

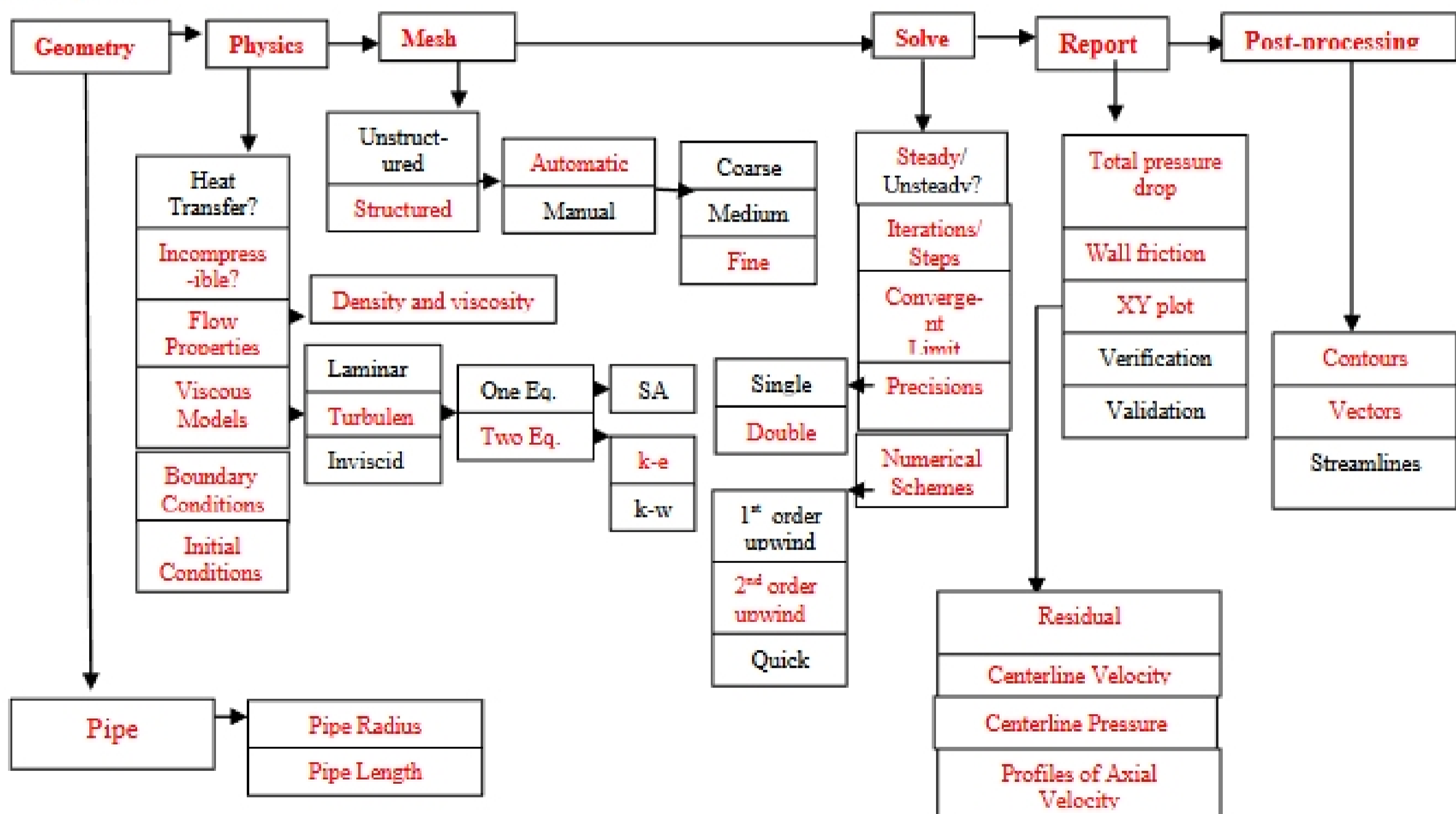
Simulation and Validation of Turbulent Pipe Flows

57:020 Mechanics of Fluids and Transport Processes
CFD LAB 1

By Tao Xing and Fred Stern
IIHR-Hydroscience & Engineering
The University of Iowa
C. Maxwell Stanley Hydraulics Laboratory
Iowa City, IA 52242-1585

1. Purpose

The Purpose of CFD Lab 1 is to teach students how to use the CFD Educational Interface (FlowLab 1.2), practice more options in each step of CFD Process, and relate simulation results to EFD and AFD concepts. Students will simulate **turbulent** pipe flow following the “CFD process” by an interactive step-by-step approach. The flow conditions will be the same as they used in EFD Lab2. Students will have “hands-on” experiences using FlowLab to compute axial velocity profile, centerline velocity, centerline pressure and friction factor. Students will compare simulation results with their own EFD data, analyze the differences and possible numerical/experimental errors, and present results in CFD Lab report.

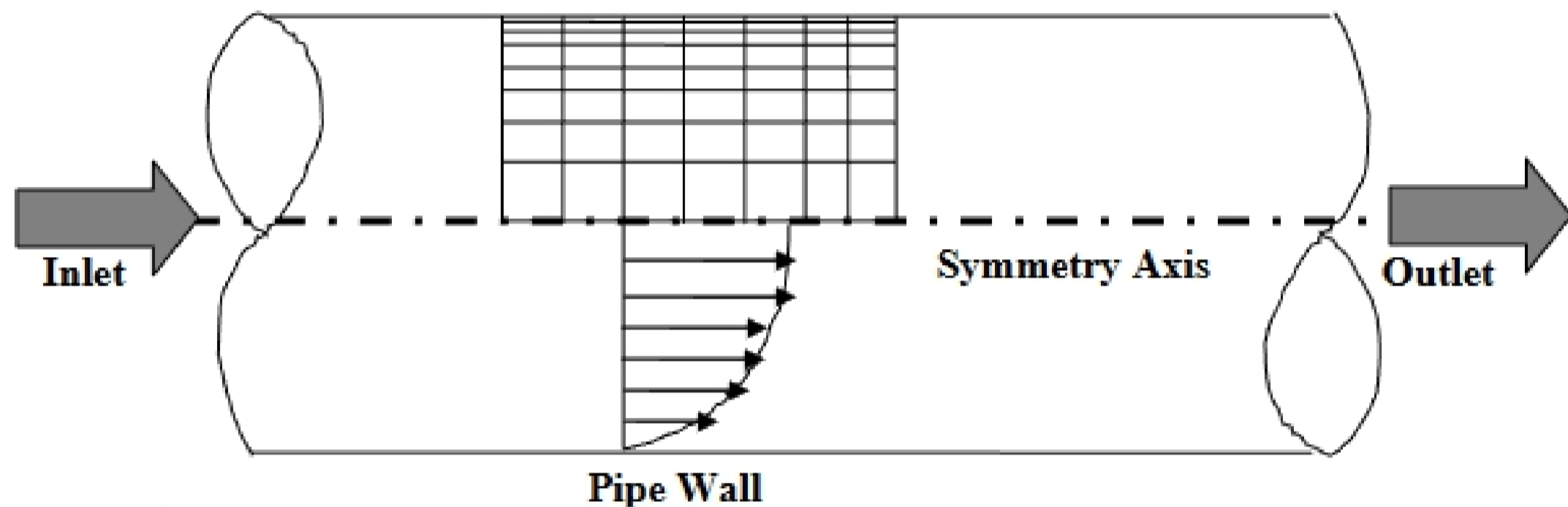


Flow chart for ISTUE teaching module for pipe flow (red color illustrates the options you will use in CFD Lab 1)

2. Simulation Design

In EFD Lab 2, you have conducted experimental study for **turbulent** pipe flow. The data you have measured include centerline pressure distribution and fully developed axial velocity profile. These data will be used in this Lab for comparisons with CFD predictions.

The problem to be solved is that of **turbulent** flows through a circular pipe. Reynolds number based on pipe diameter and inlet velocity should be **computed from your own EFD data** and is much higher than the Reynolds number used in CFD Prelab1.



The problem formulation is similar to that in CFD PreLab1 and will not be repeated here. The Reynolds number is much higher and students need specify turbulence model, and thus more variables need to be specified in boundary conditions, as will be discussed in details later.

3. CFD Process

Step 1: (Geometry)

Geometry	
Pipe Radius (R)	<input type="text" value="0.02619"/> m
Length of the Pipe (L)	<input type="text" value="6.096"/> m
<input type="button" value="Reset"/>	<input type="button" value="Create"/> <input type="button" value="Next >"/>

1. **Radius of pipe** (pipe radius in your EFD Lab2, use 0.02619 m if not available)
2. **Length of pipe** (6.096m)

NOTE: The actual length of the pipe is 30 feet (9.144m), However, in CFD simulation, we need specify “outlet pressure”, and we don’t have pressure transducer at the pipe outlet. So we choose the outlet of the pipe we will simulate to be the location of the last pressure transducer, that is 6.096 meters from the pipe inlet. Click <<Create>>, after you see the pipe geometry created, click <<Next>>.

Step 2: (Physics)

Physics

Heat Transfer

Incompressible ▾

Flow Properties ▾

Viscous Models ▾

Boundary Condition ▾

Initial Condition ▾

Re # 122769 Compute

<Back Reset Next>

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

Materials

Density Constant ▾ 1.1885 kg/m3 ▾

Viscosity 1.8396e-005 kg/m-s ▾

Reset OK

use the flow property values in your EFD Lab2. If you only know the room temperature when you conducted EFD Lab2, you can use the following website to easily get the corresponding density and dynamic viscosity for air:

<http://www.mhtl.uwaterloo.ca/old/onlinetools/airprop/airprop.html>

If the room temperature is 24°, the density and viscosity are the values shown in the above panel.

NOTE: viscosity used in FlowLab is the **dynamic viscosity** ($kg/m \cdot s$), **NOT** kinematic viscosity (m^2/s)