and thus calcium carbonate precipitation limits the concentration that can be used for chemical feeds.

The required dose for each of the bases is summarized below.

Table 14.4: Dose of each base required to change the pH of the Manzaragua water to 7.

units	$NaOH$	$NaHCO_3$	Na_2CO_3
[mmoles/L]	0.45	2.8	0.53
[mg/L]	47.21	235.0	21.19

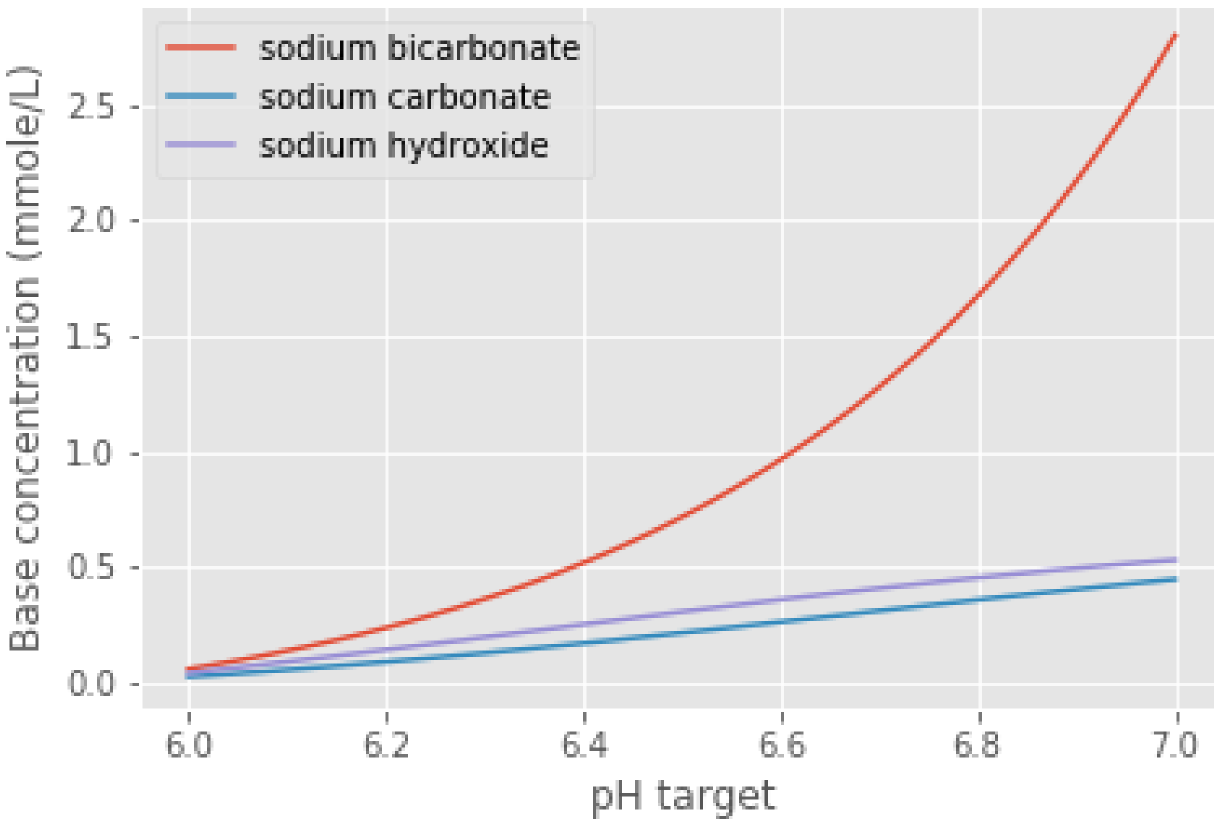


Fig. 14.3: Dose of three bases (in mole/L) required to achieve a target pH for the Manzaragua water. Carbonates provide more buffering and less change in the pH compared with $NaOH$.

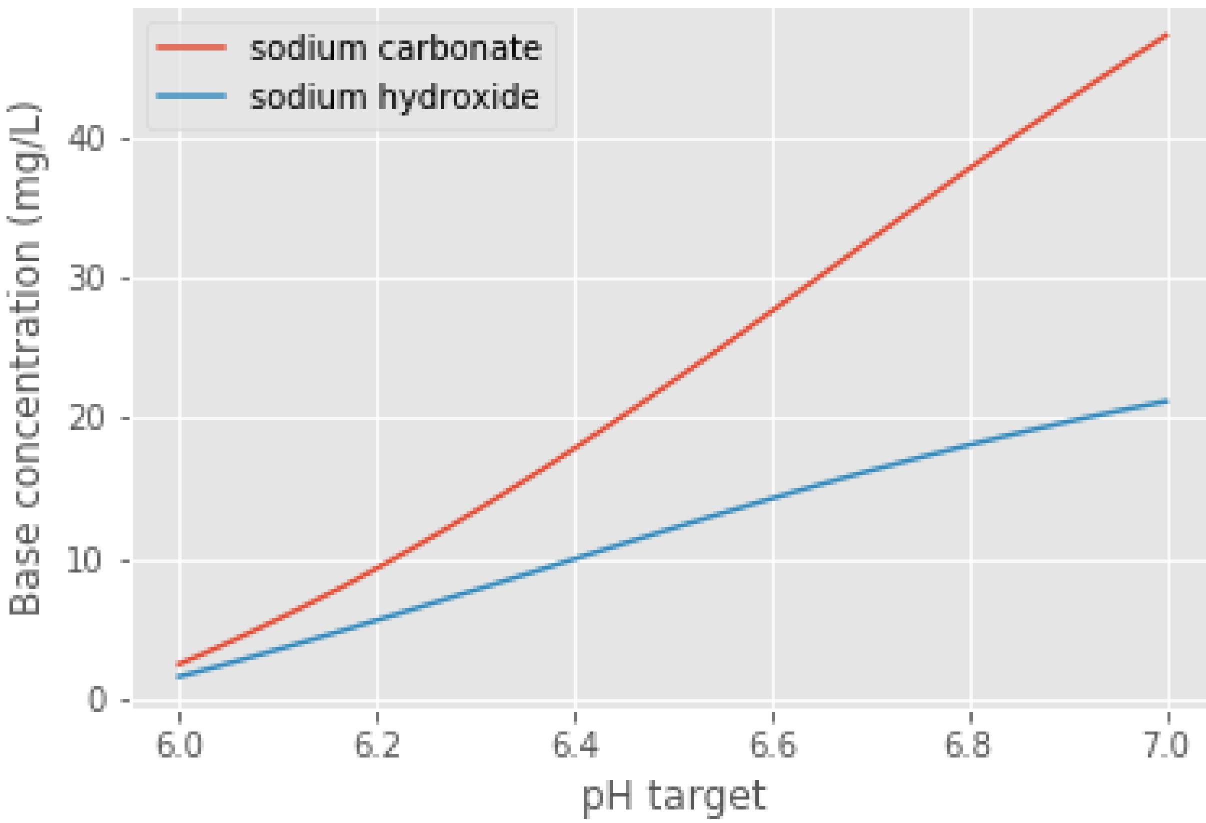


Fig. 14.4: Dose of two bases (in mg/L) required to achieve a target pH for the Manzaragua water. Carbonates provide more buffering and less change in the pH compared with $NaOH$.

CHAPTER

FIFTEEN

RAPID MIX THEORY AND FUTURE WORK

15.1 Equations for ε and G in Varying Flow Geometries

Estimation of velocity gradients for various flow geometries is the basis for the design of rapid mix, flocculators, and plate settlers. Thus, our goal is to define the velocity gradients consistently across a range of possible flow regimes. There are three approaches to calculating the average velocity gradient within a control volume. 1) Use the Navier Stokes equations and solve for the spatially averaged velocity gradient. 1) Use Computational Fluid Dynamics (CFD) to solve for the spatially averaged velocity gradient. 1) Use the total mechanical energy loss in the control volume to calculate the energy dissipation rate. Estimate the velocity gradient directly from the energy dissipation rate, $G_{CS} = \sqrt{\frac{\bar{\varepsilon}}{\nu}}$, as defined by Camp and Stein in 1943 (Camp, T. R., and Stein, P. C. (1943) Velocity Gradients and Hydraulic Work in Fluid Motion, J. Boston Soc. Civil Eng., 30, 203–221.).

The first approach would be ideal but is difficult in practice because Navier Stokes solutions are only available for limited geometries and laminar flow. CFD could be used but is difficult to use as a general engineering design approach given the large number of geometries that are used in drinking water treatment plants. For these reasons we will use the control volume approach to estimate the average velocity gradient. This method incorrectly assumes that the energy dissipation rate is completely uniform in the control volume and hence the velocity gradient is also uniform. This method results in an over estimation of the velocity gradient. The Camp-Stein estimate of G_{CS} is based on a control volume where the velocity gradient is uniform. Consider a layer of fluid of depth H and apply a velocity, v at the top of the fluid. The velocity gradient, G, is thus $\frac{v}{H}$ everywhere in the fluid. The force required to move the top of the fluid at velocity v can be obtained from the required shear, τ. From Newtons Law of Friction we have

$$\tau = \mu\frac{v}{H} = \mu G = \nu\rho G \tag{15.1}$$

Where τ is the force required per unit plan view area. The power per unit area required to move the fluid at velocity v is τv. The mass per unit area is ρH. Thus the energy dissipation rate or the power per mass is

$$\varepsilon = \frac{P}{m} = \frac{\tau v}{\rho H} = \frac{\nu\rho G v}{\rho H} = \nu G^2 \tag{15.2}$$

This equation has no approximations, but has one very important assumption. We derived this equation for a control volume where the velocity gradient was **uniform**. The reactors and control volumes that we will be using as we design water treatment plants will **not** have uniform velocity gradients. Indeed, several of the water treatment processes will be turbulent and thus the velocity gradients in the fluid will vary in both space and time. Even in laminar flow in a pipe the velocity gradient is far from uniform with high velocity gradients at the wall and zero velocity gradient at the center of the pipe.

Wed like to know if we can apply the previous equation

$$\varepsilon = \nu G^2 \tag{15.3}$$

to the case where the energy dissipation rate and velocity gradients are nonuniform by simply introducing average values of both quantities.

$$\bar{\varepsilon} \stackrel{?}{=} \nu\bar{G}^2 \tag{15.4}$$

We will test this option with a simple case. Consider a hypothetical reactor (case 2) that is 4 times as large in plan view area as the uniform velocity gradient case explored above (case 1). In addition, assume that 3/4 of the reactor has a velocity gradient of zero. The average energy dissipation rate for case 1 is

$$\bar{\varepsilon}_1 = \frac{P_1}{m_1} = \nu \bar{G}_1^2 \tag{15.5}$$

The average energy dissipation rate for case 2 is

$$\bar{\varepsilon}_2 = \frac{P_1}{4m_1} = \frac{\bar{\varepsilon}_1}{4} \tag{15.6}$$

This makes sense because we are putting in the same amount of energy into a control volume that is 4 times bigger.

Now we calculate the velocity gradients. As previously determined,

$$\bar{G}_1 = \sqrt{\frac{\bar{\varepsilon}_1}{\nu}} \tag{15.7}$$

The average velocity gradient in the second control volume is simply the volume weighted average

$$\bar{G}_2 = \bar{G}_1 \frac{1}{4} + 0 \frac{3}{4} \tag{15.8}$$

where 1/4 of the case 2 control volume has the same velocity gradient as the case 1 control volume and 3/4 of the control volume has a velocity gradient of 0. The Camp Stein method would suggest that $\bar{G}_2$ is equal to

$$\bar{G}_2 \stackrel{?}{=} \sqrt{\frac{\bar{\varepsilon}_2}{\nu}} = \sqrt{\frac{\bar{\varepsilon}_1}{4\nu}} \tag{15.9}$$

Now we check to see if the Camp Stein method of estimating the average velocity gradient, $\bar{G}$, is correct.

$$\bar{G}_2 = \frac{\bar{G}_1}{4} \neq \sqrt{\frac{\bar{\varepsilon}_1}{4\nu}} = \frac{\bar{G}_1}{2} \tag{15.10}$$

Given that the energy dissipation rate is proportional to the square of the velocity gradient the mean of the energy dissipation rate is **not** proportional to the mean of the velocity gradient. Thus the Camp Stein method of calculating the average velocity gradient is not correct except in the case of uniform velocity gradient. The Camp Stein equation is dimensionally correct and could be corrected by adding a dimensionless constant Π_{CS} that is a function of the energy dissipation rate distribution within the control volume.

$$\bar{G} = \Pi_{CS} \sqrt{\frac{\bar{\varepsilon}}{\nu}} \tag{15.11}$$

where Π_{CS} is 1 for a uniform velocity gradient and is less than one for non uniform velocity gradients. We can think Π_{CS} as a measure of the efficiency of using energy to deform the fluid. We can calculate Π_{CS} for cases where we have either a Navier Stokes or a computation fluid dynamics estimate of $\bar{G}$.

The conventional approach to design of flocculators uses the Camp Stein definition of

$$G_{CS} = \sqrt{\frac{\bar{\varepsilon}}{\nu}} \tag{15.12}$$

where G_{CS} is **not** the average velocity gradient, but is larger than the average velocity gradient by a factor of Π_{CS}. Thus we have

$$G_{CS} = \Pi_{CS} \bar{G} \tag{15.13}$$

Use of the Camp Stein velocity gradient in design of mixing units and flocculators results in an error when applying results from one reactor to another. If the energy dissipation rate distribution within the reactors is different, then Π_{CS} will be different for the two reactors and the actual average velocity gradient, $\bar{G}$ will be different for the two reactors.

Given that energy is used more efficiently to produce velocity gradients if the velocity gradients are uniform, our goal is to design mixing and flocculation units that have relatively uniform velocity gradients. If all of our reactors at both research scale and municipal scale have similar values of Π_{CS}, then we can use the Camp Stein definition of G_{CS} and not introduce any significant errors. It will not be reasonable, however, to expect similar performance based on similar values of G_{CS} if one reactor has relatively uniform energy dissipation rates and the other reactor has zones with very high energy dissipation rates and zones with very low energy dissipation rates.

We will demonstrate later that mechanically mixed reactors typically have a much wider range of energy dissipation rates than do well designed hydraulically mixed reactors. Thus comparisons between mechanically mixed and hydraulically mixed reactors must account for differences in Π_{CS}.

We will use the Camp Stein definition $G_{CS} = \sqrt{\frac{\bar{\varepsilon}}{\nu}}$ as the design parameter of convenience in this textbook.

15.1.1 Previously in Rapid Mix Intro Section (delete this header once you are reacquainted with the document)

Our understanding of rapid mix is currently quite speculative. This is an area that requires substantial research. We have anecdotal evidence that the process of transporting coagulant nanoparticles to suspended particle surfaces may be a slow, rate-limiting process. Dissolved organic matter may influence the rate of coagulant nanoparticle transport by effectively increasing the size of the coagulant nanoparticles and thus reducing the diffusion rate.

Developing a fundamental understanding of the mixing and transport processes that occur between coagulant addition and flocculation is a very high priority for the AguaClara program.

15.2 Estimates of time required for mixing processes

15.2.1 Turbulent Large Scale Eddies

The first step in mixing is at the scale of the largest eddies. The largest eddies are limited in size by the smallest dimension normal to the direction of flow. Thus in a pipe the dimension of the largest eddies is set by the pipe diameter. In a open channel the dimension of the largest eddies is usually the water depth although it could be the width of the channel for the case of a narrow tall channel.

We can use the eddy velocity to estimate how long it will take for an eddy to cross the smallest dimension of flow. Eddy velocity is $v_{eddy} \approx (\bar{\varepsilon} \, L_{eddy})^{\frac{1}{3}}$. The $\approx$ indicates that this relationship is the same order of magnitude. In a pipe we have

$$v_{eddy} \approx (\bar{\varepsilon} \, D)^{\frac{1}{3}} \tag{15.14}$$

For a long straight pipe $\bar{\varepsilon} = \frac{\text{f}}{2}\frac{\bar{v}^3}{D}$ and thus we can obtain the ratio between mean velocity and the velocity of the large scale eddies.

$$v_{eddy} \approx \left(\frac{\text{f}}{2}\frac{\bar{v}^3}{D} D\right)^{\frac{1}{3}} \tag{15.15}$$

$$\frac{v_{eddy}}{\bar{v}} \approx \left(\frac{\text{f}}{2}\right)^{\frac{1}{3}} \tag{15.16}$$

Given a friction factor of 0.02, the eddy velocity is approximately 20% of the mean velocity. We can use this ratio to estimate how many pipe diameters downstream from an injection point will the coagulant be mixed across the diameter of the pipe.

$$N_{D_{pipe}} \approx \frac{\bar{v}}{v_{eddy}} \approx \left(\frac{2}{\text{f}}\right)^{\frac{1}{3}} \tag{15.17}$$

Where $N_{D_{pipe}}$ is the distance in number of pipe diameters downstream of the injection point where complete mixing will have occurred. This estimate is a minimum distance and a factor of safety of 2 or more would reasonably be applied. In addition it is best practice to inject the coagulant in the center of the pipe. Injecting the coagulant at the side of the pipe will require considerably greater distance downstream for mixing across the pipe.

```
print((0.02/2)**(1/3))
```

15.2.2 Inner Viscous Length Scale

The smallest scale at which inertia containing eddies causes mixing is set by the final damping of inertia by viscosity. Turbulence occurs when fluid inertia is too large to be damped by viscosity. The ratio of inertia to viscosity is given by the Reynolds number, Re:

$$\mathrm{Re} = \frac{\bar{v}D}{\nu} \tag{15.18}$$

Flows with high Reynolds numbers are turbulent (inertia dominated) and with low Reynolds are laminar (viscosity dominated). The transition Reynolds number is a function of the flow geometry and the velocity and length scale that are used to characterize the flow. In all turbulent flows there is a length scale at which inertia finally loses to viscosity. The scale where viscosity wins is some multiple of the Kolmogorov length scale, which is defined as:

$$\eta_K = \left(\frac{\nu^3}{\varepsilon}\right)^{\frac{1}{4}} \tag{15.19}$$

where η_K is the Kolmogorov length scale. At the Kolmogorov length scale viscosity completely dampens the inertia of the eddies and effectively kills the turbulence.

The length scale at which most of the kinetic energy contained in the small eddies is dissipated by viscosity is the inner viscous length scale, λ_v, which is about 50 times larger than the Kolmogorov length scale. Thus we have

$$\lambda_\nu = \Pi_{K\nu}\left(\frac{\nu^3}{\varepsilon}\right)^{\frac{1}{4}} \tag{15.20}$$

{#eq:inner_viscous_length}

where $\Pi_{K\nu} = 50$

At length scales larger than the inner viscous length scale, λ_v, the dominant transport mechanism is by turbulent eddies. At length scales smaller than λ_v the dominant transport mechanism is fluid deformation due to shear. If the flow regime is completely laminar such as in a small diameter tube flocculator, then the dominant transport mechanism is fluid deformation due to shear at length scales all the way up to the diameter of the tubing.

The dividing line between eddy transport and fluid deformation controlled by viscosity can be calculated as a function of the energy dissipation rate using [@eq:inner_viscous_length].

```
""" importing """
from aide_design.play import*
from aguaclara_research.play import*
import aguaclara_research.floc_model as fm
import matplotlib.pyplot as plt
from matplotlib.ticker import FormatStrFormatter
imagepath = 'AguaClara Water Treatment Plant Design/Rapid Mix/Images/'
EDR_array = np.logspace(0,4,num=50)*u.mW/u.kg
Temperature = 20*u.degC
def Inner_viscous(EDR, Temperature):
    return fm.RATIO_KOLMOGOROV * fm.eta_kolmogorov(EDR, Temperature)
```

(continues on next page)

(continued from previous page)

```
fig, ax = plt.subplots()
ax.semilogx(EDR_array.to(u.mW/u.kg),Inner_viscous(EDR_array, Temperature).to(u.mm))
ax.yaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.xaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.set(xlabel='Energy dissipation rate (W/kg)', ylabel='Inner viscous length scale␣
↪(mm)')
ax.text(30, 6, 'Eddies cause mixing', fontsize=12,rotation=-30)
ax.text(1, 5, 'Shear and diffusion cause mixing', fontsize=12,rotation=-30)
fig.savefig(imagepath+'Inner_viscous_vs_EDR')
plt.show()
```

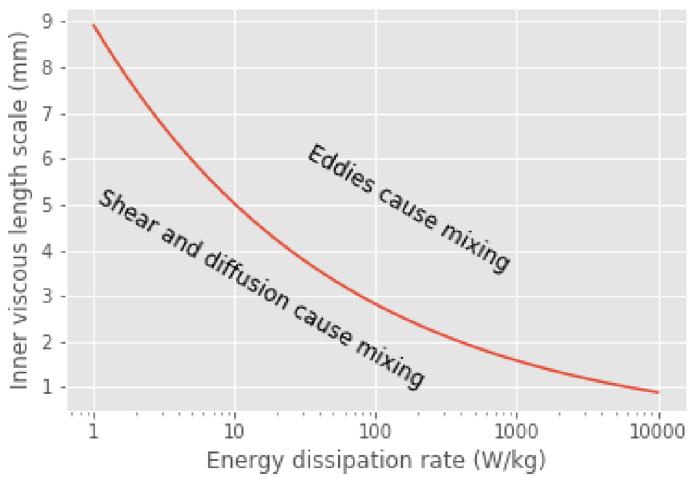


Fig. 15.1: Eddies can cause fluid mixing down to the scale of a few millimeters for energy dissipation rates used in rapid mix units and flocculators.

15.2.3 Turbulent Mixing Time as a Function of Scale

We are searching for the rate limiting step in the mixing process as we transition from the scale of the flow down to the scale of the coagulant nanoparticles. We can estimate the time required for eddies to mix at their length scales by assuming that the eddies pass all of their energy to smaller scales in the time it takes for an eddy to travel the distance equal to the length scale of the eddy. This time is known as the **'eddy turnover time <http://ceeserver.cee.cornell.edu/eac20/cee637/handouts/TURBFLOW_1.pdf>'__**, t_{eddy}. *The derivation for the equation below is found here*.

$$t_{eddy} \approx \left(\frac{L_{eddy}^2}{\bar{\varepsilon}}\right)^{\frac{1}{3}} \tag{15.21}$$

We can plot the eddy turnover time as a function of scale from the inner viscous length scale up to the scale of the flow. We will discover whether large scale mixing by eddies is faster or slower than small scale mixing by eddies.

```
EDR_graph = np.array([0.01,0.1,1,10 ])*u.W/u.kg
Temperature
"""Use the highest EDR to estimate the smallest length scale"""
Inner_viscous_graph = Inner_viscous(EDR_graph[2], Temperature)
Inner_viscous_graph
```

(continues on next page)

(continued from previous page)

```
L_flow = 0.5*u.m
L_scale = np.logspace(np.log10(Inner_viscous_graph.magnitude),np.log10(L_flow.
↪magnitude),50)
L_scale
fig, ax = plt.subplots()
for i in range(len(EDR_graph)):
  ax.semilogx(L_scale,((L_scale**2/EDR_graph[i])**(1/3)).to_base_units())

ax.legend(EDR_graph)

#ax.yaxis.set_major_formatter(FormatStrFormatter('%.f'))
#ax.xaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.set(xlabel='Length (m)', ylabel='Eddy turnover time (s)')
fig.savefig(imagepath+'Eddy_turnover_time')
plt.show()
```

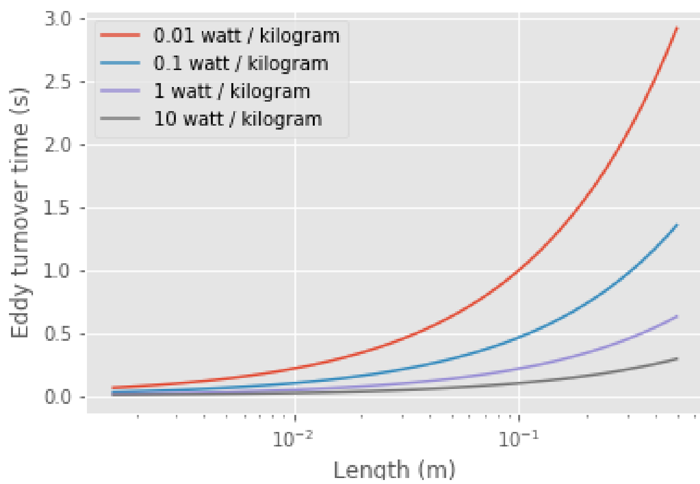


Fig. 15.2: Eddy turnover times as a function of length scale for a range of energy dissipation rates.

The eddy turnover times are longest for the largest eddies and this analysis suggests that it only takes a few seconds for turbulent eddies to mix from the scale of the flow down to the inner viscous length scale.

The large scale mixing time is critical for the design of water treatment plants for the case where the flow is split into multiple treatment trains after coagulant addition. In this case it is critical that the coagulant be mixed equally between all of the treatment trains and thus the mixing times shown in the previous graph represent a minimum time between where the coagulant is added and where the flow is divided into the parallel treatment trains.

It is likely this process of mixing from the scale of the flow down to the inner viscous length scale is commonly referred to as rapid mix. Here we showed that this mixing is indeed rapid and is really only a concern in the case where the coagulant injection point is very close to the location where the flow is split into multiple treatment trains.

15.2.4 Shear-Diffusion Transport

After the first few seconds in which mixing occurs from the length scale of the flow down to the inner viscous length scale the next step in the transport process is blending of the coagulant uniformly with the raw water. At the end of the turbulent transport the coagulant stock has been stretched out into thin bands throughout the raw water, but the two fluids are not actually blended together by turbulence. The blending is accomplished by fluid deformation and then by molecular diffusion.

15.2.5 Fluid Deformation by Shear

The time scale for fluid deformation is $1/G$ where G is the velocity gradient. This simple relationship is because the velocity of fluid deformation is proportional to the length scale and thus the time to travel any given distance in a linear velocity gradient is always the same. Velocity gradients in conventional mechanized rapid mix units are order 1000 Hz and thus the time for fluid deformation to blur concentration gradients is approximately 1 ms. This confirms the idea that blending the coagulant with the raw water is actually a very fast process with the slowest phase being the transport by turbulent eddies at the scale of reactor.

15.2.6 Einsteins Diffusion Equation

We can estimate the length scale at which fluid shear and diffusion provide transport at the same rate. Einsteins diffusion equation is

$$D_{Diffusion} = \frac{k_B T}{3\pi\mu d_P} \tag{15.22}$$

where k_B is the Boltzmann constant and d_P is the diameter of the particle that is diffusion in a fluid with viscosity ν and density ρ. The diffusion coefficient $D_{Diffusion}$ has dimensions of $\frac{[L^2]}{[T]}$ and can be understood as the velocity of the particle multiplied by the length of the mean free path.

From dimensional analysis the time for diffusion to blur a concentration gradient over a length scale, $L_{Diffusion}$ is

$$t_{Diffusion} \approx \frac{L^2_{Diffusion}}{D_{Diffusion}} \tag{15.23}$$

The shear time scale is $1/G$ and thus we can solve for the length scale at which diffusion and shear have equivalent transport rates.

$$1/G \approx t_{Diffusion} \approx \frac{L^2_{Diffusion}}{D_{Diffusion}} \tag{15.24}$$

Substitute Einsteins diffusion equation and solve for the length scale that transitions between shear and diffusion transport.

$$L^{Shear}_{Diffusion} \approx \sqrt{\frac{k_B T}{3G\pi\mu d_P}} \tag{15.25}$$

```
def L_Shear_Diffusion(G,Temperature,d_particle):
  return np.sqrt((u.boltzmann_constant*Temperature/
  (3 * G *  np.pi *pc.viscosity_dynamic(Temperature)* d_particle)).to_base_units())

G = 100*u.Hz
d_particle = fm.PACl.Diameter*u.m
x = (L_Shear_Diffusion(G,Temperature,d_particle)).to(u.nm)
print(x)
```

Molecular diffusion finishes the blending process by transporting the coagulant nanoparticles the last few hundred nanometers. The entire mixing process from the coagulant injection point to uniform blending with the raw water takes only a few seconds.

We have demonstrated that all of the steps for mixing of the coagulant nanoparticles with the raw water are very fast. Compare

1. Molecular diffusion causes some dissolved species and Al nanoparticles to aggregate. 1. Fluid shear and molecular diffusion cause Al nanoparticles with attached formerly dissolved species to collide with inorganic particles (such as clay) and organic particles (such as viruses, bacteria, and protozoans).

15.3 Length Scales of Coagulant Nanoparticles and Clay

The coagulant nanoparticles eventually will attach to clay particles. The clay particles have a diameter of approximately $5\,\mu\mathrm{m}$ and thus it is clear from the length scale in the figure above that turbulent eddies arent able to transport all the way to attachment to clay.

15.4 Diffusion and Shear Transport Coagulant Nanoparticles to Clay

The time required for shear and diffusion to transport coagulant nanoparticles to clay has previously been assumed to be a rapid process.

- Diffusion blends the coagulant with the raw water sufficiently so that the coagulant precipitates and forms nanoparticles.
- Dissolved organic molecules diffuse to the coagulant nanoparticles and adhere to the nanoparticle surface.
- The coagulant nanoparticles are transported to suspended particle surfaces by a combination of diffusion and fluid shear.

The following is a very preliminary estimate of the time required for attachment of the nanoparticles to the clay particles. This analysis includes multiple simplifying assumptions and there is a reasonable possibility that some of those assumptions are wrong. However, the core assumptions that coagulant nanoparticles are transported to clay particles by a combination of fluid deformation (shear) and molecular diffusion is reasonable.

The volume of the suspension that is cleared of nanoparticles is proportional to a collision area defined by a ring around the clay particle with width of the diameter of the nanoparticle diffusion band. This diffusion band is the length scale over which diffusion is able to transport coagulant particles to the clay surface during the time that the nanoparticles are sliding past the clay particle.

$$\propto \pi\, d_{Clay}\, L_{Diff_{NC}} \tag{15.26}$$

The volume cleared is proportional to time

$$\propto t \tag{15.27}$$

The volume cleared is proportional to the relative velocity between clay and nanoparticles. This scaling

$$\propto v_r \tag{15.28}$$

$$\bar{v}_{\text{Cleared}} = \pi d_{Clay}\, L_{Diff_{NC}} v_r t \tag{15.29}$$

Use dimensional analysis to get a relative velocity for the long range transport controlled by shear.

$$v_r = f\left(\varepsilon, \nu, \Lambda_{Clay}\right) \tag{15.30}$$

$$v_r = \Lambda_{Clay} f\left(\varepsilon, \nu\right) \tag{15.31}$$

$$v_r \approx \Lambda_{Clay} G \tag{15.32}$$

$$\Lambda_{Clay} = [L] \quad \varepsilon = \frac{[L]^2}{[T]^3} \quad \nu = \frac{[L]^2}{[T]} \tag{15.33}$$

15.4.1 Diffusion band thickness

The time required for shear to transport all of the fluid past the clay so that diffusion can transport the coagulant nanoparticles to the clay surface is significant.

$$D_{Diffusion} = \frac{k_B T}{3\pi \, \mu \, d_P} \tag{15.34}$$

$$L_{Diff} \approx \sqrt{D_{Diffusion} t_{Diffusion}} \tag{15.35}$$

The time for nanoparticles to diffuse through the boundary layer around the clay particle is equal to the distance they travel around the clay particle divided by their velocity. The distance they travel scales with d_{Clay} and their average velocity scales with the thickness of the diffusion layer/2 * the velocity gradient.

$$t_{Diffusion} = \frac{2 d_{Clay}}{L_{Diff} G} \tag{15.36}$$

$$L_{Diff} \approx \left(\frac{2 k_B T d_{Clay}}{3\pi \, \mu \, d_{NC} G} \right)^{\frac{1}{3}} \tag{15.37}$$

Lets estimate the thickness of the diffusion band

```
T_graph = np.linspace(0,30,4)*u.degC
G = np.arange(50,5000,50)*u.Hz

def L_Diff(Temperature,G):
  return (((2*u.boltzmann_constant*Temperature * fm.Clay.Diameter*u.m)/(3 * np.pi *pc.
↪viscosity_dynamic(Temperature)*  (fm.PACl.Diameter*u.m)*G))**(1/3)).to_base_units()

fig, ax = plt.subplots()
for i in range(len(T_graph)):
  ax.semilogx(G,L_Diff(T_graph[i],G).to(u.nm))

ax.legend(T_graph)
ax.yaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.xaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.set(xlabel='Velocity gradient (Hz)', ylabel='Diffusion band thickness ($nm$)')
fig.savefig(imagepath+'Diffusion_band_thickness')
plt.show()
```

Using the equation for L_{Diff} above, we can solve for the time required to reach a target efficiency of application of coagulant nanoparticles to clay:

$$t_{coagulant,\, application} = \frac{2.3 pC_{NC} \, \Lambda^2_{Clay}}{\pi G k \, d_{Clay} \, L_{Diff_{NC}}} \tag{15.38}$$

The time required for the coagulant to be transported to clay surfaces is strongly dependent on the turbidity as indicated by the average spacing of clay particles, Λ_{Clay}. As turbidity increases the spacing between clay particles decreases and the time required for shear to transport coagulant nanoparticles to the clay decreases. Increasing the shear also results in faster transport of the coagulant nanoparticles to clay surfaces. The times required are strongly influenced by the size of the coagulant nanoparticles because larger nanoparticles diffuse more slowly.

Below we estimate the time required to achieve 80% attachment of nanoparticles in a 10 NTU clay suspension.

```
"""I needed to attach units to material properties due to a bug in floc_model. This␣
↪will need to be fixed when floc_model is updated."""
def Nano_coag_attach_time(pC_NC,C_clay,G,Temperature):
  """We assume that 70% of nanoparticles attach in the average time for one collision.
↪"""
```

(continues on next page)

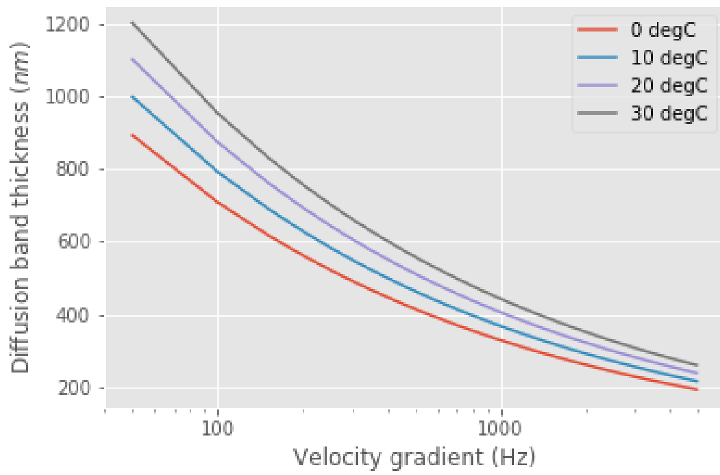


Fig. 15.3: Molecular diffusion band thickness as a function of velocity gradient. This length scale marks the transition between transport by fluid deformation and by diffusion.

(continued from previous page)

```
  k_nano = 1-np.exp(-1)
  num=2.3*pC_NC*(fm.sep_dist_clay(C_clay,fm.Clay))**2
  den = np.pi * G* k_nano * fm.Clay.Diameter*u.m * L_Diff(Temperature,G)
  return (num/den).to_base_units()

C_Al = 2 * u.mg/u.L
C_clay = 10 * u.NTU
pC_NC = -np.log10(1-0.8)
"""apply 80% of the coagulant nanoparticles to the clay"""

G = np.arange(50,5000,10)*u.Hz

fig, ax = plt.subplots()

for i in range(len(T_graph)):
  ax.semilogx(G,Nano_coag_attach_time(pC_NC,C_clay,G,T_graph[i]))

ax.semilogx(Mix_G.to(1/u.s),Mix_HRT.to(u.s),'o')
ax.legend([*T_graph, "Conventional rapid mix"])
"""* is used to unpack T_graph so that units are preserved when adding another legend␣
↪item."""
ax.yaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.xaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.set(xlabel='Velocity gradient (Hz)', ylabel='Nanoparticle attachment time (s)')
fig.savefig(imagepath+'Coag_attach_time')
plt.show()
```

15.4.2 Energy Tradeoff for Coagulant Transport

$$\Delta h = \frac{G^2 \nu \theta}{g} \tag{15.39}$$

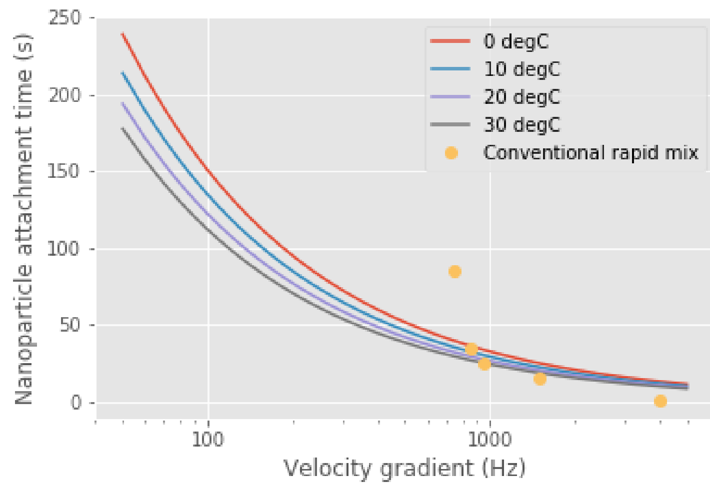


Fig. 15.4: An estimate of the time required for 80% of the coagulant nanoparticles to attach to clay particles given a raw water turbidity of 10 NTU.

```
Nano_attach_time = Nano_coag_attach_time(pC_NC,C_clay,G,Temperature)

def HL_coag_attach(pC_NC,C_clay,G,Temperature):
  return (G**2*pc.viscosity_kinematic(Temperature)*Nano_attach_time/u.gravity).to(u.
↪cm)

fig, ax = plt.subplots()

for i in range(len(T_graph)):
  ax.loglog(G,HL_coag_attach(pC_NC,C_clay,G,T_graph[i]))

ax.legend(T_graph)
ax.yaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.xaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.set(xlabel='Velocity gradient (Hz)', ylabel='Head loss (cm)')
fig.savefig(imagepath+'Coag_attach_head_loss')
plt.show()
```

There is an economic tradeoff between reactor volume and energy input. The reactor volume results in a higher capital cost and the energy input requires both higher operating costs and higher capital costs. This provides an opportunity to optimize rapid mix design once we have a confirmed model characterizing the process.

The total potential energy used to operate an AguaClara plant is approximately 2 m. This represents the difference in elevation between where the raw water enters the plant and where the filtered water exits the plant. If we assume that the rapid mix energy budget is a fraction of that total and thus for subsequent analysis we will assume somewhat arbitrarily that the energy available to attach the coagulant nanoparticles to the raw water particles is 50 cm.

We solve the coagulant transport model, $t_{coagulant,\, application} = \frac{2.3pC_{NC}\,\Lambda^2_{Clay}}{\pi Gk\, d_{Clay}\, L_{Diff_{NC}}}$, for G given a head loss.

$$G_{coagulant,\, application} = d_{Clay}\left(\frac{\pi k\, g\Delta h}{2.3pC_{NC}\,\Lambda^2_{Clay}\nu}\right)^{\frac{3}{4}}\left(\frac{2k_BT}{3\pi\,\mu\, d_{NC}}\right)^{\frac{1}{4}} \qquad (15.40)$$

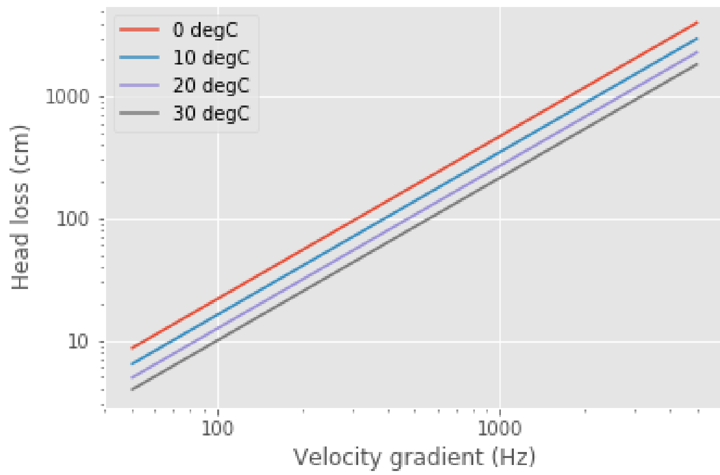


Fig. 15.5: The total energy required to attach coagulant nanoparticles to raw water inorganic particles increases rapidly with the velocity gradient used in the rapid mix process.

```
"""find G for target head loss"""
HL_nano_transport = np.linspace(10,100,10)*u.cm
def G_max_head_loss(pC_NC,C_clay,HL_nano_transport,Temperature):
  k_nano = 1-np.exp(-1)
  num = u.gravity * HL_nano_transport * np.pi * k_nano
  den= 2.3 * pC_NC * (fm.sep_dist_clay(C_clay,fm.Clay))**2 * pc.viscosity_
↪kinematic(Temperature)
  num2 = 2 * u.boltzmann_constant * Temperature
  den2 = 3 * np.pi * pc.viscosity_dynamic(Temperature) * (fm.PACl.Diameter*u.m)
  return fm.Clay.Diameter*u.m*((((num/den)**(3) * (num2/den2)).to_base_units())**(1/
↪4))
"""Note the use of to_base_units BEFORE raising to the fractional power.
This prevents a rounding error in the unit exponent."""

G_max = G_max_head_loss(pC_NC,C_clay,20*u.cm,Temperature)
print(G_max)

"""The time required?"""
Nano_attach_time = Nano_coag_attach_time(pC_NC,C_clay,G_max,Temperature)
print(Nano_attach_time)
print(G_max*Nano_attach_time)
```

According to the analysis above, the maximum velocity gradient that can be used to achieve 80% coagulant nanoparticle attachment using only 20 cm of head loss is 142 Hz. This requires a residence time of 100 seconds. These model results must be experimentally verified and it is very likely that the model will need to be modified.

The analysis of the time required for shear and diffusion to transport the coagulant nanoparticles the last few millimeters suggests that it is this last step that requires the most time. Indeed, the time required for coagulant nanoparticle attachment to raw water particles is comparable to the time that will be required for the next step in the processs, flocculation.

15.5 Coagulant Attachment Mechanism

We do not yet understand the origin of the bonds that form between coagulant nanoparticles, between a coagulant nanoparticle and suspended particles, and between coagulant nanoparticles and dissolved organic molecules. Historically the role of the coagulant was assumed to be to reduce the repulsive force between particles so that the particles could get close enough for Van der Waals forces to hold the particles together.

- Surface charge neutralization hypothesis
  - coagulant nanoparticles attach to each other
  -
- Polar bonds
  - Electronegativity reveals that the aluminum - oxygen bond is more polar than the hydrogen - oxygen bond
  - The bond between a coagulant nanoparticle and a clay surface can potentially be stronger than the bond between a water molecule and the clay surface.

15.6 Conventional Mechanical Rapid Mix

15.6.1 Maximum Velocity Gradients

```
Mix_HRT = np.array([0.5,15,25,35,85])*u.s
Mix_G = np.array([4000,1500,950,850,750])/u.s
Mix_CP = np.multiply(Mix_HRT, np.sqrt(Mix_G))
Mix_Gt = np.multiply(Mix_HRT, Mix_G)
Mix_EDR = (Mix_G**2*pc.viscosity_kinematic(Temperature))

fig, ax = plt.subplots()
ax.plot(Mix_G.to(1/u.s),Mix_HRT.to(u.s),'o')
ax.yaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.xaxis.set_major_formatter(FormatStrFormatter('%.f'))
ax.set(xlabel='Velocity gradient (Hz)', ylabel='Residence time (s)')
fig.savefig(imagepath+'Mechanical_RM_Gt')
plt.show()
```

Conventional rapid mix units use mechanical or potential energy to generate intense turbulence to begin the mixing process. Conventional design is based on the use of $\bar{G}$ (an average velocity gradient) as a design parameter. We dont yet know what the design objective is for rapid mix and thus it isnt clear which parameters matter. We hypothesize that both velocity gradients that cause deformation of the fluid and time for molecular diffusion are required to ultimately transport coagulant nanoparticles to the surfaces of clay particles.

The velocity gradient can be obtained from the rate at which mechanical energy is being dissipated and converted to heat by viscosity.

$$\varepsilon = G^2\nu \tag{15.41}$$

where ε is the energy dissipation rate, G is the velocity gradient, and ν is the kinematic viscosity of water. We can estimate the power input required to create a target energy dissipation rate for a conventional design by noting that power is simple the energy dissipation rate times the mass of water in the rapid mix unit.

$$P = \bar{\varepsilon}\forall\rho \tag{15.42}$$

$$P = \bar{G}^2\nu\forall\rho \tag{15.43}$$

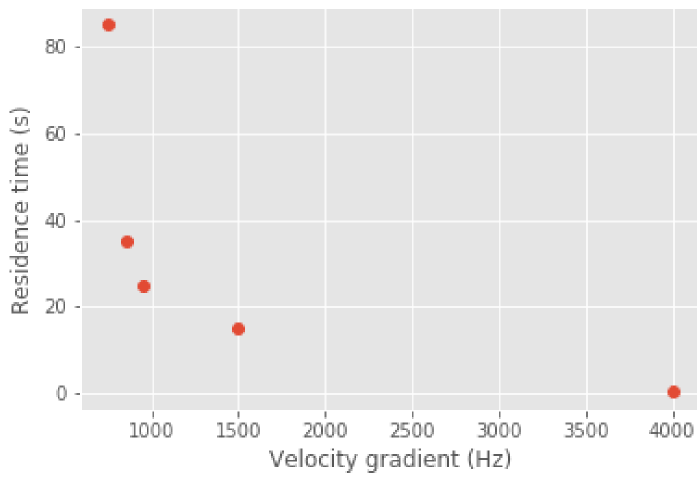


Fig. 15.6: Mechanical rapid mix units use a wide range of velocity gradients and residence times.

We can relate reactor volume to a hydraulic residence time, θ, and volumetric flow rate, Q.

$$P = \rho \bar{G}^2 \nu Q \theta \tag{15.44}$$

This equation is perfectly useful for estimating electrical motor sizing requirements for mechanical rapid mix units. For gravity powered hydraulic rapid mix units it would be more intuitive to use the change in water surface elevation, Δh instead of power input.

$$P = \rho g Q \Delta h \tag{15.45}$$

Combining the two equations we obtain.

$$\Delta h = \frac{G^2 \nu \theta}{g} \tag{15.46}$$

Table 15.1: Typical values for conventional rapid mix residence time and average velocity gradients

Residence Time (s)	Velocity gradient G (1/s)	Energy dissipation rate (W/kg)	Equivalent height (m)
0.5	4000	16	0.8
10 - 20	1500	2.25	2.3 - 4.6
20 - 30	950	0.9	1.8 - 2.8
30 - 40	850	0.72	2.2 - 2.9
40 - 130	750	0.56	2.3 - 7.5

From Environmental Engineering: A Design Approach by Sincero and Sincero. 1996. page 267.

Rotating propellers can either be installed in open tanks or enclosed in pipes. From a mixing and fluids perspective it doesnt make any difference whether the tank is open to the atmosphere or not. The parameters of interest are the rate of fluid deformation and the residence time in the mixing zone.

CHAPTER

SIXTEEN

FLOCCULATION INTRODUCTION

```
# %%
from aide_design.play import*
from aguaclara_research.play import*
from pytexit import py2tex
from sympy import*
from scipy.optimize import root
from scipy.optimize import brentq
import pandas as pd
```

16.1 Flocculation

Flocculation transform inorganic (clays such as kaolinite, smectite, etc. and metallic oxy-hydroxides such as goethite and gibbsite) and organic (viruses, bacteria and protozoa) primary particles into flocs (particle aggregates). Flocculation doesnt remove any particles from suspension. Instead it causes particle aggregation and then floc blankets, lamellar sedimentation, and sand filtration will be used to separate those flocs from the water. Sedimentation can remove flocs more easily than it can remove primary particles because flocs have a higher terminal sedimentation velocity. Floc blankets and sand filtration rely primarily on capture based on interception and interception is much more efficient when the particles are larger. Thus the purpose of flocculation is to join **all** of the primary particles together into flocs.

It is also possible that a difference in a physical property between primary particles and flocs plays a role in enhanced removal of flocs in floc blankets and filters. For example, the many relatively weak connection points between the primary particles in the flocs enables the flocs to deform. It is possible that deformation plays an important role right at the moment of collision. Presumably the bond strength required to lock the colliding particles together is less if the particles can deform as they are colliding.

16.1.1 Primary particles cant attach to large flocs

One of the mysteries of flocculation has been why it is such a slow process and yet it appears to be a very rapid process. Plant operators observe that with high raw water turbidities that they can see flocculation progressing after about 0.5 minutes of flocculation. We can estimate the collision potential, $G\theta$ that corresponds to making visible flocs.

$$\bar{G} = \sqrt{\frac{gh_e}{\theta\nu}} \tag{16.1}$$

```
HL_floc = 43*u.cm
HRT = 8 * u.min
Temperature =20 * u.degC
```

(continues on next page)

(continued from previous page)

```
G_floc = ((pc.gravity*HL_floc/(HRT*pc.viscosity_kinematic(Temperature)))**0.5).to_
↪base_units()
print(G_floc)
Gt_floc = G_floc*HRT
HRT_floc_visible = 0.5*u.min
Gt_floc_visible = (G_floc*HRT_floc_visible).to_base_units()
print(Gt_floc_visible)
```

Here initial flocculation is visible at a $Gl/theta$ of less than 3000. Given that flocculation is visible at this low collision potential, it is unclear why recommended $Gl/theta$ are as high as 100,000. This is one of the great mysteries that motivated the search for a flocculation model that was based on hypotheses that were consistent with laboratory and field observations. ## History The mechanism of particle-particle aggregation was thought to be controlled by an average surface charge. Apparently no one was able to develop a model of how that mechanism would influence particle attachment efficiency and the result was that no predictive models for flocculation were developed. There were several observations that were at odds with conventional explanations of flocculation. 1. Efficient flocculation at coagulant dosages that led to positive surface charge. This led to a second flocculation mechanism that was called sweep floc and that was used to describe any observations that didnt fit the charge neutralization flocculation hypotheses 1. Flocculation time for highly turbid suspensions was expected to proceed very rapidly and produce very low turbidity settled water. This expectation was not observed and led to the hypothesis that flocs were continually breaking up and producing primary particles or at least very small flocs. 1. The floc break up hypotheses led to the expectation that high turbidity suspensions would have significantly higher settled water turbidity than low turbidity suspensions. This expectation was also not observed.

Evidence that the charge neutralization hypothesis doesnt explain flocculation of surface waters has been accumulating for decades. *Sweep* flocculation has been proposed as an alternative category that described common observations that didnt fit the charge neutralization hypothesis. However, similar to the charge neutralization hypothesis, the *sweep* hypothesis didnt result in the development of predictive equations to describe the process.

For example, in 1992 Ching, Tanaka, and Elimelech published their research on Dynamics of coagulation of kaolin particles with ferric chloride. They found that the electrophoretic mobility which is a measure of the clay particle surface charge was never neutralized at pH 7.8 and was neutralized at $10\mu M$ at pH 6.0. The results were interpreted by the authors to mean that some combination of sweep floc and charge patchiness was responsible for the observed results.

Figure x. Electrophoretic_Mobility for final pH (after coagulant addition) of 6.0 and 7.8 as a function of :math:'FeCl_3 dose <https://doi.org/10.1016/0043-1354(94)90007-8>'__

The settled water turbidity was almost independent of pH even though the electrophoretic mobility was quite different for the two pH values tested.

At pH 6.0 the ferric hydroxide precipitates are positively charged and at pH 7.8 they are close to neutral. Thus it is apparent that neutralization of the clay surface charge can not explain these results.

Figure x. Settled water turbidity (jar tests) for final pH (after coagulant addition) of 6.0 and 7.8.

Electrostatic charge neutralization hypothesis * The coagulant precipitate self aggregates – this is inconsistent with the positive charge that the electrostatic hypothesis asserts will prevent aggregation * Electrostatic repulsion extends only a few nm from the surface of a particle – and the coagulant adhesive nanoparticles are many times larger than the reach of the repulsive electrostatic force * The hypothesis that London van der Waals forces result in attachment neglects to account for the presence of water in the system. Water molecules will also be attracted to surfaces by London van der Waals forces and thus there will be competition between the coagulant and water. Thus eliminating repulsion is NOT sufficient to produce a bond between the particles. (see hydration repulsion, page 21) * *The theory of DLP was a great step forward in that it appeared to circumvent the whole intractable problem of many body forces through its use of measured bulk dielectric response functions. However, it must be stressed again that it is a perturbation theory. That is, it depends on the assumption that an intervening liquid between interacting surfaces has bulk liquid properties up to a molecular distance from the surfaces. This is thermodynamically inconsistent, being equivalent to the statement that surface energies (or alternatively, the positions of the Gibbs dividing surfaces) are changed infinitesimally with* *distance of separation. This limits the theory to ‘‘large distances (Young–Laplace vs. Poisson again) where* ‘large is undefined. <https://doi.org/10.1016/S0001-8686(99)00008-1>‘__

```
# %%
#Assumptions
Pi_VC = .62 #Vena contracta coefficient of an orifice
Ke = ((1/Pi_VC**2)-1)**2 #expansion coefficient

#Functions to calculate key parameters

def Gave(G_theta,h_floc,Temp):
    """Calculates average G given target minimum collision potential, total headloss,␣
↪and design temperature
    equation from flocculation slides"""
    G_ave = (pc.gravity*h_floc/(G_theta*pc.viscosity_kinematic(Temp))).to(1/u.s)
    return G_ave

def restime(G_theta,G_ave):
    """Calculates residence time given collision potential and average G
    equation from flocculation slides"""
    theta = G_theta/G_ave
    return theta


def Dpipe(Ke,Pi_HS,Q,G_ave,Temp,SDR):
    """Calculates the actual inner diameter of the pipe
    equation from flocculation slides"""
    D_pipe = ((Ke/(2*Pi_HS*pc.viscosity_kinematic(Temp)*G_ave**2))*(4*Q.to(u.m**3/u.
↪s)/np.pi)**3)**(1/7)
    return D_pipe

def Keactual(ID_pipe,G_ave,Temp,Pi_HS,Q):
    """estimates actual expansion coefficient given the actual inner diameter and␣
↪other relevant inputs
    equation from flocculation slides"""
    Ke_actual = np.pi**3*ID_pipe**7*G_ave**2*pc.viscosity_kinematic(Temp)*Pi_HS/(32*Q.
↪to(u.m**3/u.s)**3)
    return Ke_actual



def Aorifice(ID_pipe,Ke_actual,Temp,Q):
    """Calculates the orifice area given pipe inner diameter, expansion coefficient,␣
↪Temperature, and flow"""
    A1 = (pc.area_circle(ID_pipe)).to(u.cm**2).magnitude #Pipe area
    Nu = pc.viscosity_kinematic(Temp) #kinematic viscocity
    Re = pc.re_pipe(Q,ID_pipe,Nu) #reynolds number

    def f_orif(A2,A1,Ke_actual,Re): #root of this function is the orifice area
        return (2.72+(A2/A1)*(4000/Re))*(1-A2/A1)*((A1/A2)**2-1)-Ke_actual

    A_orifice = (brentq(lambda A2: f_orif(A2,A1,Ke_actual,Re), -1, 2*A1))*u.cm**2
↪#numerical optimization

    return A_orifice


def eave(G_ave,Temp):
```

(continues on next page)

(continued from previous page)

```
    """Calculates the average energy dissipation rate"""
    e_ave = (pc.viscosity_kinematic(Temp)*G_ave**2).to(u.mW/u.kg)
    return e_ave

def Hchip(A_orifice,ID_pipe):
    """This function calculates the height of the chip based on the orifice area and␣
↪pipe diameter
    The function uses numerical optimization to solve the transcendental equation"""
    A_flow = A_orifice.magnitude #orifice area stripped of units
    r=(ID_pipe/2).magnitude #radius stripped of units
    c = A_flow/r**2 #left hand side of equation

    def f(a,c): #roots of this function are theta
        return a-sin(a)*cos(a)-c

    theta = brentq(lambda a: f(a,c), 0, 13) #numerical optimization
    r_u = r*u.cm #radius with units
    y = r_u - r_u*np.cos(theta) #height of orifice

    H_chip = ID_pipe-y #height of chip
    return H_chip

def Cost_Length(L_pipe,ND_pipe):
    """This function calculates the total cost of the system and the total length of␣
↪the system"""
    #Length of pipe and number of fittings needed
    OD_pipe = pipe.OD(ND_pipe)
    Total_Pipe = L_pipe + .5*u.m
    Number_T = np.ceil(Total_Pipe.magnitude)
    Number_Elbow = np.ceil(Total_Pipe.magnitude)

    if ND_pipe.magnitude == 3:
        Cost_T = 3.94*u.dollar
        Cost_Elbow = 3.53*u.dollar
        Cost_Pipe = (17.14/10*(u.dollar/u.foot)).to(u.dollar/u.m)
        Cost_Valve = 10*u.dollar
        Width_T = (3.99*u.inch).to(u.cm)
        Width_Elbow = (3.97*u.inch).to(u.cm)


    if ND_pipe.magnitude ==4:
        Cost_T = 7.16*u.dollar
        Cost_Elbow = 5.40*u.dollar
        Cost_Pipe = (21.5/10*(u.dollar/u.foot)).to(u.dollar/u.m)
        Cost_Valve = 10*u.dollar
        Width_T = (5.06*u.inch).to(u.cm)
        Width_Elbow = (5.06*u.inch).to(u.cm)

    if ND_pipe.magnitude ==6:
        Cost_T = 7.16*u.dollar
        Cost_Elbow = 5.40*u.dollar
        Cost_Pipe = (21.5/10*(u.dollar/u.foot)).to(u.dollar/u.m)
        Cost_Valve = 10*u.dollar
        Width_T = (5.06*u.inch).to(u.cm)
        Width_Elbow = (5.06*u.inch).to(u.cm)
```

(continues on next page)

(continued from previous page)

```
    Total_Cost = Cost_Pipe*Total_Pipe + Cost_T*Number_T + Cost_Elbow*Number_Elbow +␣
↪Cost_Valve*Number_Elbow
    Floor_Length = Number_T*(Width_T+Width_Elbow-OD_pipe).to(u.m)
    Output=[Total_Cost,Floor_Length]
    return Output
```

```
#Inputs
D_Sed = 2.5*u.cm
A_Sed = pc.area_circle(D_Sed)
v_Sed = 2*u.mm/u.s
Q = (v_Sed*A_Sed).to(u.mL/u.s)
print('The flow rate is',Q)

Temp = 15*u.degC
h_floc = 50*u.cm #standard for Aguaclara plants
G_theta = 20000 #standard for Aguaclara plants
Pi_HS = 6  ##3-6 is a good range, more research needed
SDR = 41 #Standard ratio
```

```
#Calculate G average using functions listed above and given inputs
G_ave = Gave(G_theta,h_floc,Temp)
theta = restime(G_theta,G_ave)
e_ave = eave(G_ave,Temp)
print('The average G value is ',G_ave)
print('The residence time in the flocculator is ',theta)
print('The average energy dissipation rate is ', e_ave)
```

```
#Calculate the pipe diameter, both inner and nominal and determine area of pipe using␣
↪inner diameter output
D_pipe = (Dpipe(Ke,Pi_HS,Q,G_ave,Temp,SDR)).to(u.cm)
#Calculate nominal diameter of pipe
ND_pipe = pipe.ND_SDR_available(D_pipe,SDR)
#Calculate nominal diameter of pipe
ID_pipe = pipe.ID_SDR(ND_pipe,SDR).to(u.cm)

ID_pipe = 5*u.mm
#Calculate inner diameter of pipe
A_pipe = (pc.area_circle(ID_pipe)).to(u.cm**2)

print('The ideal inner diameter of the pipe would be ',D_pipe)
print('The nominal diameter of the pipe is ',ND_pipe, ', and the inner diameter is ',␣
↪ID_pipe)
print('The area of the pipe is ', A_pipe)
```

```
#Calculate the actual Ke as a result of the calculated inner pipe diameter
Ke_actual = (Keactual(ID_pipe,G_ave,Temp,Pi_HS,Q)).to(u.dimensionless)
print('The initial expansion minor loss coefficient was ',Ke)
print('The actual expansion minor loss coefficient is ',Ke_actual)
```

```
#Calculate the orifice area
A_orifice = Aorifice(ID_pipe,Ke_actual,Temp,Q)
print('The orifice area is ',A_orifice)
```

```
# The following line of code needs to be removed once the orifice area equation is␣
↪corrected.
```

(continues on next page)

(continued from previous page)

```
H_chip = Hchip(A_orifice,ID_pipe)
print('The height of the chip is ', H_chip)
```

```
#Calculate average velocity
v_avg = (Q/pc.area_circle(ID_pipe)).to(u.m/u.s) #first calculate average velocity
print('The average velocity is ',v_avg)

#Calculate pipe length
L_pipe = (v_avg*theta).to(u.m) #then multiply velocity by residence time to get the␣
↪required length of pipe
print('The length of the pipe is ',L_pipe)
```

#references Coagulation and Flocculation in Water and Wastewater Treatment, iwapublishing

CHAPTER SEVENTEEN

FLOCCULATION DESIGN

Welcome to the **fourth** summary sheet of CEE 4540! These documents will be guides and references for you throughout the semester. Since Professor Monroes class time is limited, so too is the amount of material he can fit on the slides while ensuring that they remain understandable. Thus, these summary sheets will supplement the powerpoints by going into further detail on the course concepts introduced in the slides.

Equations, universal constants, and other helpful goodies can be found in the aide_design repository on GitHub. Most equations and constants you find in these summary sheets will already have been coded into aide_design, and will be shown here in the following format:

Variable: `pc.gravity`
Function: `pc.area_circle(DiamCircle).`

The letters before the `.`, in this case `pc`, indicate the file within aide_design where the variable or function can be found. In the examples above, `pc.gravity` and `pc.area_circle(DiamCircle)` show that the variable `gravity` and function `area_circle(DiamCicle)` are located inside the physchem.py (`pc`) file. You are strongly recommended to look up any aide_design equations you plan to use within in their aide_design file before using them, even if they are given here in this summary sheet. This is because each equation has comments in its original file describing what the specific conditions are to using it.

For the most part, hyperlinks in these documents will contain supplementary information. The information contained in the linked external sites is there in case you dont feel completely comfortable with a concept, but is not necessary to learn thoroughly and will not be tested.

Important Note: This chapter introduces uncertainty and empirical design. Some of the parameters used to design AguaClara flocculators are based on what has been shown to work in the field, as opposed to having been derived scientifically. To make sure that the reader is aware of these concepts and parameters that dont yet have a thorough basis in research, they will be highlighted in red when they appear.

17.1 Hydraulic Flocculators, the AguaClara Approach

17.1.1 Important Terms

1. Collision potential
2. Energy dissipation rate
3. Baffle
4. Baffle module

5. Baffle space
6. Obstacle

Important Equations ~

1. Minor Loss equation

17.2 Introduction to Hydraulic Flocculation

The reason that flocculation is widely used in water treatment is because of sedimentation. Sedimentation is the process that actually removes particles like clay, dirt, organic matter, and bacteria from water. As you learned in the *introduction on treatment trains*, sedimentation is the process of particles falling because they have a higher density then the water, and its governing equation is:

$$\bar{v}_t = \frac{D_{particle}^2 g}{18\nu} \frac{\rho_p - \rho_w}{\rho_w} \tag{17.1}$$

Such that:
$\bar{v}_t$ = terminal velocity of a particle, its downwards speed if it were in quiescent (still) water
$D_{particle}$ = particle diameter
ρ = density. The p subscript stands for particle, while w stands for water

To increase $\bar{v}_t$ and make sedimentation more efficient, flocculation aims to increase the diameter d of the particles. This is done by applying a coagulant to the dirty water and helping the coagulant to stick evenly to all particles during Rapid Mix ***(DOUBLE CHECK THAT THIS IS IN RAPID MIX ONCE RAPID MIX IS WRITTEN)***. Being covered in coagulant allows the particles to collide, merge, and grow bigger during flocculation.

Our goal in designing a flocculator is to facilitate particle collisions. How can we do this?

17.2.1 Collision Potential, $\bar{G}\theta$, and Energy Dissipation Rate, ε

Collision potential :math:'(bar G theta)' is a term with a very straightforward name. It represents the magnitude of potential particle collisions in a fluid. It is a *dimensionless* parameter which is often used as a performance metric for flocculators; big $\bar{G}\theta$ values indicate lots of collisions (good) while small values indicate fewer collisions (not so good). AguaClara flocculators usually aim for a collision potential of $\bar{G}\theta = 37,000$, which has worked well in AguaClara plants historically. However, this value may change as research continues. The value for collision potential is obtained by multiplying $\bar{G}$, a parameter for average fluid shear with units of $\frac{1}{[T]}$, and θ, the residence time of water in the flocculator, with units of $[T]$. θ is intuitive to measure, calculate, and understand. $\bar{G}$ is a bit more difficult. First, an intuitive explanation. Fig. 17.1 , which shows the velocity profile of flowing water.

G measures the magnitude of shear by using the velocity gradient of a fluid in space, $\frac{\Delta \bar{v}}{\Delta h}$. This is essentially the same as the $\frac{\delta u}{\delta y}$ term in fluid mechanics, which is found in the ubiquitous fluid-shear problem.

$\bar{G}$ represents the average $\frac{\Delta \bar{v}}{\Delta h}$ for the entire water volume under consideration, and is the parameter we will be using from now on. Unfortunately, it is unrealistic to measure $\frac{\Delta \bar{v}}{\Delta h}$ for every parcel of the water in our flocculator and take an average. We need to approximate $\bar{G}$ using measureable parameters.

The parameter that serves as the basis for obtaining $\bar{G}$ is ε, which represents the **energy dissipation** rate of a fluid *normalized by its mass*. The units of ε are Watts per kilogram:

$$\varepsilon = \left[\frac{W}{Kg}\right] = \left[\frac{J}{s \cdot Kg}\right] = \left[\frac{N \cdot m}{s \cdot Kg}\right] = \left[\frac{kg \cdot m \cdot m}{s^2 \cdot s \cdot Kg}\right] = \left[\frac{m^2}{s^3}\right] = \left[\frac{[L]^2}{[T]^3}\right] \tag{17.2}$$

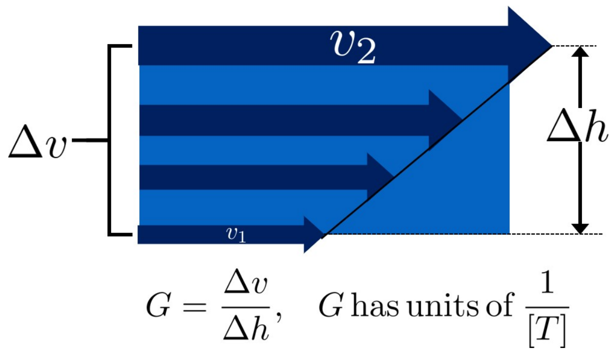


Fig. 17.1: Velocity profile for the case of uniform shear.

There are at least two ways to think about ε. One is through G. Imagine that a fluid has *no viscosity*; there is no internal friction caused by fluid flow. No matter how high G becomes, no energy is dissipated. Now image a honey, which has a very high viscosity. Making honey flow fast requires a lot of energy over a short period of time, which means a high energy dissipation rate. This explanation allows us to understand the equation for ε in terms of G and ν. See this textbook for the derivation of the following equation:

$$\varepsilon = \nu G^2 \tag{17.3}$$

Which means we can solve for G:

$$G = \sqrt{\frac{\varepsilon}{\nu}} \tag{17.4}$$

Energy dissipation rate is, fortunately, easier to determine than collision potential. This is due to the second way to think about ε, which is using head loss. In any reactor, a flocculator in this case, the total energy dissipated is simply the head loss, h_L. The amount of time required to dissipate that energy is the residence time of the water in the reactor, θ. Accounting for the fact that head energy is due to gravity g, we have all the parameters needed to determine another equation for energy dissipation rate:

$$\bar{\varepsilon} = \frac{g h_L}{\theta} \tag{17.5}$$

Note that the equation above is for $\bar{\varepsilon}$, not ε. Since the head loss term we are using, h_L, occurs over the entire reactor, it can only be used to find an average energy dissipation rate for the entire reactor. Combining the equations above, $G = \sqrt{\frac{\varepsilon}{\nu}}$ and $\bar{\varepsilon} = \frac{g h_L}{\theta}$, we can get an equation for $\bar{G}$ in terms of easily measureable parameters:

$$\bar{G} = \sqrt{\frac{g h_L}{\nu \theta}} \tag{17.6}$$

We can use this to obtain a final equation for collision potential of a reactor:

$$\bar{G}\theta = \sqrt{\frac{g h_L \theta}{\nu}} \tag{17.7}$$

Note: When we say $G\theta$ we are almost always referring to $\bar{G}\theta$.

17.2.2 Generating Head Loss with Baffles

What are Baffles?

Now that we know how to measure collision potential with head loss, we need a way to actually generate head loss. While both major or minor losses can be the design basis, it generally makes more sense to use major losses only for very low-flow flocculation (lab-scale) and minor losses for higher flows, as flocculation with minor losses tends to be more space-efficient. Since this book focuses on town and village-scale water treatment (5 L/S to 120 L/S), we will use minor losses as our design basis.

To generate minor losses, we need to create flow expansions. AguaClara does this with **baffles**, which are obstructions in the channel of a flocculator to force the flow to switch directions by 180ř. Baffles in AguaClara plants are plastic sheets, and all of the baffles in one flocculator channel are connected to form a **baffle module.** Fig. 17.2 shows an AguaClara flocculator and Fig. 17.3 shows the assembly of a baffle module.

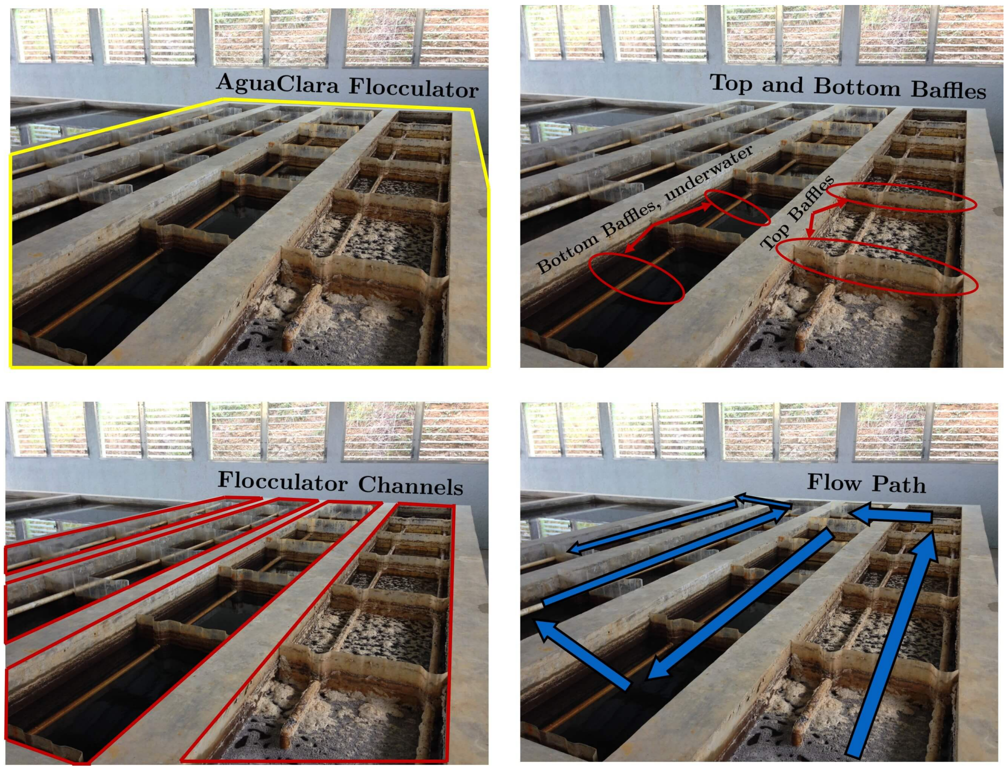


Fig. 17.2: AguaClara vertical flow hydraulic flocculator.

AguaClara flocculators, like the one pictured above, are called **vertical hydraulic flocculators** because the baffles force the flow vertically up and down. If the baffles were instead arranged to force the flow side-to-side, the flocculator would be called a **horizontal hydraulic flocculator**. AguaClara uses vertical flocculators because they are more efficient when considering plant area. They are deeper than horizontal flocculators, which allows them to have a smaller plan-view area and thus to be cheaper.

Finding the Minor Loss of a Baffle

Before beginning this section, it is important to make sure that the we understand how water flows through a baffled flocculator (see Fig. 17.4). Take note of the red lines, they indicate the compression of the flow around a baffle.

Since baffles are the source of head loss via minor losses, we need to find the minor loss coefficient of one baffle if we want to be able to quantify its head loss. To do this, we apply fluid mechanics intuition and check it against a computational fluid dynamics (CFD) simulation. Flow around a 90ř bend has a vena contracta value of around $\Pi_{vc} = 0.62$. Flow around a 180ř bend therefore has a value of $\Pi_{vc,\, baffle} = \Pi_{vc}^2 = 0.384$. This number is roughly confirmed with CFD, as shown in the image below.

We can therefore state with reasonable accuracy that, when most contracted, the flow around a baffle goes through 38.4% of the area it does when expanded, or $A_{contracted} = \Pi_{vc,\, baffle} A_{expanded}$. Through the *third form of the minor*

Fig. 17.3: AguaClara baffle module for a hydraulic flocculator.

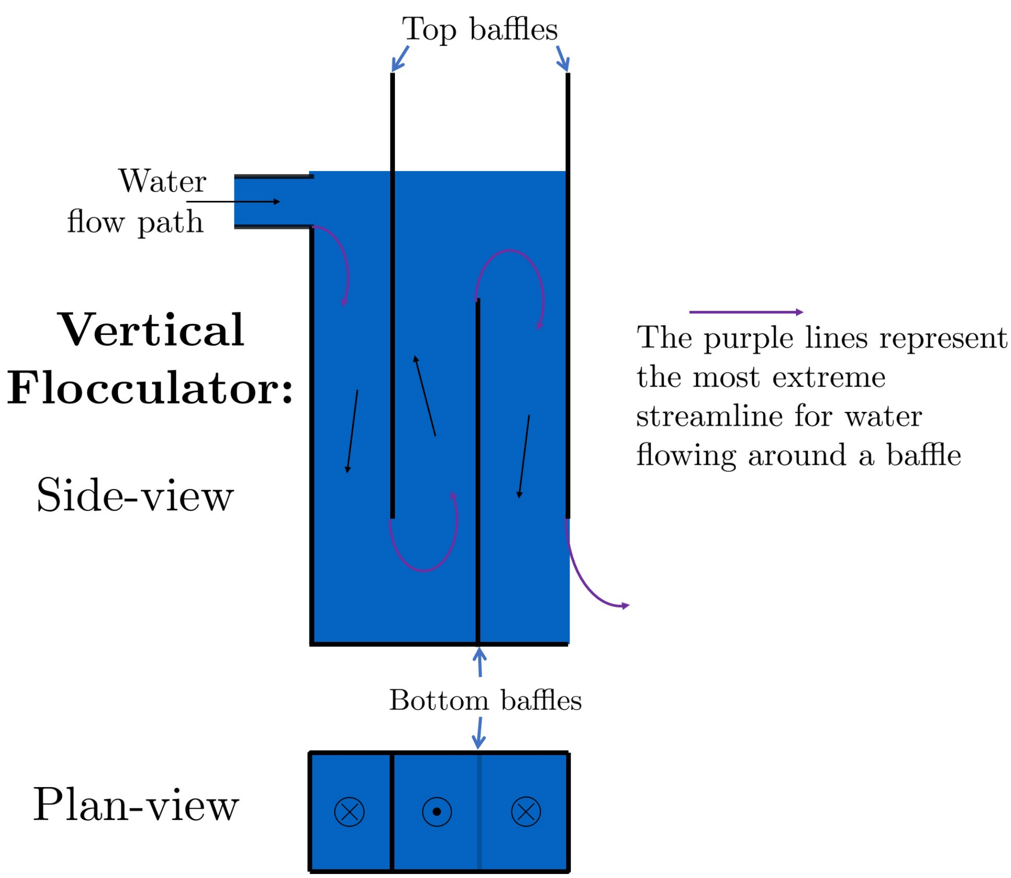


Fig. 17.4: Flow path through a vertical flow hydraulic flocculator.

loss equation, $h_e = K\frac{\bar{v}_{out}^2}{2g}$ and its definition of the minor loss coefficient, $K = \left(\frac{A_{out}}{A_{in}} - 1\right)^2$, we can determine a K for flow around a single baffle:

$$K_{baffle} = \left(\frac{A_{expanded}}{A_{contracted}} - 1\right)^2 \tag{17.8}$$

$$K_{baffle} = \left(\frac{\cancel{A}_{expanded}}{\Pi_{vc,\,baffle}\cancel{A}_{expanded}} - 1\right)^2 \tag{17.9}$$

$$K_{baffle} = \left(\frac{1}{0.384} - 1\right)^2 \tag{17.10}$$

$$K_{baffle} = 2.56 \tag{17.11}$$

This K_{baffle} has been used to design many flocculators in AguaClara plants. However, its value has not yet been rigorously tested for AguaClara plants the field. Therefore it might actually deviate from 2.56. Research and testing the K of a baffle in an AguaClara plant is ongoing, but for now the designs made under the assumption that $K_{baffle} = 2.56$ are functioning very well in AguaClara plants. Although research has been done by many academics on the minor loss coefficient, including this paper by Haarhoff in 1998, the K_{baffle} values found are context dependent and empirically based. For AguaClara flocculator parameters, literature suggest a K_{baffle} value between 2.5 and 4.

Flocculator Efficiency ~

When designing an effective and efficient flocculator, there are two main problems that we seek to avoid:

1. Having certain sections in the flocculator with such high local G values that our big, fluffy flocs are sheared apart into smaller flocs. 1. Having dead space. Dead space means volume within the flocculator that is not being used to facilitate collisions. Dead space occurs after the flow has fully expanded from flowing around a baffle and before it reaches the next baffle.

Fortunately for us, both problems can be quantified with a single ratio:

$$\Pi_{\bar{G}}^{G_{Max}} = \frac{G_{Max}}{\bar{G}} \tag{17.12}$$

High values of $\Pi_{\bar{G}}^{G_{Max}}$ occur when one or both of the previous problems is present. If certain sections in the flocculator have very high local G values, then G_{Max} becomes large. If the flocculator has a lot of dead space, then $\bar{G}$ becomes small. Either way, $\Pi_{\bar{G}}^{G_{Max}}$ becomes larger.

Note: Recall the relationship between G and ε: $G = \sqrt{\frac{\varepsilon}{\nu}}$. From this relationship, we can see that $G \propto \sqrt{\varepsilon}$. Thus, by defining $\Pi_{\bar{G}}^{G_{Max}}$, we can also define a ratio for Max to average energy dissipation rate:

$$\Pi_{\bar{\varepsilon}}^{\varepsilon_{Max}} = \left(\Pi_{\bar{G}}^{G_{Max}}\right)^2 \tag{17.13}$$

Therefore, by making our $\Pi_{\bar{G}}^{G_{Max}}$ as small as possible, we can be sure that our flocculator is efficient, and we no longer have to account for the previously mentioned problems. A paper by Haarhoff and van der Walt in 2001 uses CFD to show that the minimum $\Pi_{\bar{G}}^{G_{Max}}$ attainable in a hydraulic flocculator is $\Pi_{\bar{G}}^{G_{Max}} = \sqrt{2} \approx 1.4$, which means that $\Pi_{\bar{\varepsilon}}^{\varepsilon_{Max}} = \left(\Pi_{\bar{G}}^{G_{Max}}\right)^2 \approx 2$. So how do we optimize an AguaClara flocculator to make sure $\Pi_{\bar{G}}^{G_{Max}} = \sqrt{2}$?

We define and optimize a performance metric:

$$\frac{H_e}{S} = \Pi_{H_eS} \tag{17.14}$$

Where H_e is the distance between flow expansions in the flocculator and S is the spacing between baffles. For now, H_e is approximated as the height of water in the flocculator.

Since G_{Max} is determined by the fluid mechanics of flow around a baffle, our main concern is eliminating dead space in the flocculator. We do this by placing an upper limit on $\frac{H_e}{S}$. To determine this upper limit, we need to find the distance it takes for the flow to fully expand after it has contracted around a baffle. We base this on the rule of thumb for flow expansion, ***RESEARCHED BY GERHART JIRKA FIND A REFERENCE THATS BETTER THAN ONE OF MONROES POWERPOINTS***: a jet doubles its initial diameter/length once it travels 10 times the distance of its original diameter/length. If this is confusing, refer to the equation and image below:

$$\frac{x}{10} = D - D_0 \tag{17.15}$$

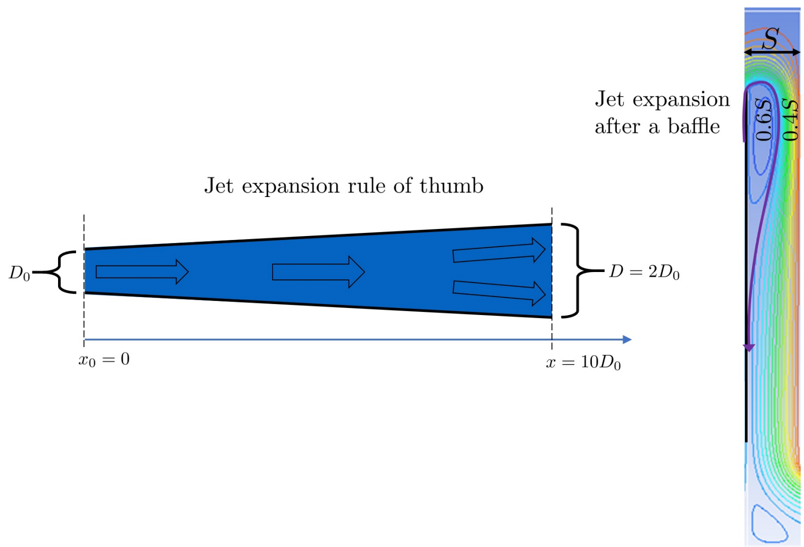


Fig. 17.5: This is a caption.

Using the equation and image above, we can find the distance required for the flow to fully expand around a baffle as a function of baffle spacing S. We do this by substituting $D_0 = (0.384S)$ along with $D = S$ to approximate how much distance, $x = H_e$, the contracted flow has to cover.

$$\frac{H_e}{10} = S - (0.384S) \tag{17.16}$$

$$\frac{H_e}{10} = 0.616S \tag{17.17}$$

$$H_e = 6.16S \tag{17.18}$$

$$\frac{H_e}{S} = 6.16 \tag{17.19}$$

$$\Pi_{H_eS_{Max}} = \frac{H_e}{S} = 6.16 \approx 6 \tag{17.20}$$

This is the highest allowable Π_{H_eS} that we can design while ensuring that there is no dead space in the flocculator.

In order to have a robust design process for a baffle module, we need to have some flexibility in the $\Pi_{H_eS} = \frac{H_e}{S}$ ratio. Since we found $\Pi_{H_eS_{Max}}$ previously, we must now find the lowest functional $\frac{H_e}{S}$ ratio, $\Pi_{H_eS_{Min}}$.

AguaClara uses a fairly straightforward way of setting $\Pi_{H_eS_{Min}}$. It is based on the distance between the water level and the bottom baffle (which is the same distance between the flocculator floor and a top baffle). This distance is referred to as the slot width Haarhoff 1998 and is defined by the slot width ratio, which describes the slot width as a function of baffle spacing S. Slot width is shown in the following image:

AguaClara uses a slot width ratio of 1 for its flocculators. This number has been the topic of much hydraulic flocculation research, and values between 1 and 1.5 are generally accepted for hydraulic flocculators. See the following paper and book respectively for more data on slot width ratios and other hydraulic flocculator parameters: Haarhoff 1998, Shulz and Okun 1984. We base our slot width ratio of 1 on research done by Haarhoff and van der Walt in 2001 on optimizing hydraulic flocculator parameters to maximize flocculator efficiency.

The minimum Π_{H_eS} allowable depends on the slot with ratio. If Π_{H_eS} is less than twice the slot width ratio, the water would flow straight through the flocculator without having to bend around the baffles. This means that the flocculator would not be generating almost any head loss, and the top and bottom of the flocculator will largely be dead space. See the following image for an example:

Thus, $\Pi_{H_eS_{Min}}$ should be at leasts twice the slot width ratio, $\Pi_{H_eS_{Min}} = 2$. Historically, AguaClara plants have been designed using $\Pi_{H_eS_{Min}} = 3$. This adds a safety factor of sorts, ensuring that the flow does not short-circuit through the flocculator and also allowing more space for the flow to expand after each contraction.

$$\Pi_{H_eS_{Min}} = \frac{H_e}{S} = 3 \tag{17.21}$$

Finally, we describe a range of Π_{H_eS} that we can use to design an AguaClara flocculator:

$$3 < \Pi_{H_eS} < 6 \tag{17.22}$$

Obstacles

Knowing that efficient flocculators require an $\frac{H_e}{S}$ ratio that lies between 3 and 6, we need to understand how that impacts the flocculator design. Keeping $\frac{H_e}{S}$ between two specific values limits the options for baffle spacing and quantity, due to the flocculator having certain size constraints before beginning the design of the baffles. This limitation places an upper limit on the amount of head loss that a baffled flocculator can generate, since the number of baffles is limited by space and baffles are what cause head loss. This is unfortunate, it means that baffled flocculators under certain size specifications cant be designed to generate certain values of $\bar{\varepsilon}$ and $\bar{G}$ *while remaining efficient and maintaining* $3 < \Pi_{H_eS} < 6$. This problem only arises for low flow plants, usually below $Q_{Plant} = 20\frac{\text{L}}{\text{s}}$

To get around this problem, AguaClara included obstacles, or half-pipes to contract the flow after the flow expands around one baffle and before it reaches the next baffle. The purpose of these obstacles is to provide extra head loss in between baffles. They also generate head loss via minor losses, *and one obstacle is designed to have the same :math:'K' as one baffle*. Introducing obstacles slightly alters how we think about H_e. In a flocculator where there are just baffles and no obstacles, then $H_e = H$, since the height of water in the flocculator is equal to the distance between expansions. When obstacles are added, however, then $H_e = \frac{H}{1+n_{obstacles}}$, where $n_{obstacles}$ is the number of obstacles between two baffles.

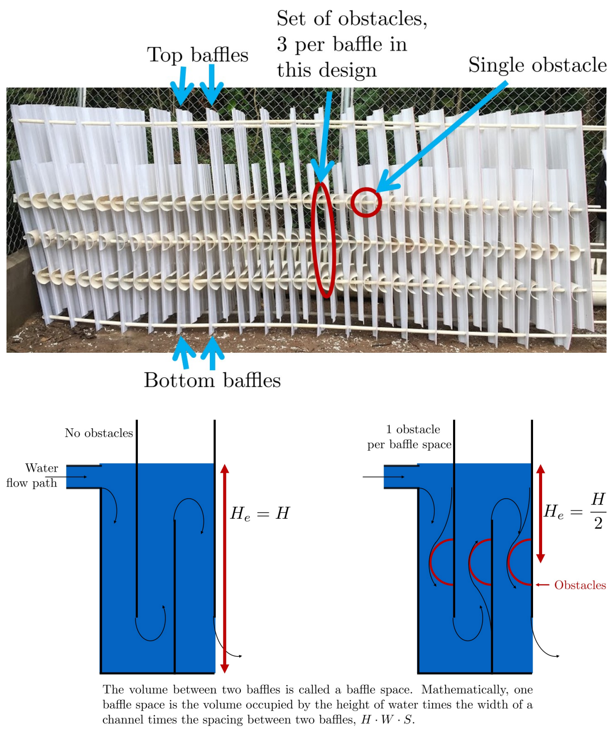
Baffle space is the term we use for the space between two baffles. The number of flow expansions per baffle space is $n_{expansions} = 1 + n_{obstacles}$. The 1 is because the baffle itself causes a flow expansion.

These obstacles serve as pseudo-baffles. They allow for $\frac{H}{S}$ to exceed 6, while maintaining maximum flocculator efficiency since, $\frac{H_e}{S}$ can still be between 3 and 6. Obstacles make it possible to design smaller flocculators without compromising flocculation efficiency. Fig. 17.2.2 and Fig. 17.2.2 show these obstacles and how they affect the flow in a flocculator.

Images/Floc_flow_with_obstacles.jpg

17.3 AguaClara Design of Hydraulic, Vertical Flow Flocculators

AguaClaras approach to flocculator design is the same as it is for any other unit process. First, critical design criteria, called inputs, are established. These criteria represent the priorities that the rest of the design will be based around.



Once these parameters are established, then the other parameters of the design, which are dependent on the inputs, are calculated based on certain constraints.

Take the CDC as an example of this design process in *Flow Control and Measurement Design*; its inputs are $h_{L_{Max}}$, $\sum K$, Π_{Error}, and the discrete dosing tube diameters D that are available at hardware stores or pipe suppliers. Its dependent variables include the number and length of the dosing tubes and the flow through the CDC system.

The flocculator is more complex to design than the CDC, as it has more details and parameters and the equations for those details and parameters are very interdependent. Therefore, there are many ways to design an AguaClara flocculator, and many different sets of critical design criteria to begin with. Enumerated below is the current AguaClara approach.

1. Input parameters

   - Specify:

     - $h_{L_{floc}}$, head loss
     - $\bar{G}\theta$, collision potential
     - Q, plant flow rate
     - H, height of water *at the end of the flocculator*
     - $L_{Max,\, sed}$, max length of a flocculator channel based on sedimentation tank length
     - $W_{Min,\, human}$ minimum width of a single channel based on the width of the average human hip (someones got to go down there)

   - Find:

     - $\bar{G}$, average velocity gradient
     - θ, hydraulic retention time
     - $\forall_{floc}$, flocculator volume

2. Physical dimensions
    - Calculate:
        - $L_{channel}$, actual channel length
        - $n_{channels}$, amount of channels
        - $W_{channel}$, actual channel width
3. Hydraulic parameters
    - Calculate:
        - H_e, distance between baffle/obstacle induced flow expansions
        - $n_{obstacles}$, amount of obstacles per baffle space
        - S, baffle spacing, distance between baffles

Flocculator_physical_parameters.jpg

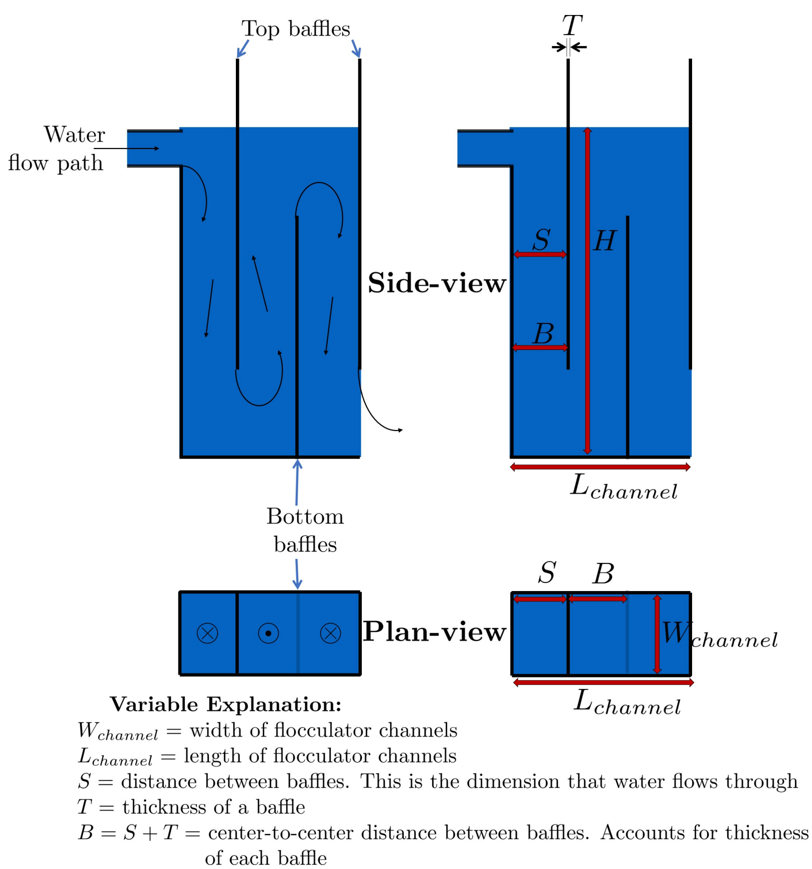


Fig. 17.6: This is a caption.

Input Parameters ~

Specify

We start by making sure that our flocculator will be able to flocculate effectively by defining $h_{L_{floc}}$ and $\bar{G}\theta$. Fixing these two parameters initially allows us to easily find all other parameters which determine flocculator performance. Here are the current standards in AguaClara flocculators: - $h_{L_{floc}} = 40\,\text{cm}$ - $\bar{G}\theta = 37,000$

The plant flow rate Q is defined by the needs of the community that the plant is being desiged for. Additionally, the height of water *at the end* of the flocculator, H, the *maximum* length of the flocculator based on the length of the sedimentation tank length, $L_{Max,\, sed}$, and the *minimum* width of a flocculator channel required for a human to fit inside, $W_{Min,\, human}$, are also defined initially. Ordinarilly in AguaClara plants, the flocculator occupies the same length dimension as the sedimentation tanks, which is why the length constraint exists. See the image below for a representation of how the flocculator and sedimentation tanks are placed in a plant.

- $H = 2\,\mathrm{m}$
- $L_{Max,\, sed} = 6\,\mathrm{m}$
- $W_{Min,\, human} = 45\,\mathrm{cm}$

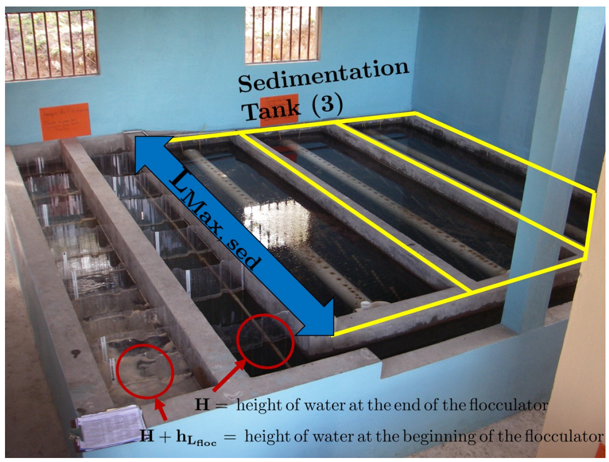


Fig. 17.7: Here is a caption

Find

We can rearrange the equation for $\bar{G}$ from the section on collision potential, $\bar{G} = \sqrt{\frac{gh_L}{\nu\theta}}$, to solve for $\bar{G}$ in terms of $\bar{G}\theta$:

$$\bar{G} = \frac{gh_{L_{floc}}}{\nu(\bar{G}\theta)} \tag{17.23}$$

Now that we have $\bar{G}$, we can very easily find θ:

$$\theta = \frac{\bar{G}\theta}{\bar{G}} \tag{17.24}$$

Finally, we take retention time θ over plant flow rate Q to get the required volume of the flocculator:

$$\forall_{floc} = \frac{\theta}{Q} \tag{17.25}$$

Now that we have the basic parameters defined, we can start to design the details of the flocculator, starting from the physical dimensions.

Physical Dimensions ~

Deriving the equations required to find the physical dimensions now and the hydraulic parameters (baffle/obstacle design) in the next section requires many steps. To simplify this design explanation the equation derivations are developed in *Review: Fluid Mechanics Derivations*. All complex equations which seemingly came out of nowhere will be derived in the derivation sheet.

Length

Flocculator length, $L_{channel}$ must meet two constraints: it must be less than or equal to the length of the sedimentation tanks, as the flocculator is adjacent to the sed tanks. This constraint is $L_{Max,\,sed}$. Next, the flocculator must be short enough to make sure the target volume of the flocculator is met, while still allowing for a human to fit inside $L_{Max,\,\rlap{-}V}$. **The constraint that wins out is the one that results in the *smaller* length value**.

.. math:: L_{Max, , sed} = 6 , {rm m} .. math:: L_{Max, , rlap{-}V} = frac{rlap{-} V}{n_{Min, , channels} W_{Min, , human} H} Such that: $n_{Min,\,channels} = 2$

The reason why $W_{Min,\,human}$ is used is because it represents the absolute minimum of flocculator channel width. If the width ends up being larger, the length will decrease. $n_{Min,\,channels} = 2$ to make sure that the flow ends up on the correct side of the sedimentation tank, as the image below shows. Note that there can only be an even number of flocculator channels, as explained in the images caption.

The equation for *actual* flocculator length is therefore:

$$L_{channel} = \min(L_{Max,\,sed},\, L_{Max,\,\rlap{-}V}) \tag{17.26}$$

There are an even amount of flocculator channels to keep the AguaClara plant layout consistent. This ensures that the entrance tank, filter box, and filters can be kept in the same places across plants. Only for exceptionally low flow plants (<5 LPS) is there one channel.

Fig. 17.8: Here is a caption

17.3.1 Width and Number of Channels

The width of a single flocculator channel must meet the following conditions: - Maintain $\bar{G}$ at the value found in the inputs section - Allow for $3 < \frac{H_e}{S} < 6$. Recall that $\frac{H_e}{S} = \Pi_{H_eS}$ - Allow for a human to be able to fit into a flocculator channel

The first two conditions are wrapped up into the following equation, *which is derived here*:

$$W_{Min,\,\Pi_{H_eS}} = \frac{\Pi_{H_eS} Q}{H_e} \left(\frac{K}{2 H_e \nu \bar{G}^2} \right)^{\frac{1}{3}} \tag{17.27}$$

This equation represents the absolute smallest width of a flocculator channel if we consider the lowest value of Π_{H_eS} and the highest possible value of H_e:

$H_e = H_{e_{Max}} = H = 2\,\mathrm{m}$, this implies that there are no obstacles between baffles
$\Pi_{H_eS} = \Pi_{HS_{Min}} = 3$

Recall our other width constraint, $W_{Min,\,human} = 45\,\mathrm{cm}$, which is based on our desire to have a human be able to fit into the channels. The governing constraint is the *larger* value of W_{Min}:

$$W_{Min} = \max(W_{Min,\,\Pi_{H_eS}},\, W_{Min,\,human}) \tag{17.28}$$

We can find the number of channels, $n_{channels}$ and their actual width in one last step, by finding the *total flocculator width* if there were no channels and dividing that by the minimum flocculator width, W_{Min}, found above. The equation for total flocculator width is based on our target volume:

$$W_{total} = \frac{\forall}{H L_{channel}} \tag{17.29}$$

Finally:

.. math:: color{purple}{ n_{channels} = frac{W_{total}}{W_{Min}} }
Such that:
$n_{channels}$ is an even number and is not 0. Usually,
$n_{channels}$ is either 2 or 4.

Now that we know $n_{channels}$, we can find the actual width of a channel, $W_{channel}$.

$$W_{channel} = \frac{W_{total}}{n_{channels}} \tag{17.30}$$

17.3.2 Hydraulic Parameters

Now that the physical dimensions of the flocculator have been defined, the baffle module needs to be designed. The parameter on which most others are based is the distance between flow expansions, H_e. Recall that $H_e = H$ when there are no obstacles in between baffles.

Height Between Expansions :math:‘H_e‘ and Number of Obstacles per Baffle Space :math:‘n_{obstacles}‘

We have a range of possible H_e values based on our window of $3 < \frac{H_e}{S} < 6$. However, we have a limitation and a preference which shape how we design H_e. Our limitation is that there can only be an integer number of obstacles. Our preference is to have as few obstacles as possible to make the baffle module as easy to fabricate as possible. Therefore, we want H_e to be closer to 6 than it is to 3; we are looking for $H_{e_{Max}}$.

We calculate $H_{e_{Max}}$ based on the physical flocculator dimensions. The equation for H_e is obtained by rearranging one of the equations for minimum channel width found above, $W_{Min,\, \Pi_{H_eS}} = \frac{\Pi_{H_eS} Q}{H_e} \left(\frac{K}{2 H_e \nu \bar{G}^2} \right)^{\frac{1}{3}}$. Because we have already design the channel width, we substitute $W_{channel}$ for $W_{Min,\, \Pi_{H_eS}}$. Since we are looking for $H_{e_{Max}}$, we also substitute $\Pi_{HS_{Max}}$ for Π_{H_eS}. The result is:

$$H_{e_{Max}} = \left[\frac{K}{2 \nu \bar{G}^2} \left(\frac{Q \Pi_{HS_{Max}}}{W_{channel}} \right)^3 \right]^{\frac{1}{4}} \tag{17.31}$$

Note that this is the *maximum* distance between flow expansions, and does not account for the limitation that there must be an integer number of obstacles per baffle space. Thus, we need to find the *actual* distance between flow expansions. To do this, we determine and round up the number of expansions per baffle space using the ceiling function:

$$n_{expansions} = \text{ceil} \left(\frac{H}{H_{e_{Max}}} \right) \tag{17.32}$$

If we had used the floor() function instead, we would find that H_e would be larger than our upper bound, $H_{e_{Max}}$. From here, we can easily get to the actual number of flow expansions per baffle spacing:

$$H_e = \frac{H}{n_{expansions}} \tag{17.33}$$

Finally, we can obtain the number of obstacles per baffle space. The -1 in the equation is because the baffles themselves provide one flow expansion per baffle space.

$$n_{obstacles} = \frac{H}{H_e} - 1 \tag{17.34}$$

Baffle Spacing :math:‘S‘

Finally, we can find the space between baffles, S. The equation for S is taken from an intermediate step *in the :math:‘W_{Min, , Pi_{H_eS}} :ref:‘derivation <title_Flocculation_Derivations>*, $W = \frac{Q}{S} \left(\frac{K}{2 H_e \nu \bar{G}^2} \right)^{\frac{1}{3}}$. Rearranging for S, we get:

$$S = \left(\frac{K}{2 H_e \bar{G}^2 \nu} \right)^{\frac{1}{3}} \frac{Q}{W_{channel}} \tag{17.35}$$

Fortunately, we either know or have already design for all the parameters in this equation

17.4 Checking the Flocculator Design

Due to the complex and interconnected nature of flocculator design, there is a chance that the parameters did not come together as intended. Now that we have calculated all of our design parameters required to build an AguaClara flocculator, we need to check that this flocculator we just designed will actually work. The three parameters we will check are:

1. Total baffle spaces in the flocculator 1. Average velocity of water in the flocculator 1. Residence time of the water in the flocculator

Total Baffle Spaces Check ~

Does our flocculator actually generate the collision potential we want it to? First, calculate how many baffle spaces are in the flocculator you designed:

$$n_{spaces,\, actual} = \text{floor} \left(\frac{L_{channel} n_{channels}}{S} \right) \tag{17.36}$$

Note: The floor( ) function is used instead of the ceil( ) function for a very good reason. Having a baffle at the end of the flocculator less than S distance from the wall creates a high velocity gradient G, which can break up the big, fluffy flocs that we worked so hard to create. So instead of risking having a spacing less than S, we have one space per channel that is slightly larger than S.

We check $n_{spaces,\, actual}$ against the amount of baffle spaces that would be required to generate the collision potential we want, $n_{spaces,\, required}$. To find $n_{spaces,\, required}$, we first find the collision potential generated in one baffle space:

$$\bar{G}\theta_{1space} = \sqrt{\frac{gh_{L_{1space}}\theta_{1space}}{\nu}} \tag{17.37}$$

$$\bar{G}\theta_{1space} = \sqrt{(n_{expansions}K)\,\frac{\bar{v}^2\theta_{1space}}{2\nu}} \tag{17.38}$$

$$\bar{G}\theta_{1space} = \sqrt{(n_{expansions}K)\,\frac{HQ}{2\nu WS}} \tag{17.39}$$

Now, we divide the total collision potential by the collision potential per baffle space:

$$n_{spaces,\, required} = \frac{\bar{G}\theta}{\bar{G}\theta_{1space}} \tag{17.40}$$

We then compare $n_{spaces,\, required}$ to $n_{spaces,\, actual}$ to make sure that they are equal.

17.4.1 Average Velocity in the Flocculator Check

As water flows through the flocculators, the flocs will get larger and larger. As a result, their terminal sedimentation velocity will increase. This is what we want. However, we need to make sure that the flocs dont settle in the flocculator; that they instead all settle in the sedimentation tank. To make sure of this, we need to make sure that the velocity of water in the flocculator is high enough to scour any flocs that fall to the bottom of the flocculator. The velocity required to scour flocs from the bottom and avoid floc accumulation is around $v_{scour} = 15\,\frac{\text{cm}}{\text{s}}$. We need to check our average velocity $\bar{v}$ against this value.

$$\bar{v} = \frac{Q}{W_{channel}S} \tag{17.41}$$

17.4.2 Residence Time of Water in the Flocculator Check

It is now time to make our final check. We need to make sure that our actual residence time is *at least* as much as we designed for. Fortunately, in our design we did not account for the change in water level throughout the flocculator due to head loss. Therefore, the actual volume of water in the flocculator is actually greater than $\forall_{floc}$. See Fig. 17.9 for clarification.

Thus, the actual average water level in the flocculator is $H + \frac{h_{L_{floc}}}{2}$. Thus, the actual residence time is:

$$\theta_{actual} = \frac{n_{channels}L_{channel}W_{channel}\left(H + \frac{h_{L_{floc}}}{2}\right)}{Q} \tag{17.42}$$

Check to see if θ_{actual} is greater than θ.

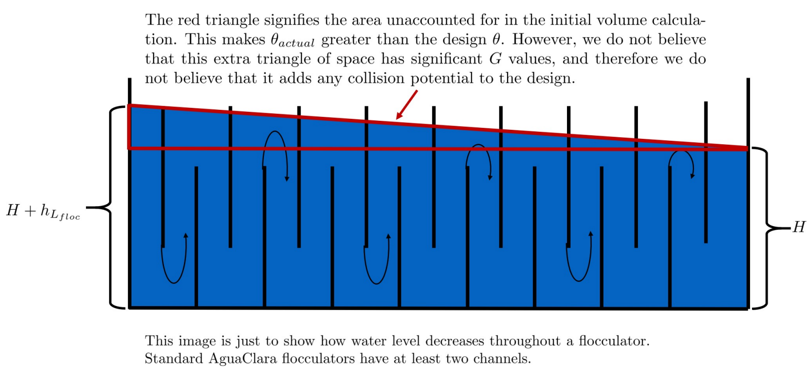


Fig. 17.9: Here is a caption

CHAPTER

EIGHTEEN

FLOCCULATION DERIVATIONS

18.1 Design Equations for the Flocculator

This document contains the derivation for the minimum allowable width of a flocculator channel based on the requirements that $3 < \Pi_{H_eS} < 6$ and that we maintain the $\bar{G}$ that serves as a basis for design. The final parameter derived is $W_{Min,\,\Pi_{H_eS}}$.

18.1.1 Width

Our two restrictions are: - Ensuring that we maintain the $\bar{G}$ we get based on our input parameters - Ensuring that $3 < \frac{H_e}{S} < 6$

First, we begin by setting the two equations for energy dissipation rate, $\bar{\varepsilon} = \nu \bar{G}^2$ and $\bar{\varepsilon} = \frac{g h_{L_{floc}}}{\theta}$ equal to each other to bring $\bar{G}$ into the equation.

$$\nu \bar{G}^2 = \frac{g h_{L_{floc}}}{\theta} \tag{18.1}$$

Very Important Note:

For the following steps, we will consider the flow through ***a single flow expansion :math:'H_e', not through the entire flocculator***. This could be from baffle to obstacle, obstacle to baffle, obstacle to obstacle, or baffle to baffle depending on how many obstacles are in the design. This means that we are briefly redefining θ to be the time it takes for the flow to fully expand after a flow contraction. θ no longer represents the time it takes for the flow to go through the entire flocculator.

From here we make three subsequent substitutions: first $h_{L_{floc}} = K\frac{\bar{v}^2}{2g}$, then $\theta = \frac{H_e}{\bar{v}}$, and finally $\bar{v} = \frac{Q}{WS}$

$$\nu \bar{G}^2 = K\frac{\bar{v}^2}{2\theta} \tag{18.2}$$

$$\nu \bar{G}^2 = K\frac{\bar{v}^3}{2H_e} \tag{18.3}$$

$$\nu \bar{G}^2 = \frac{K}{2H_e}\left(\frac{Q}{WS}\right)^3 \tag{18.4}$$

Now we can solve this equation for channel width, W.

$$W = \frac{Q}{S}\left(\frac{K}{2H_e \nu \bar{G}^2}\right)^{\frac{1}{3}} \tag{18.5}$$

From here, we can define $\Pi_{H_eS} = \frac{H_e}{S}$ and substitute $S = \frac{H_e}{\Pi_{H_eS}}$ into the previous equation for W to get $W_{Min,\,\Pi_{H_eS}}$:

$$W_{Min,\,\Pi_{H_eS}} = \frac{\Pi_{H_eS}Q}{H_e}\left(\frac{K}{2H_e\nu\bar{G}^2}\right)^{\frac{1}{3}} \tag{18.6}$$

This equation represents the absolute smallest width of a flocculator channel if we consider the lowest value of Π_{H_eS} and the highest possible value of H_e:

$H_e = H$, this implies that there are no obstacles between baffles

$\Pi_{H_eS} = 3$

CHAPTER

NINETEEN

SEDIMENTATION INTRODUCTION

The improved performance is due to 3 factors. First, the inlet manifold has a diffuser system that straightens the fluid jets that are exiting the manifold so that they have no horizontal velocity component. This is critical because even a small horizontal velocity causes a large scale circulation that transports flocs directly to the top of the sedimentation tank. Inlet manifolds without flow straightening diffusers are commonly used in vertical flow sedimentation tanks including designs by leading competitors.

Second, the diffusers create a line jet that spans the entire length of the sedimentation tank. The line jet enters a jet reverser and the vertical upward jet momentum is used to resuspend flocs that have settled to the bottom of the sedimentation tank. The resuspended flocs form a fluidized bed (floc blanket) with a suspended solids concentrations of approximately 1-5 g/L. The high concentration of particles leads to an increase in collisions and particle aggregation. The floc blanket reduces settled water turbidity by a factor of 10 (Garland et al., 2017) and provides two additional benefits. The floc blanket creates a uniform vertical velocity of water entering the plate settlers and the floc blanket transports excess flocs to a floc hopper for final removal by opening a small drain valve. Third, the bottom geometry is shaped so that all flocs that settle are transported to the jet reverser. Thus there is no accumulation of settled flocs in the main sedimentation basin. Sludge that is allowed to accumulate in the bottom of sedimentation tanks in tropical and temperate decomposes anaerobically and generates methane. The methane forms gas bubbles that carry suspended solids to the top of the sedimentation tank and cause a reduction in particle removal efficiency. The AguaClara sedimentation tank bottom geometry prevents sludge accumulation. The hydraulic self cleaning sedimentation tank with a high performing floc blanket, zero sludge accumulation, and with no moving parts outperforms conventional sedimentation tanks on capital cost, performance, and maintenance costs. Mechanical sludge removal systems are well known to be costly to install and a challenge to maintain.

19.1 Floc blankets

See the Pan American Health Organization, (PAHO) manual on theory of rapid sand filtration plants (page 289) for reasons why floc blankets should not be used! According to PAHO floc blankets are not recommended for small communities who lack highly trained personal to operate the plant and floc blanket should only be used where plant flow rates and water quality are constant. Each of these constraints was due to the inadequate design of previous floc blanket reactors that made operation difficult.

19.1.1 Floc blanket hypotheses

The floc blanket mechanism responsible for reduced settled water turbidity has been elusive. - not flocculation between particles delivered from the flocculator because $Gtheta$ generated by the shear of the settling flocs and the hydraulic residence time of the floc blanket is insufficient to cause significant - The floc blanket consists of settling flocs that are maintained in suspension by the upwardly flowing water.

19.2 Floc Hopper

19.3 Plate Settlers

19.4 Manifold Hydraulics

CHAPTER TWENTY

SEDIMENTATION EXAMPLES

Design a tube settler for a laboratory scale sedimentation tank. The vertical section of the sedimentation tank has a net upflow velocity of 3 mm/s. This velocity is maintained in the tube settler, V_α. The target capture velocity is 0.2 mm/s. The tube settler diameter is 2.54 cm.

$$\frac{\bar{v}_\uparrow}{v_c} = \frac{L}{D}\cos\alpha\sin\alpha + \sin^2\alpha \tag{20.1}$$

$$\bar{v}_\uparrow = \bar{v}_\alpha \sin\alpha \tag{20.2}$$

Solve for the length of the tube settler.

$$L = \frac{D}{\cos\alpha}\left(\frac{\bar{v}_\alpha}{\bar{v}_c} - \sin\alpha\right) \tag{20.3}$$

```
from aide_design.play import*
v_alpha = 3 * u.mm/u.s
v_c = 1 * u.mm/u.s
D = 2.54 * u.cm
alpha = 60 * u.deg

def L_settler(D,alpha,v_alpha,v_c):
  return D/np.cos(alpha)*(v_alpha/v_c - np.sin(alpha))

print(L_settler(D,alpha,v_alpha,1*u.mm/u.s))
print(L_settler(D,alpha,v_alpha,0.2*u.mm/u.s))
```

The tube settler above the floc hopper needs to be 72 cm long. The tube settler should provide a capture velocity of at least 1 mm/s prior to the floc hopper. Thus there should be 11 cm below the floc hopper.

CHAPTER

TWENTYONE

SEDIMENTATION THEORY AND FUTURE WORK

21.1 Floc recycle

We hypothesize that the flocs in floc blankets serve as collectors that primary particles attach to. We suspect that collisions between primary particles and large flocs are possible in the sedimentation tank because the rotational velocity of the flocs is small relative to the sedimentation velocity of the flocs. If the rotational velocity of the flocs is small, then a stagnation point will exist on the floc and a finite flow of fluid will come within a primary particle radius of the floc. Thus we expect primary particle removal in floc blankets to be proportional to the number of collectors that a primary particle passes while in the floc blanket.

The number of collectors that a primary particle passes is proportional to the solids concentration (a surrogate for the number concentration of flocs), the primary particle residence time in the floc blanket, and the sedimentation velocity of the flocs. The sedimentation velocity of the flocs is important because that is what causes a relative velocity between the primary particles and the flocs.

As we have explored increasing the upflow velocity in sedimentation tanks the performance has dropped markedly. This is undoubtedly due in part to the combined effective of a very dilute floc blanket at high upflow velocities AND a low residence time for the primary particles.

Would it be possible to increase the concentration of the floc blanket and thus increase the collision rate? At 3 mm/s upflow velocity there are very few flocs that can stay in the floc blanket. We need a mechanism to transport flocs to the bottom of the floc blanket and return them again after they are carried to the top of the floc blanket.

We propose to test this by installing a settled floc recycle line. The recycle line will connect to the bottom surface of the tube settler below the location of the floc weir. From there is will carry concentrated sludge to the very bottom of the sedimentation tank where it will pass through the wall of the sedimentation tank. Increasing the amount of recycle flow will both increase the solids concentration in the floc blanket and decrease the primary particle residence time in the floc blanket.

There must be an optimal amount of recycled flocs for a floc blanket. Of course, one possiblity is that the optimal recycle is zero. Recycled flocs increase the floc blanket concentration and thus increase the rate of collisions between primary particles and flocs. The recycled flocs also decrease the residence time in the floc blanket and thus decrease the total number of collisions between primary particles and flocs. It may be more complicated than this because the hindered sedimentation velocity of the flocs in the floc blanket is also a function of their concentration.

Our goal is to find the optimal recycle ratio. Optimal is defined as the maximum collision potential. Collision potential for the floc blanket is proportional to to the collision rate times the hydraulic residence time. The collision rate is proportional to the solids concentration and the hindered sedimentation velocity of those flocs. The collision potential is thus proportional to the total number of flocs that a primary particle passes on its way through the floc blanket.

$$CP_{fb} \propto C_{fb}\theta_{fb}v_{hindered} \tag{21.1}$$

The residence time in the floc blanket is given by

$$\theta_{fb} = \frac{H_{fb}}{v_{fb}} \tag{21.2}$$

$$v_{fb} = \frac{Q_{plant} + Q_{recycle}}{A_{fb}} \tag{21.3}$$

$$Q_{recycle} = \Pi_{recycle} Q_{plant} \tag{21.4}$$

The velocity up through the floc blanket without recycle is defined as

$$v_{up} = \frac{Q_{plant}}{A_{fb}} \tag{21.5}$$

$$v_{fb} = v_{up}\left(1 + \Pi_{recycle}\right) \tag{21.6}$$

Now we need equations for the concentration in the floc blanket. This is based on mass conservation such that the mass in the floc blanket is constant. There is a hindered sedimentation velocity of the flocs that results in a reduction of the mass flux out of the top of the control volume.

$$C_{fb}\left(\frac{Q_{plant} + Q_{recycle}}{A_{fb}} - v_{hindered}\right) A_{fb} = C_{plant} Q_{plant} + C_{recycle} Q_{recycle} \tag{21.7}$$

$$C_{fb}\left(\frac{Q_{plant} + \Pi_{recycle} Q_{plant}}{A_{fb}} - v_{hindered}\frac{Q_{plant}}{Q_{plant}}\right) A_{fb} = C_{plant} Q_{plant} + C_{recycle} \Pi_{recycle} Q_{plant} \tag{21.8}$$

$$C_{fb}\left(1 + \Pi_{recycle} - \frac{v_{hindered}}{v_{up}}\right) = C_{plant} + C_{recycle}\Pi_{recycle} \tag{21.9}$$

$$C_{fb} = \frac{C_{plant} + C_{recycle}\Pi_{recycle}}{\left(1 + \Pi_{recycle} - \frac{v_{hindered}}{v_{up}}\right)} \tag{21.10}$$

Now we can substitute to get the collision potential as a function of the flow rates.

$$CP_{fb} \propto \frac{C_{plant} + C_{recycle}\Pi_{recycle}}{\left(1 + \Pi_{recycle} - \frac{v_{hindered}}{v_{up}}\right)\left(1 + \Pi_{recycle}\right)} \frac{H_{fb} v_{hindered}}{v_{up}} \tag{21.11}$$

We estimate the hindered sedimentation velocity to be 1 mm/s since that is what occurs in a 1 mm/s upflow velocity floc blanket. Ideally we would have a hindered sedimentation velocity as a function of the concentration of flocs in the floc blanket. The concentration of recycled flocs is assumed to be approximately 20 g/L based on Casey Garland measurements of the solids concentration in the floc hopper sludge.

```
from aide_design.play import*
D_fb=2.5*u.cm
A_fb = pc.area_circle(D_fb)
H_fb = 1 * u.m
v_hindered = 1 * u.mm/u.s
C_fb_conventional = 3 * u.g/u.L
C_recycle = 20 * u.g/u.L
C_plant = 100 * u.NTU
v_up = 3 * u.mm/u.s


def CP(H_fb,v_up,v_hindered,Pi_recycle,C_plant,C_recycle):
  return (H_fb*v_hindered/v_up*(C_plant+C_recycle*Pi_recycle)/((1+Pi_recycle)*(1+Pi_
↪recycle-v_hindered/v_up))).to_base_units()
Pi_recycle_max = 2
Pi_recycle = np.arange(0,Pi_recycle_max,0.1)
fig, ax = plt.subplots()
x=np.array([0,Pi_recycle_max])
yscale = (C_fb_conventional*H_fb*v_hindered/(1*u.mm/u.s)).to_base_units()
yscale
```

(continues on next page)

(continued from previous page)

```
y=np.array([1,1])*yscale
ax.plot(x,y)
ax.plot(Pi_recycle,CP(H_fb,v_up,v_hindered,Pi_recycle,C_plant,C_recycle))
imagepath = 'Sedimentation/Images/'
ax.set(xlabel='recycle ratio', ylabel='Collision Potential (kg/m^2)')
ax.legend(["no recycle at 1 mm/s","with recycle at 3 mm/s"])
fig.savefig(imagepath+'fb_recycle_ratio')
plt.show()
```

Here are the results.

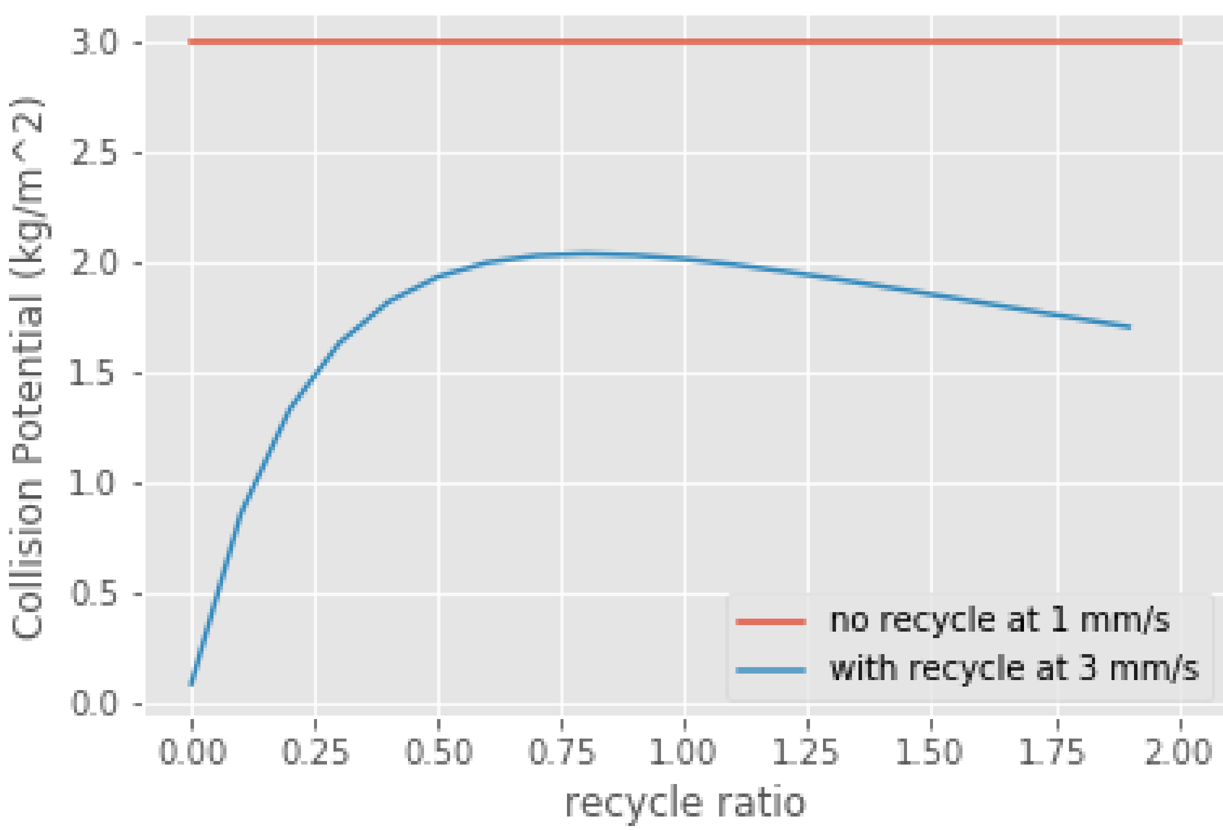


Fig. 21.1: Collision potential comparison in a 1 m deep floc blanket.

This analysis suggest that a recycle flow rate that is between 0.5 and 1.5 at a net upflow velocity of 3 mm/s could produce collision potential that is 2/3 of the collision potential with a 1 mm/s upflow velocity. Thus a 3 mm/s sed tank with 1.5 m of floc blanket and recycle might be able to perform at the same level as a 1 mm/s sed tank with a 1 m floc blanket.

The next step is to design the recycle tube. The recycle tube could be inclined to promote additional consolidation to reduce the amount of water that is recycled. The slope would need to be about 60 degrees. We could experiment with the design of the recycle line if it were made of flexible tubing.

It is expected that the consolidated sludge will flow by gravity because of its higher density. The big unknown is what diameter recycle line is needed for a lab scale test with a 2.5 cm diameter sedimentation tank.

The recycle sludge has a density given by

$$\rho_{sludge} = \left(1 - \frac{\rho_{H_2O}}{\rho_{Clay}}\right) C_{sludge} + \rho_{H_2O} \tag{21.12}$$

The piezometric head (measured in equivalent change in height of the recycle line liquid) that is causing the flow through the recycle line is equal to the difference in density between the recycled sludge and the floc blanket times the height of the floc blanket normalized by the recycle line density.

$$H_l = H_{fb} \frac{\rho_{sludge} - \rho_{fb}}{\rho_{sludge}} \tag{21.13}$$

Substitute to replace the sludge and floc blanket densities.

$$H_l = H_{fb} \frac{\left(1 - \frac{\rho_{H_2O}}{\rho_{Clay}}\right) C_{sludge} + \rho_{H_2O} - \left[\left(1 - \frac{\rho_{H_2O}}{\rho_{Clay}}\right) C_{fb} + \rho_{H_2O}\right]}{\left(1 - \frac{\rho_{H_2O}}{\rho_{Clay}}\right) C_{sludge} + \rho_{H_2O}} \tag{21.14}$$

Simplify the equation for the head loss in the recycle tube.

$$H_l = H_{fb} \frac{C_{sludge} - C_{fb}}{C_{sludge} + \frac{\rho_{H_2O}\rho_{Clay}}{\rho_{Clay} - \rho_{H_2O}}} \tag{21.15}$$

The recycle tube is assumed to be sloped at 60 degrees from the horizontal to enable further consolidation. The length of the recycle tube is

$$L_{tube} = H_{fb}/sin(60) \tag{21.16}$$

We will assume that the dynamic viscosity of the sludge is the same as the dynamic viscosity of water. We will calculate the kinematic viscosity of the sludge by dividing the dynamic viscosity of water by the density of the recycle.

Now we can solve for the required tube diameter

```
from aide_design.play import*
Temperature= 20*u.degC
D_fb=2.5*u.cm
A_fb = pc.area_circle(D_fb)
H_fb = 1.5 * u.m
Angle_tube = 60*u.deg
L_tube = H_fb/np.sin(Angle_tube)
density_clay=2650*u.kg/u.m**3

H_l = H_fb*(C_recycle-C_fb)/(C_recycle+((pc.density_water(Temperature)*density_clay)/
↪(density_clay-pc.density_water(Temperature))))
H_l
Q_plant=v_up*A_fb
Pi_recycle=0.5
density_recycle = (1 - pc.density_water(Temperature)/density_clay)*C_recycle + pc.
↪density_water(Temperature)
nu_recycle = pc.viscosity_dynamic(Temperature)/density_recycle
D_recycle = pc.diam_pipe(Q_plant*Pi_recycle,H_l,L_tube,nu_recycle,0.01*u.mm,2)
D_recycle.to(u.mm)
D_recycle.to(u.inch)
```

The head loss in the recycle tube is approximately 1.6 cm in a 1.5 m deep floc blanket.

The recycle line will be installed between the bottom of the tube settler and the inlet to the sedimentation tank. The recycle line will connect directly to the side of the sedimentation tank to minimize minor losses. We will use a 0.25

ID, 3/8OD clear flexible tube for the recycle line. We will use PVC glue to attach the flexible tube to the rigid clear PVC tubing.

It is possible that it will be necessary to prevent flow in the recycle line initially so that it doesnt flow upward. Once the tube begins filling with solids it should be possible for it to start flowing downwards.

21.2 Floc Volcanoes

Floc volcanoes are caused by differences in temperature between the water that is in a sedimentation tank and the incoming water. If the incoming water is warmer than the water that is already in the sedimentation tank, then the incoming water will be buoyant and will rise quickly to the top of the sedimentation tank and carry flocs to the effluent launder.

Temperature fluctuations can be especially pronounced with small scale water supplies where small streams and small diameter transmission lines can be exposed to the sun and can warm up dramatically oduring a few hours of sunshine. Given that temperature changes and density changes can not easily be engineered, the only solution that we have is to reduce the time that water spends in the sedimentation tank so that the influent water is closer to the average temperature of the water in the sedimentation tank. Solar heating causing the raw water temperature to go from a minimum at 6 am to a maximum at 1 pm. AguaClara sedimentation tanks currently have a residence time of approximately 2 m / (1 mm/s) or 2000 s. We anticipate that by increasing the upflow velocity and by introducing floc recycle that the effects of temperature induced floc volcanoes will be reduced.

PDF and LaTeX versions[1].

Notes

[1] PDF and LaTeX versions may contain visual oddities because it is generated automatically. The website is the recommended way to read this textbook. Please visit our GitHub site to submit an issue, contribute, or comment.