Open Source CFD

# Tutorial #3: Orifice plate

## Axisymmetrical Flow: Orifice Plate with simpleFoam

Robert Castilla
Department of Fluid Mechanics

2025-26
Version 1.0
Learning Objectives:

- Create a blockMesh dictionary with python
- Extrude the mesh to create a axisymmetrical simmulation
- Run a simple steady-state simulation
- Monitor residuals from terminal
- Analyze basic results in terminal and paraview
- Validate results with standard publication

This tutorial introduces the use of a python interface to create a blockMesh dictionary. The study case is the a simple orifice plate, which will be simulated with `simpleFoam`. It is not available in tutorials and it will be generated from a template case. Pressure loss and dimensionless coefficient will be obtained both with post-processing utilities in terminal and with paraview.

## Contents

# 1 Introduction and Theoretical Background

## 1.1 What is an orifice plate?

An orifice plate is a device used to experimentally measure the flow rate in a pipe. It is basically a circular plate, with an orifice in the center, that restricts the flow and produce a pressure loss. This pressure loss is mathematically related to the flow rate in the pipe.

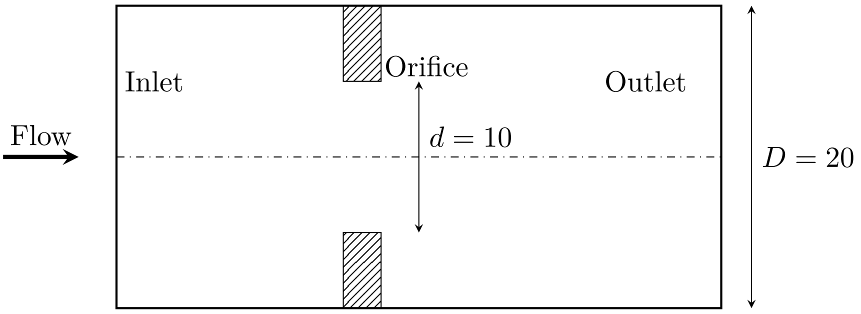

Figure 1: Orifice plate geometry. Dimensions are in mm.

More information can be found on Wikipedia.

## 1.2 Flow rate measurement

In the orifice plate flow rate is computed with equation (1)

$$Q = C_d \frac{A_o}{\sqrt{1-\beta^4}} \sqrt{\frac{2\Delta p}{\rho}} \tag{1}$$

where $\beta = d/D$,$A_o$ is the area of the orifice, $\rho$ is the density of the fluid , $\Delta p$ is the pressure loss in the orifice and $C_d$ is the **coefficient of discharge**. The approximate value of $Cd$ is 0.6, but it can be more precisely estimated with the Reader-Harris/Gallagher equation

$$
\begin{aligned}
C_d =& 0.5961 + 0.0261\beta^2 - 0.216\beta^8 + 0.000521\left(\frac{10^6\beta}{Re_D}\right)^{0.7} \\
&+ (0.0188 + 0.0063A)\beta^{3.5}\left(\frac{10^6}{Re_D}\right)^{0.3} \\
&+ (0.043 + 0.080e^{-10L_1} - 0.123e^{-7L_1})(1 - 0.11A)\frac{\beta^4}{1-\beta^4} \\
&- 0.031(M_2' - 0.8M_2'^{1.1})\beta^{1.3} + 0.011(0.75 - \beta)\left(2.8 - \frac{D}{0.0254}\right)
\end{aligned}
\tag{2}
$$

where

$$
\begin{aligned}
\beta &= d/D \quad \text{(diameter ratio)} \\
Re_D &= \text{Reynolds number based on pipe diameter} \\
A &= \left(\frac{19000\beta}{Re_D}\right)^{0.8} \\
M_2' &= \frac{2L_2'}{1-\beta}
\end{aligned}
$$

The value of $\Delta p$ depends on where the pressure are measured upstream and downstream with relation to the orifice. In this case we are going to measure the pressure upstream ($p_1$) at a distance $D$ and downstream ($p_2$) at a distance $\frac{D}{2}$. According to ISO 5167 standard, for this case $L_1' = 1$ and $L_2' = 0.47$.

## 2 Simulation of the flow in the orifice plate

### 2.1 Case Overview

| **Parameter** | **Value** |
|---|---|
| Solver | `simpleFoam` (steady, incompressible) |
| Turbulence model | $k$-$\omega$ - SST |
| Flow regime | Turbulent |
| Geometry | orifice plate |
| Reynolds number | $2 \times 10^4$ (based on pipe diameter) |
| Expected runtime | 60-80 seconds |

Table 1: Orifice plate case specifications

### 2.2 Case copy

Since this case is not available in the tutorials folder, we have two options. We can either adapt a tutorial (for instance, the pitzDaily tutorial) or we can use a `template` case, that can be found in `$FOAM_ETC/templates`.

```
# load OpenFOAM if it is not included in the .bashrc file
source /usr/lib/openfoam/openfoam-2312/etc/bashrc

# Make a local folder
mkdir -p ~/Tutorials/Tutorial_3

# Copy the inflowOutflow in this folder
cp -r $FOAM_ETC/inflowOutflow ~/Tutorials/Tutorial_3/orificePlate
# Note that we have changed the name with the copy

# Go to the tutorial place
cd ~/Tutorials/Tutorial_3/orificePlate
```

This case is a simple example of a fluid domain with an inlet and an outlet. In the `README` file you can read the instructions to use the template. It is intended to be used with a general 3D geometry meshed with **`snappyHexMesh`**. Since we are going to use **`blockMesh`** with an axisymmetrical mesh, this file can be removed

```
# remove the README file
rm README
```

**Acknowledgments:** The author acknowledges the assistance of DeepSeek, an AI assistant created by DeepSeek Company, for providing valuable support in structuring this document, formatting LaTeX code, and developing the tutorial content.

**Disclaimer:** All content has been reviewed, modified, and validated by the author. The author takes full responsibility for the accuracy, completeness, and educational quality of this material.