

FOSSEE, IIT Bombay – Hackathon 2023 : OpenFOAM Hackathon

Project Report File

Contributor Name : Muralidharan P T
Academic Affiliation : B. Tech., Mechanical Engineering (IV-Year),
Puducherry Technological University.

Project Title: **76 - Numerical Simulation of Laminar and Turbulent flow through a circular pipe (in OpenFOAM v10)**

Procedure:

1. Verified the input files, such as the Velocity, Pressure and Turbulence parameters, Transport and physical Properties, Geometry and Mesh along with the solver control files. Verified the files are compatible with the version in use.

2. Proceeded with the input commands in order:

blockMesh

checkMesh (to check the Mesh quality)

icoFoam (solver for Laminar Case [01])

pimpleFoam (solver for Turbulent Case [02, 03, 04])

paraFoam (or) **touch filename.foam** (to create the foam file for Post-Processing)

3. Post-Processing Results:

Attached the results of,

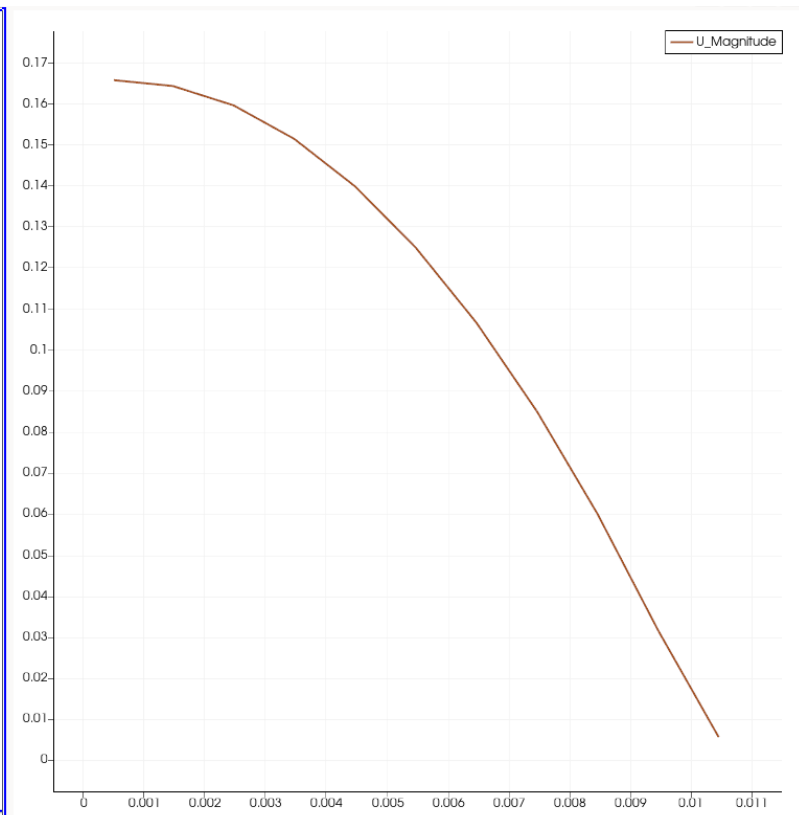
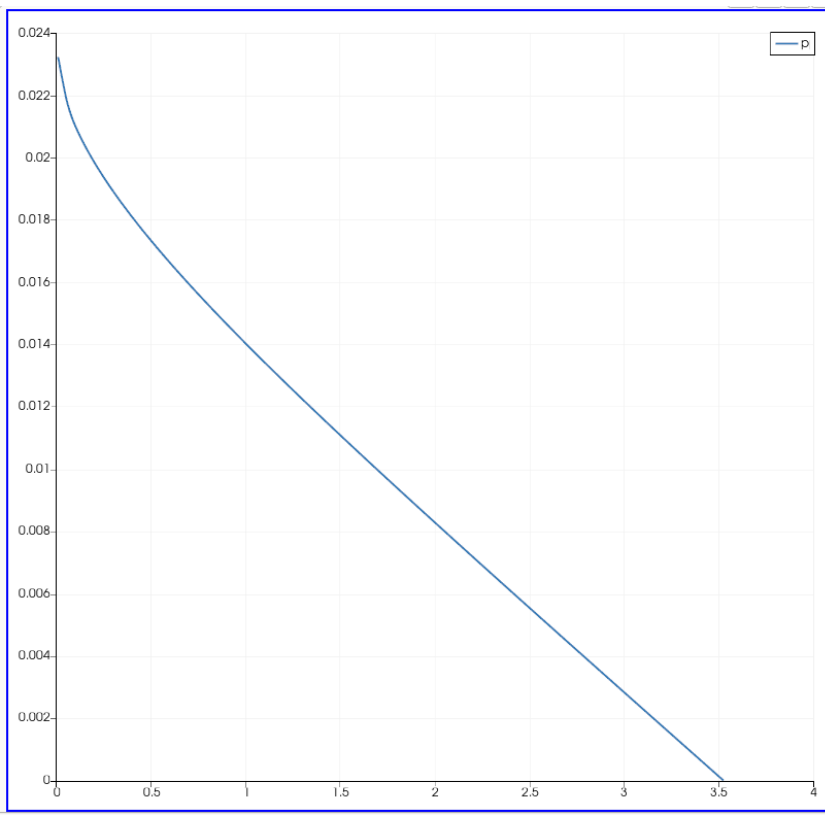
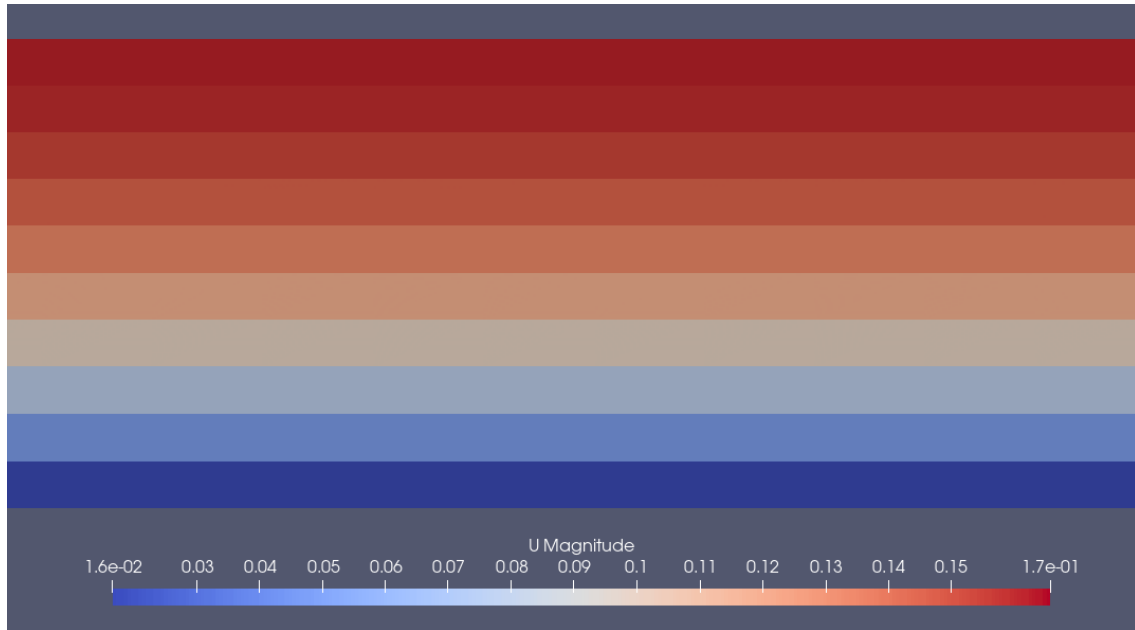
(i) Final Velocity magnitude, and

(ii) Plotted Graph of, Pressure vs Length &

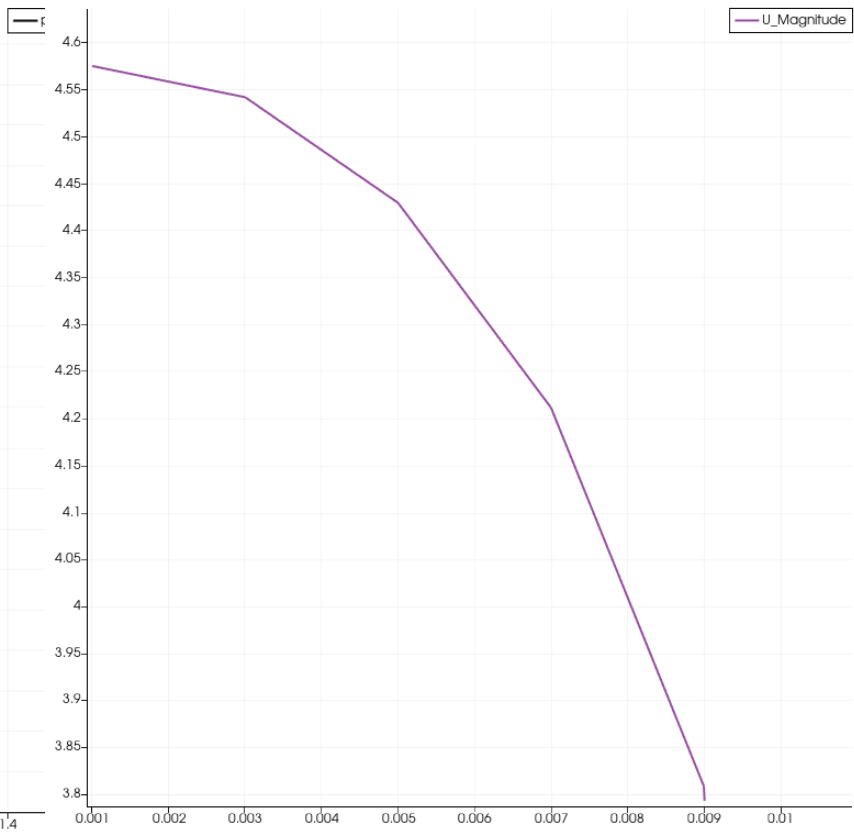
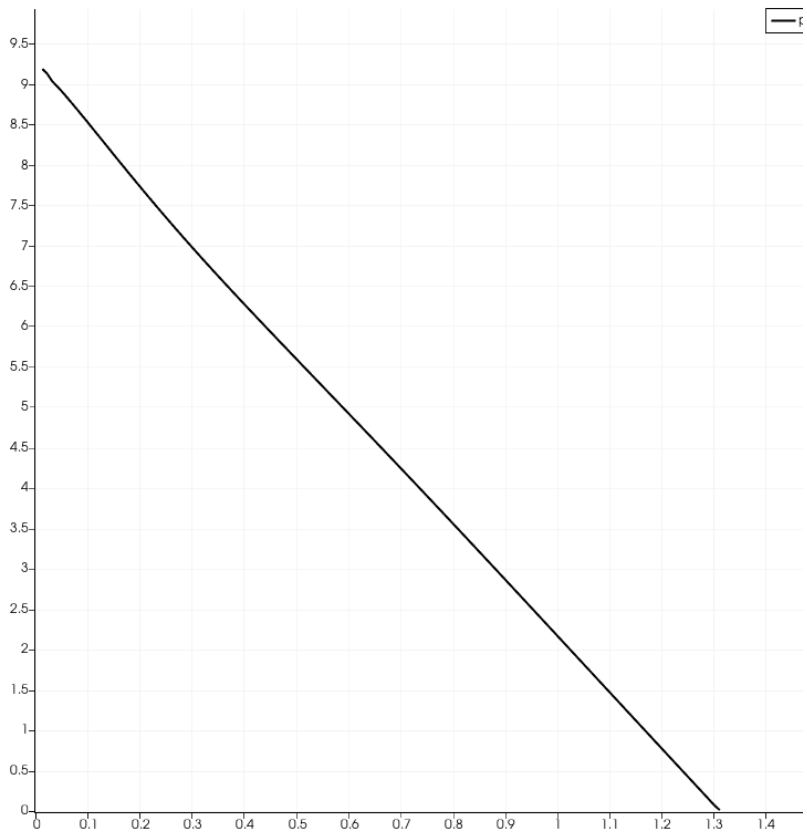
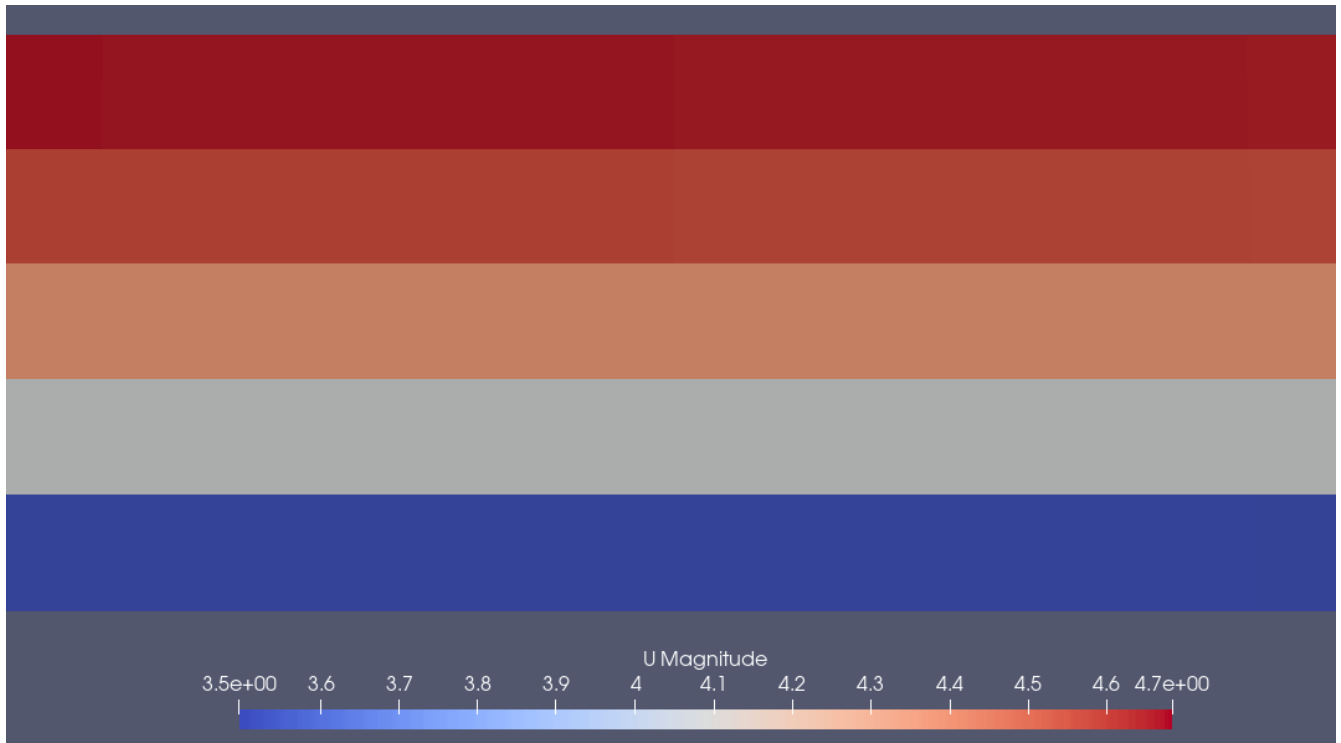
Velocity vs Radial Distance ,

for all 4 cases (1 Laminar + 3 Turbulent).

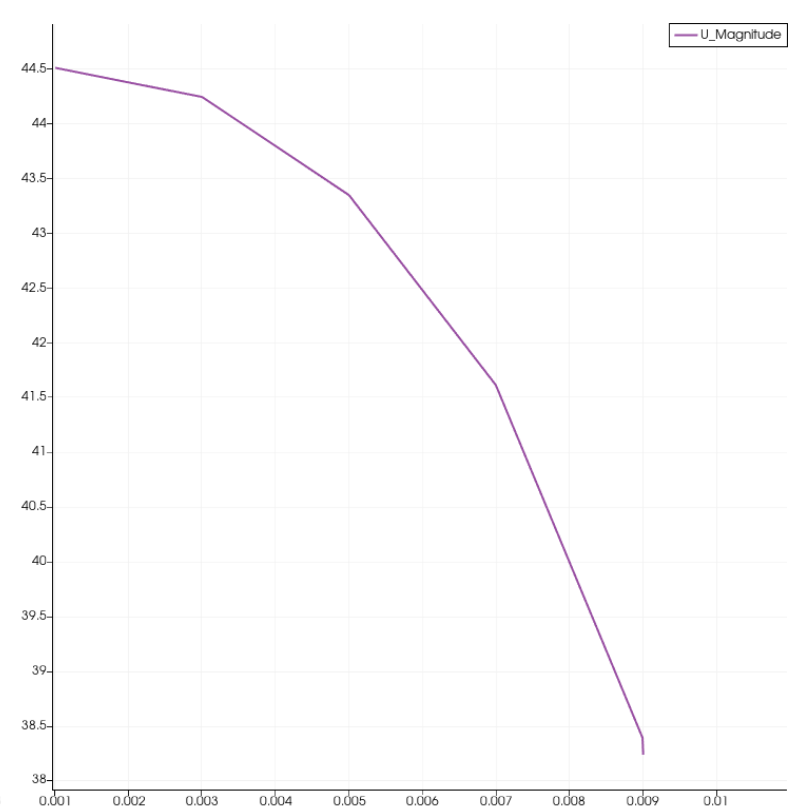
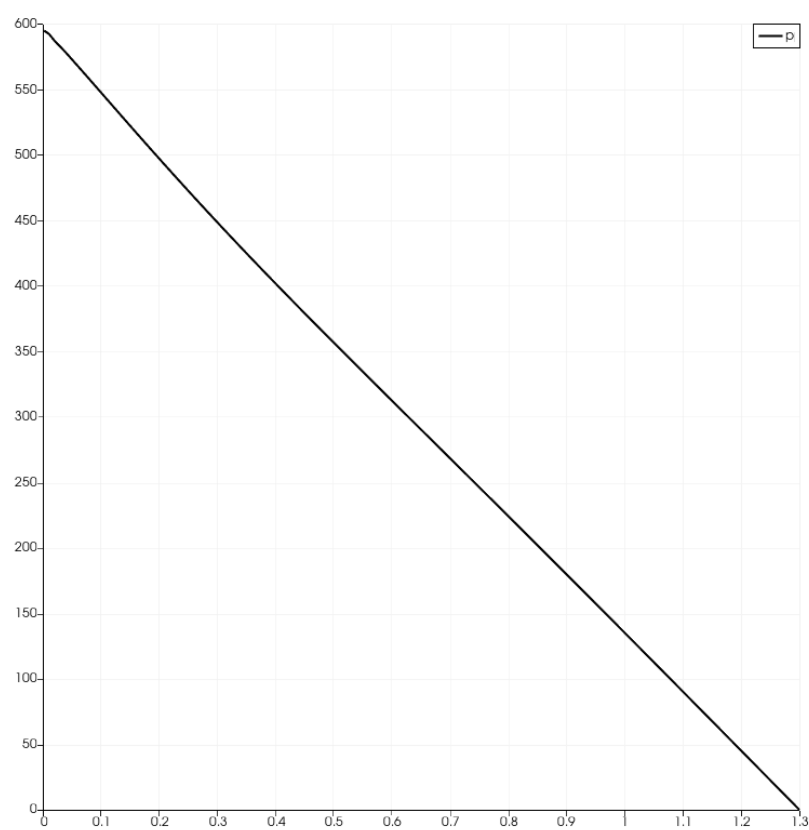
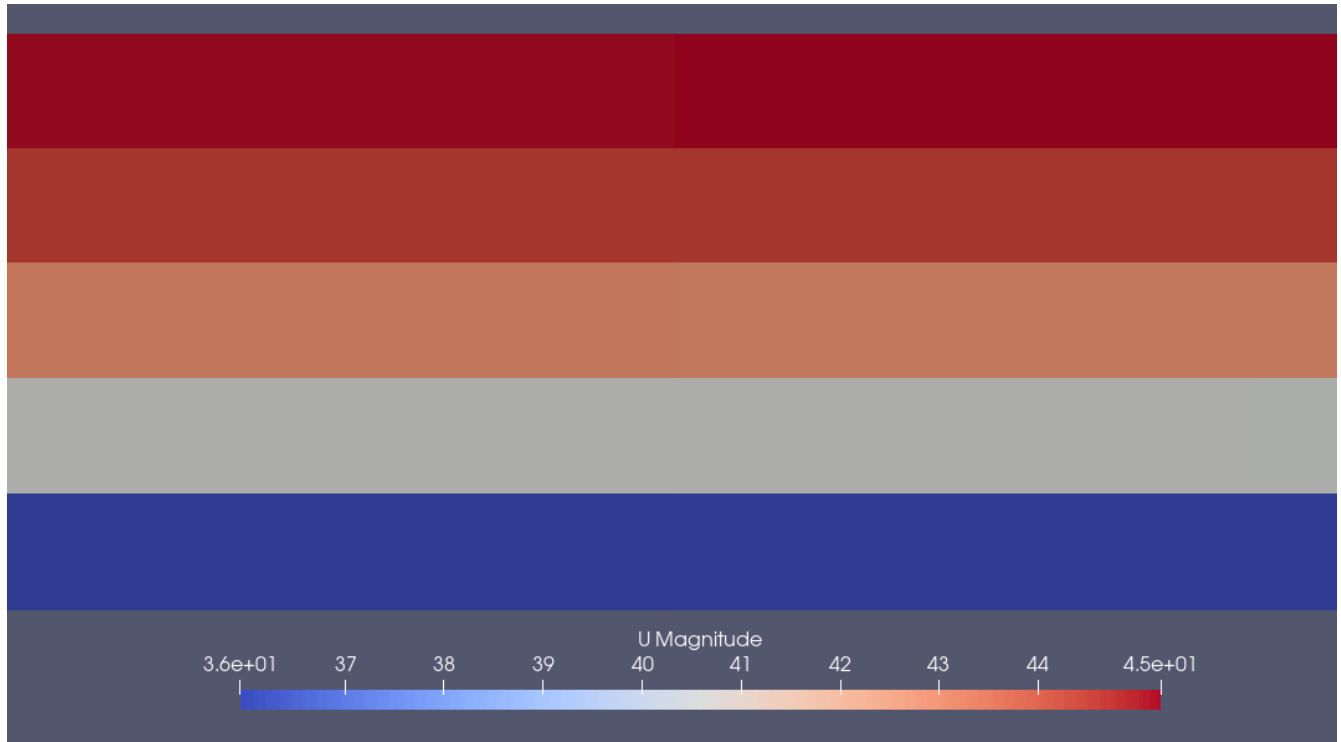
Case 1 (Laminar):



Case 2 (Turbulent) :



Case 3 (Turbulent) :



Case 4 (Turbulent) :

