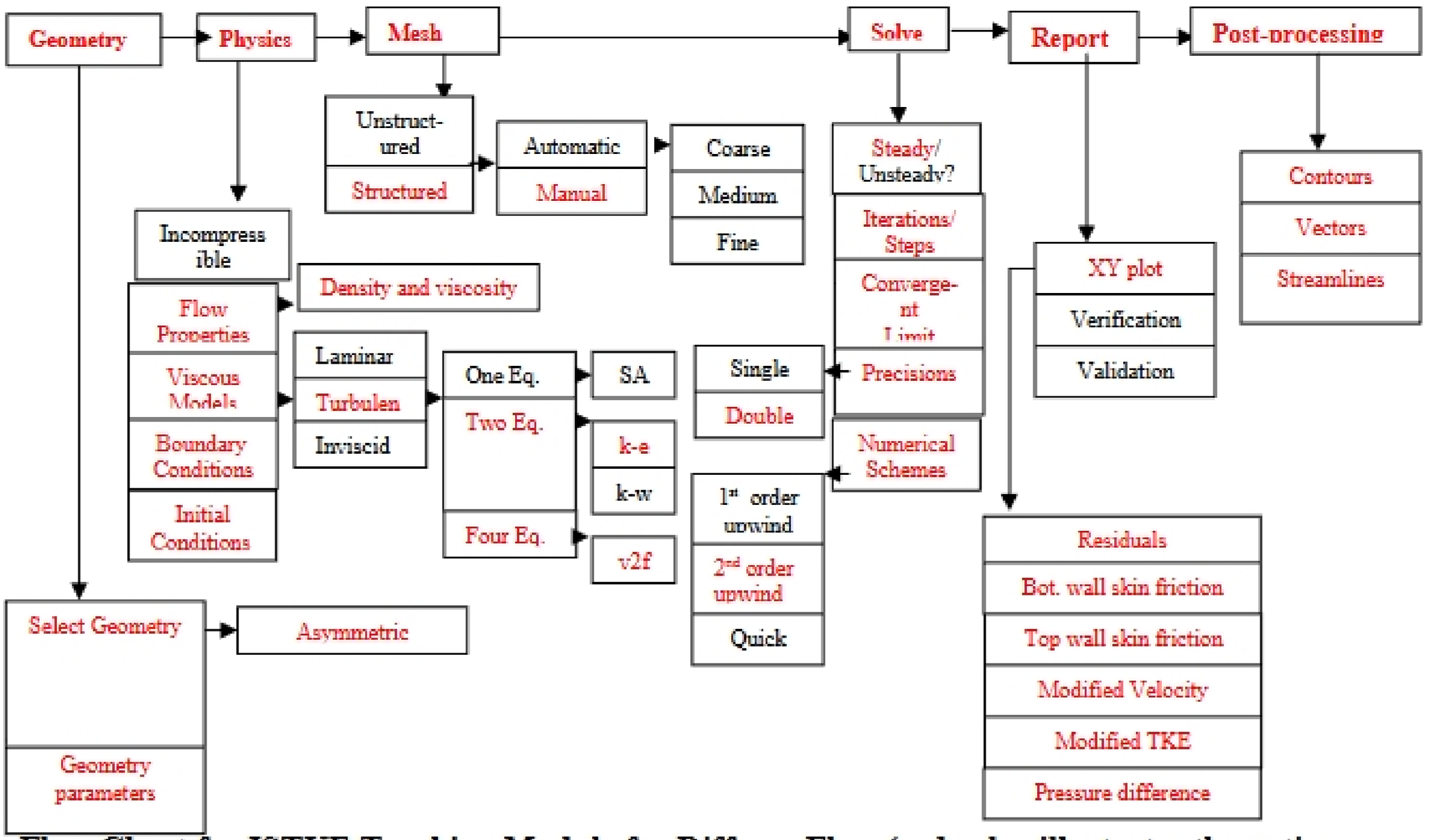
Simulation of Turbulent Flow in an Asymmetric Diffuser

58:160 Intermediate Mechanics of Fluids
CFD LAB 3

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

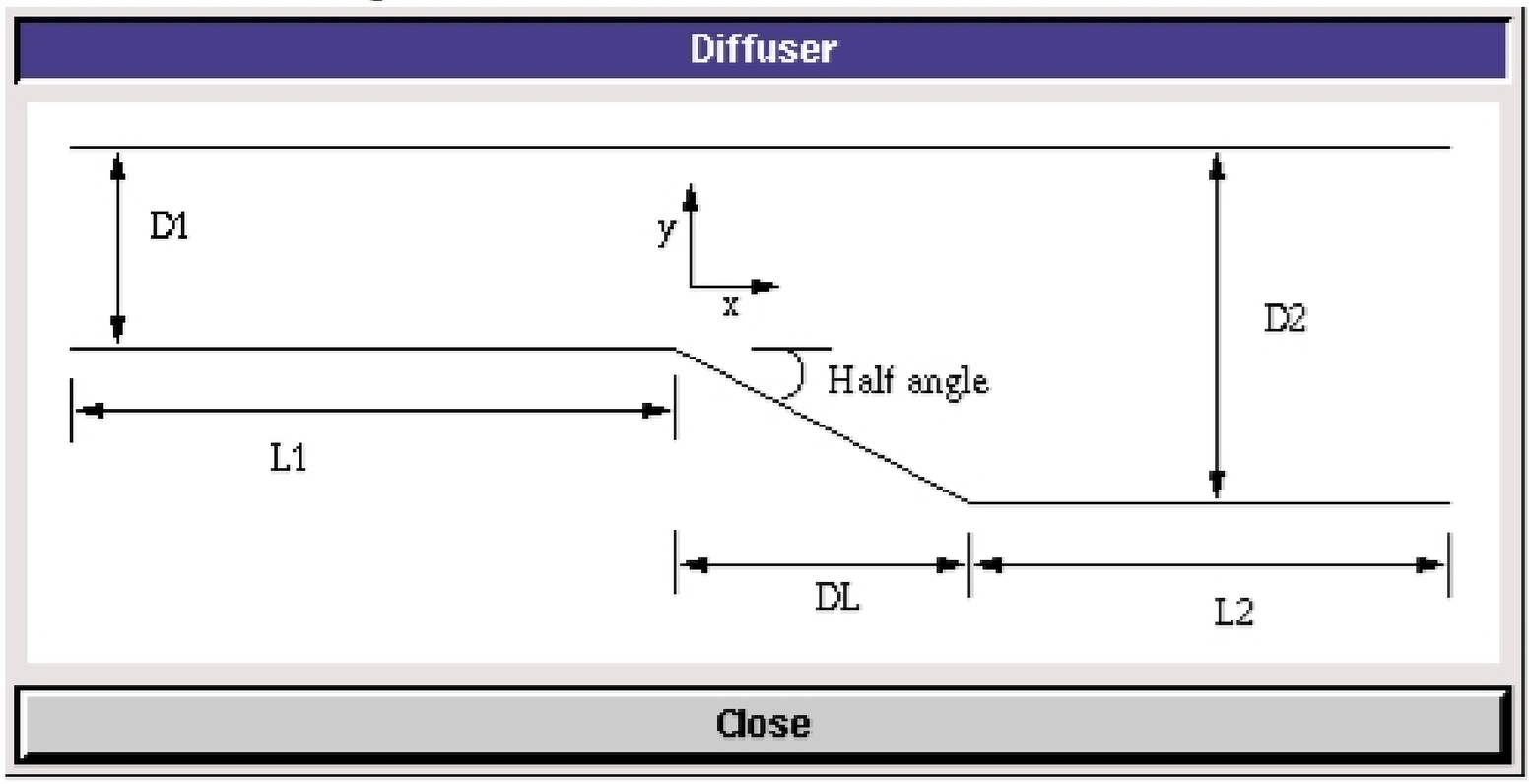
The Purpose of CFD Lab 3 is to simulate **turbulent** flows inside a diffuser following the "CFD process" by an interactive step-by-step approach and conduct verifications using CFD Educational Interface (FlowLab 1.2). Students will have "hands-on" experiences using FlowLab to conduct **validation of velocity, turbulent kinetic energy, and skin friction factor. Effect of turbulent models will be investigated, with/without separations**. Students will manually generate meshes, solve the problem and use post-processing tools (contours, velocity vectors, and streamlines) to visualize the flow field. Students will analyze the differences between CFD and EFD and present results in a CFD Lab report.



Flow Chart for ISTUE Teaching Module for Diffuser Flow (red color illustrates the options you will use in CFD Lab 3)

2. Simulation Design

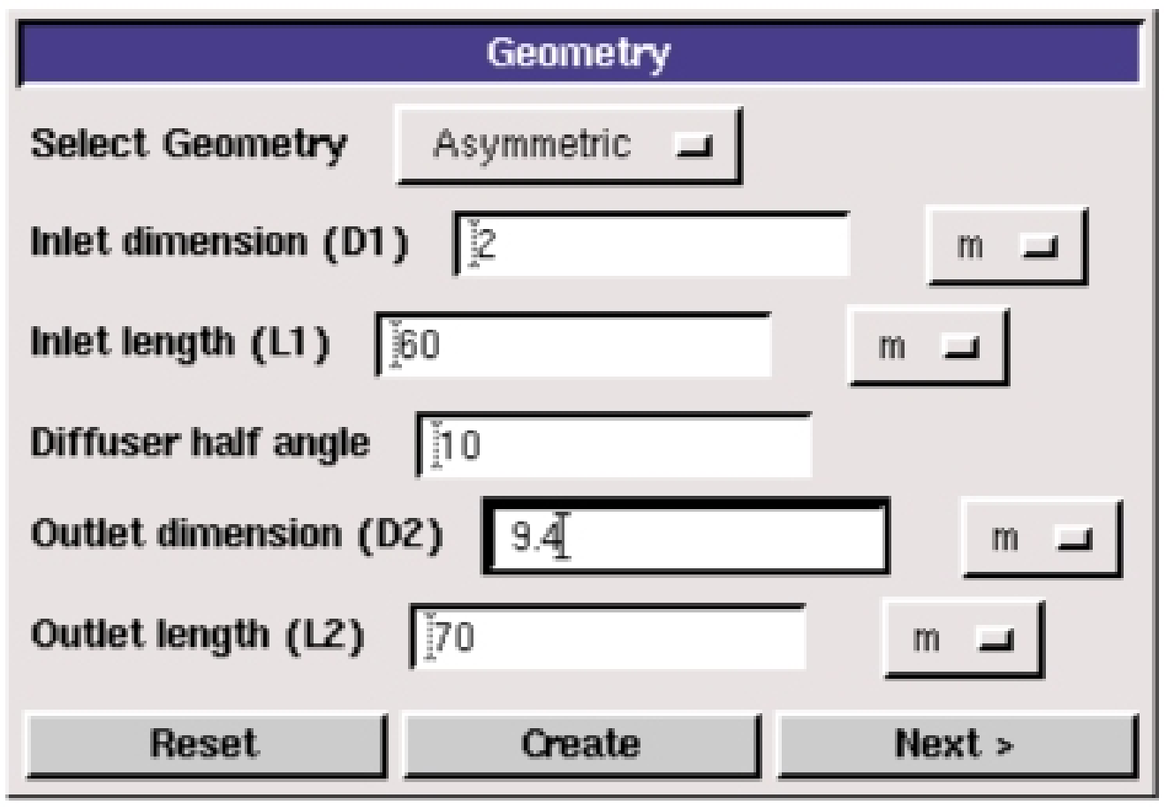
The problem to be solved is that of turbulent flows inside an asymmetric diffuser (2D). Reynolds number is 17,000 based on inlet velocity and inlet dimension (D1). The following figure shows the sketch window you will see in FlowLab with definitions for all geometry parameters. Before the diffuser, a straight channel was used for generating fully developed channel flow at the diffuser inlet. The origin of the coordinates is placed at the inlet of the channel before diffuser.



In CFD Lab3, all EFD data for turbulent airfoil flow in this Lab will be provided by the TA and saved on the Fluids lab computers.

3. CFD Process

Step 1: (Geometry)

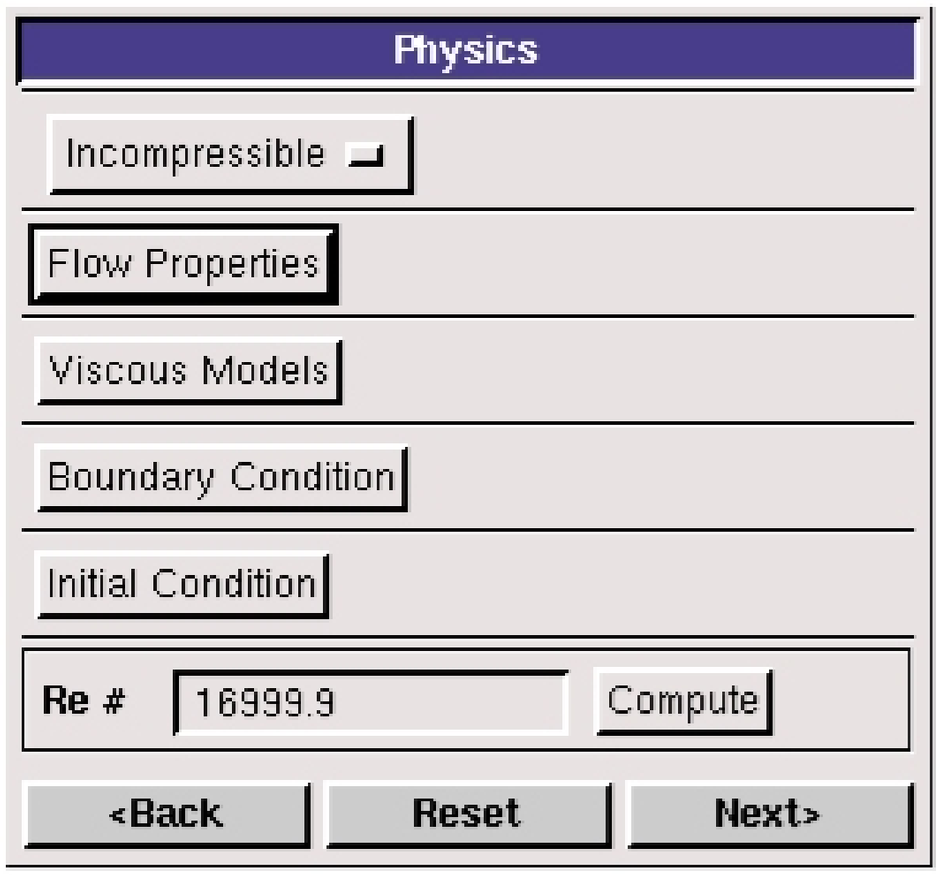


1. Select Geometry: Asymmetric
2. Inlet dimension (D1) (2 m)
3. Inlet length L1 (60 m)

4. Diffuser half angle (10 or 4, read exercises at the end)
5. Outlet dimension (D2) (9.4 m)
6. Outlet length (L2) (70 meters).

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

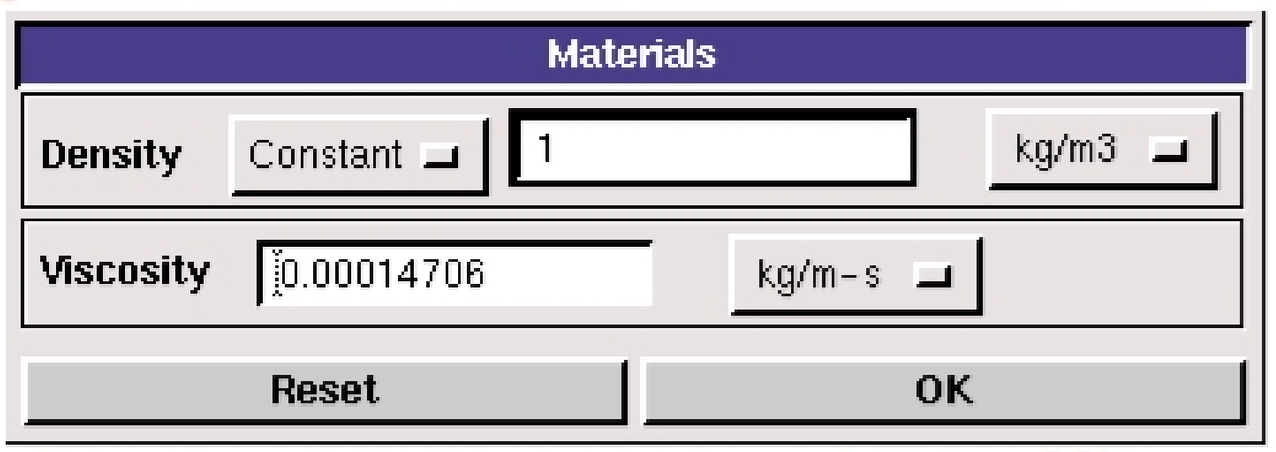
Step 2: (Physics)



(1). Incompressible

"Incompressible", which is the default setup.

(2). Flow Properties



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

(3). Viscous Model