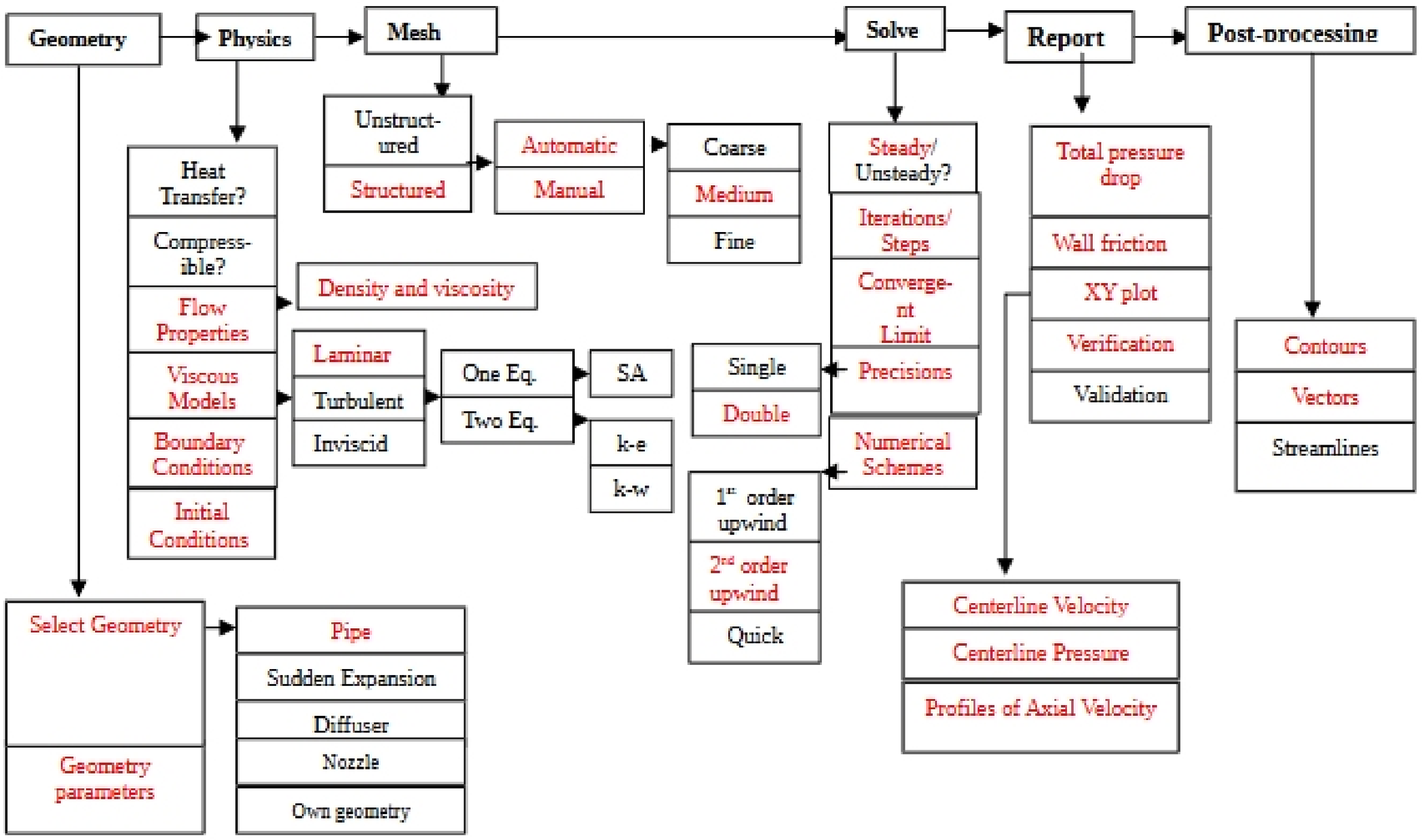
Simulation and Verification of Laminar Pipe Flows

57:020 Intermediate Mechanics of Fluids
CFD PRELAB 1

By Tao Xing and Fred Stern

1. Purpose

The Purpose of CFD PreLab 1 is to teach students how to use the CFD educational interface (FlowLab 1.2), be familiar with the options in each step of CFD Process, and relate simulation results to AFD concepts. Students will simulate **laminar** pipe flow following "CFD process" by an interactive step-by-step approach. Students will have "hands-on" experiences using FlowLab to compute axial velocity profile, centerline velocity, and friction factor on three different meshes (**Verification**). Students will compare simulation results with AFD data, analyze the differences and possible numerical errors, and present results in Lab report.

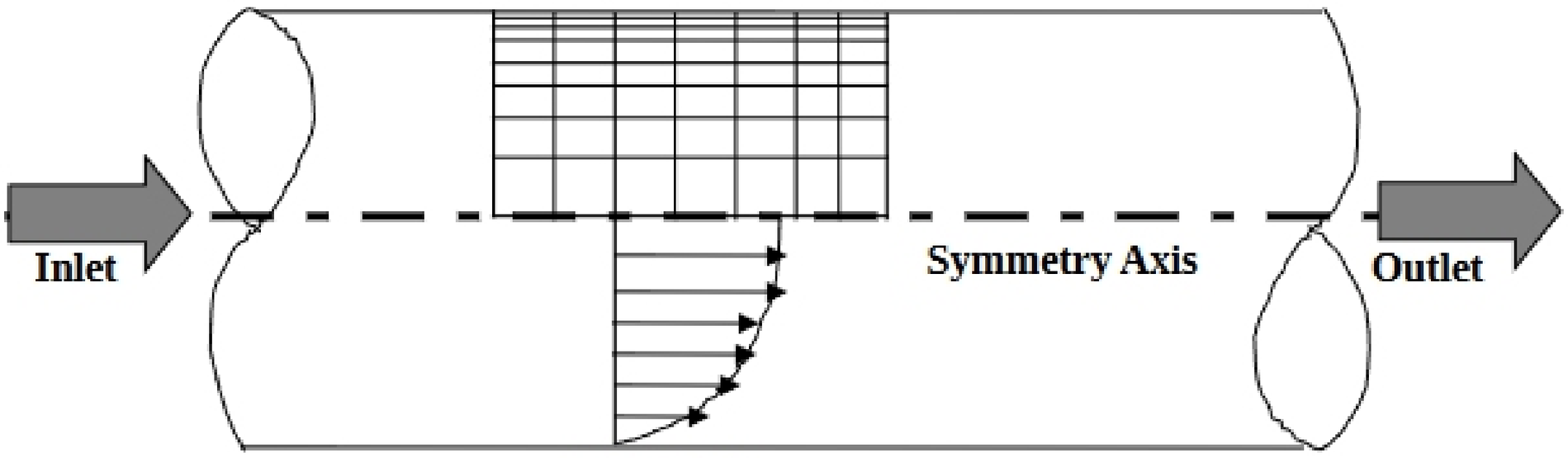


Flow Chart for ISTUE Teaching Module for Pipe flow (red color illustrates the options you will use in this CFD PreLab 1)

2. Simulation Design

In EFD Lab 2, you have conducted experimental study for **turbulent** pipe flow. The data you have measured will be used for CFD Lab 1. In CFD PreLab 1, simulation will be conducted only for **laminar** pipe flows, i.e. the inlet velocity (or Reynolds number) will be less than that for turbulent pipe flows. Comparisons between CFD and AFD can be conducted.

The problem to be solved is that of laminar/turbulent flows through a circular pipe. Reynolds number is 655 for laminar pipe flow, based on pipe diameter.

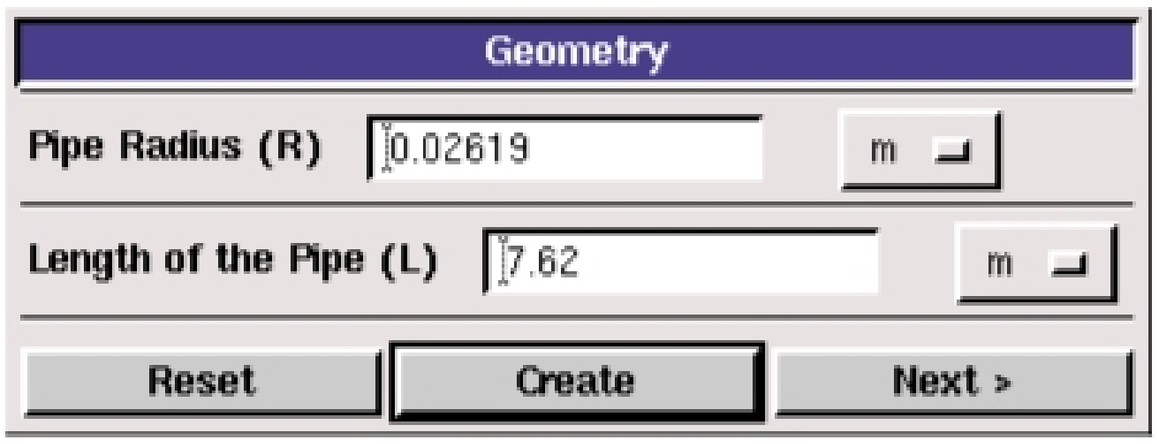


Since the flow is axisymmetric we only need to solve the flow in a single plane from the centerline to the pipe wall. Boundary conditions need to be specified for inlet, outlet, wall, and axis, as described later. Uniform flow was specified at inlet, the flow will reach the fully developed regions after a certain distance downstream. No-slip boundary condition will be used for the wall and constant pressure for outlet. Symmetric boundary condition will be applied on the axis. Since the flow is laminar, turbulence models are not necessary.

Analytical solutions (AFD) for Laminar Pipe Flow will be provided by TA of this Lab.

3. CFD Process

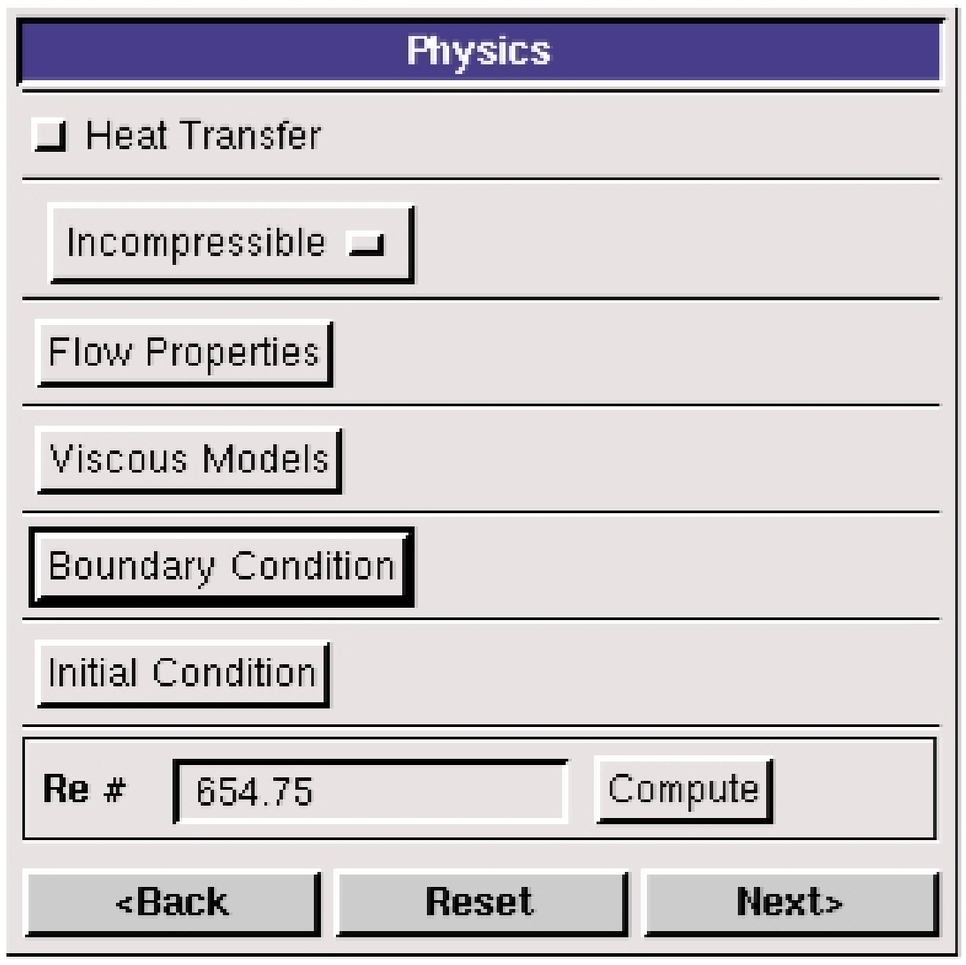
Step 1: (Geometry)



1. *Radius of pipe (0.02619 m)*
2. *Length of pipe (7.62 m)*

Click <<Create>>, after you see the pipe geometry created, click <<Next>>.

Step 2: (Physics)



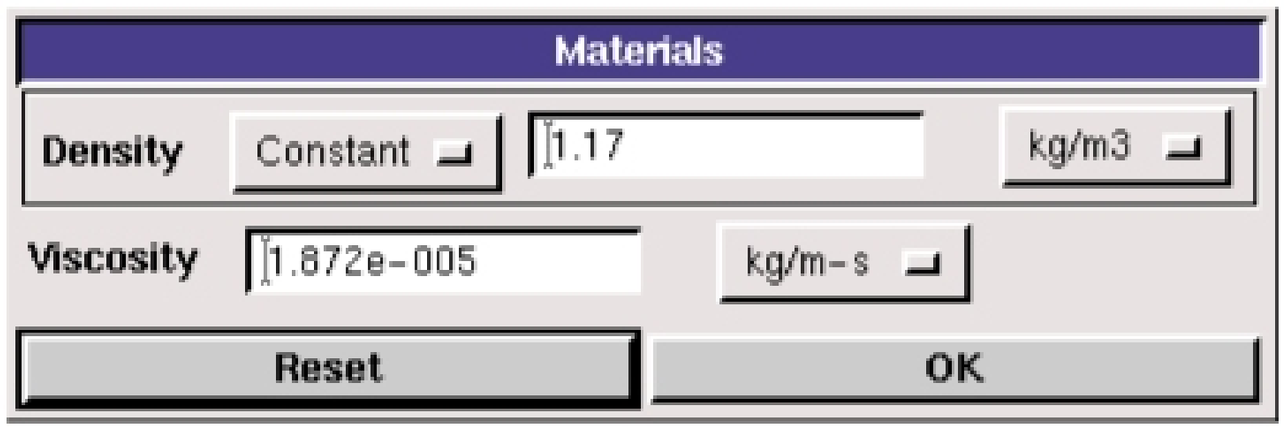
1. With or without Heat Transfer?

Since we are dealing only with the flow and not with the thermal aspects of the flow like heat transfer etc, switch the *<<heat transfer >>* button off, which is the default setup.

2. Incompressible or compressible

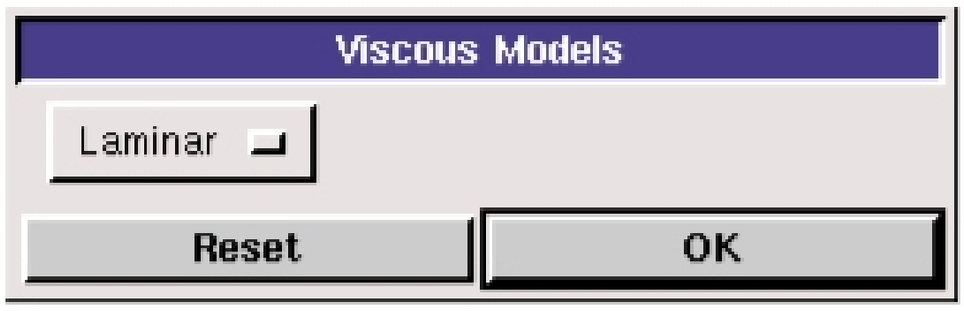
Choose “Incompressible”, which is the default setup.

3. Flow Properties



use the values shown in the above figure. Input the values and click **<<OK>>**

4. Viscous Model



In CFD PreLab 1, Choose laminar model and click **<<OK>>.**