



Tutorial Read Me

Converging Nozzle using PyFluent- ReadMe

Developed and curated by the Ansys Academic Development Team

Gautham Varma

education@ansys.com

1. Overview

This Simulation Tutorial gives an understanding of how-to setup a simple Convergence nozzle problem using the help of PyFluent. In this tutorial, you will be able to understand how you can use pythonic API to run simulations in Ansys Fluent®, fluid simulation software.

2. Learning Objectives

Upon completion of this tutorial

1. Understand the physics of Converging Nozzle mainly Bernoulli's Principle.
2. Understand the API used to setup a simulation using PyAnsys (in this case PyFluent).
3. Have an awareness of basic concepts of simulation, CFD, and scripting

3. Student Knowledge Prerequisites

This PyFluent tutorial is scripted assuming that students have a basic knowledge of Ansys Fluent software and knowledge of basic scripting in python.

4. 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 an Integrated Development Environment (IDE) – without needing to display the Ansys Fluent user interface (though this has been presented as an option if desired).

5. 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 software (release 2024 R2)
2. Python 3.11.4
3. PyFluent libraries:
 - a. `ansys.fluent.core` (version 0.24.2)
 - b. `ansys.fluent.visualization` (version 0.12.0)
4. These additional Python libraries:
 - a. `trame` (version 3.6.3)

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

6. First Time Setup

- The Python scripts could be run using any of the IDEs like Visual Studio Code, Jupyter Notebook, or Spyder. Please make sure environment has access to the versions of the libraries and python as mentioned in the previous sections.
- The zip file contains a readme file, the python script and a mesh file. Please make sure that the python script and the mesh file are in the same folder before you attempt to run the case using the IDE.
- Please also read through the texts available before running the code to understand the steps. This is important so that if you have older versions installed you can change the scripts accordingly.

7. Python Support

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

© 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 tutorial is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansyes 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.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
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 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.