



# **Ansys Fluent Lab Exercises**

## **Investigating Potential Flow Theory through Simulation of Flow around a Circular Cylinder**

Susannah Cooke

Edited by the Ansys Academic Development Team

[education@ansys.com](mailto:education@ansys.com)

## Overview

This virtual lab experiment simulates steady flow around a circular cylinder at a range of Reynolds Numbers and investigates the resulting pressure and velocity fields. These results are then compared to the predictions of potential flow theory, and the reasons for the differences are discussed.

## Learning Objectives

Upon completion of this lab, students will be able to:

1. Explain the assumptions and limitations of Potential Flow Theory
2. Perform a flow simulation around a circular cylinder using Ansys Fluent and PyFluent
3. Have an awareness of basic concepts of simulation, CFD, and scripting

## Student Knowledge Prerequisites

This lab is written assuming that students have already been introduced to Potential Flow Theory, but have not yet had a chance to test its assumptions.

## PyAnsys and Scripting with Ansys

This resource utilizes the open source PyFluent library, developed by Ansys as part of the PyAnsys initiative to allow Ansys simulation tools to be controlled programmatically through Python. This allows the power of simulation and visualization to be presented to students in a simpler interface for introductory courses, such as here where a simulation can be run, and velocity and pressure fields plotted, from inside a Jupyter Lab environment – without needing to display the Ansys Fluent user interface (though this has been presented as an option if desired).

## System/Library Requirements

In order to run, the following software and libraries must be installed on the computer. (The version numbers listed below in brackets are those which were used to create this resource; while it may run successfully with other versions, this has not been tested.)

1. Ansys Fluent (release 2023 R2)
2. Jupyter Lab (version 4.0.7) or Jupyter Notebook (version 7.0.6)
3. PyFluent libraries:
  1. `ansys.fluent.core` (version 0.18.0)
  2. `ansys.fluent.visualization` (version 0.7.1)
4. These additional Python libraries:
  1. `ipywidgets` (version 8.1.1)
  2. `trame` (version 3.2.7)

If you are unfamiliar with installing Python packages, guidance can be found on the [Python website](#) or elsewhere online.

## Jupyter Lab First Time Setup

1. Open the zip file and extract its contents to a single location. The virtual lab is contained within the Jupyter Notebook file `AnsysCylinderFlowLab.ipynb`, and it uses the other files within this folder (mesh and images) as well.
2. Before giving the lab to students, we suggest running the simulation to understand how long it will take on the computers where you have it installed, especially if you wish to take advantage of the option not to show them Fluent's user interface. To do this:
  - a. Use the fast forward button in Jupyter Lab to run all cells.
  - b. Click the first tickbox to show Ansys Fluent on your screen
  - c. Click the 'Start Simulation' button below the next code cell to initialize Fluent, import the mesh and check it
  - d. Wait until the Fluent console shows 'Checking Mesh..... Done' in its console, then click the 'Run Simulation' button below code cell 5
  - e. In Fluent, you should be able to see the residuals being plotted as the simulation proceeds. When it completes, a velocity plot will also appear.
  - f. You now know how long steps c. and e. take, and can guide students expectations accordingly if they run this without showing Fluent on their screen.
3. Once you are happy that the lab runs successfully, it may then be used with students. All the instructions and information need are contained within the `.ipynb` file.
4. Depending on how much you want students to engage with the Python code, you may wish to hide the code cells (in Jupyter Lab, do this by selecting a cell and then clicking the blue vertical bar that appears to the left of it).

## Optional Extensions

There is an optional section towards the end of the lab for discussion of steady-state versus transient simulations. However, it is possible to extend this lab in multiple different ways; since the behavior is controlled by the Python code, you are free to change variables or functionality as best suits your specific learning objectives. This could be as simple as changing the input velocity range allowed, or choosing different variables to plot, but this lab could also be modified to consider different geometry, or to let students explore for themselves how to output specific variables, or implement buttons and other widgets within Jupyter – the possibilities are endless!

## Python Support

Documentation for Jupyter and ipywidgets can be easily found online. Documentation for PyFluent – and other PyAnsys libraries – can be found on the [Ansys Developer Portal](#), along with examples, support articles and a user community.

## Acknowledgments

This resource would not have been created without the assistance of many people.

I would like to acknowledge that the inspiration for this lab exercise came from the Thermofluids Lab used to teach students in the Department of Engineering, University of Oxford. Many thanks to Dr Christopher Vogel for confirming the dimensions of the problem as run in a physical wind tunnel there.

Within Ansys, I would like to acknowledge: Vishal Ganore, Olivier Ricordel, Cedric Bellanger, Sean Pearson, Aseem Jain, Ryan O'Connor and Gautham Varma.

© 2024 ANSYS, Inc. All rights reserved.

### Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

### Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

### Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit [www.ansys.com/education-resources](http://www.ansys.com/education-resources).

### Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at [education@ansys.com](mailto:education@ansys.com).

**ANSYS, Inc.**  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit [www.ansys.com](http://www.ansys.com) for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2024 ANSYS, Inc. All Rights Reserved.