

MEE 340



Fluid Mechanics

Virtual Fluids Lab Demonstration
Using FlowLAB by Fluent

Prepared by Dan Wu

Under the guidance and supervision of Prof. M. Kostic



Introduction of Flowlab software

- ◆ The FLOWLAB can help you understand the physical phenomena and reinforce concepts in fluid mechanics.
- ◆ FLOWLAB receives user input by means of its Graphical User Interface (GUI). A part of the FLOWLAB GUI is problem dependent. This is managed by a problem specific template file.
- ◆ FLOWLAB is equipped with the ready-made templates of classic problem taught in many engineering curriculum



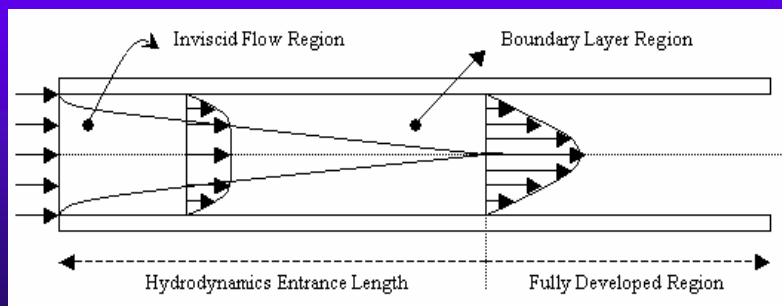
Introduction of FLOWLAB software Cont'd

- ◆ What comes along with FLOWLAB?
 1. A friendly graphical user interface to setup of parameters, the solution control, the convergence monitoring, etc.
 2. Graphical representation of results (vector plots, contour plots, path lines etc.)
 3. X-Y plots comparison of the numerical results with the theoretical or experimental results



Starting FLOWLAB

- ◆ Case 1:
Incompressible Viscous Flow Through Pipes





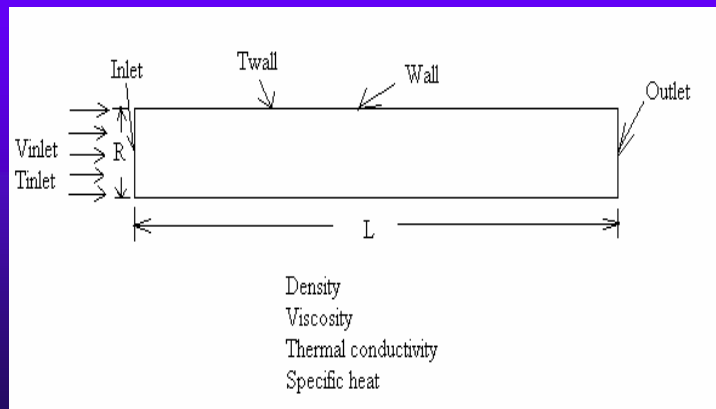
Overview of Case 1

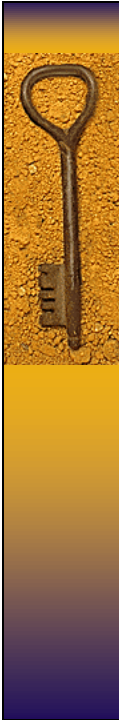
- ◆ In this lab you can model viscous flow in circular pipe with or without heat transfer
- ◆ The objective of this lab is to introduce the incompressible viscous flow in circular pipe
- ◆ We will observe the entrance length, the distribution of friction factor coefficient and Nusselt number, the change of outlet temperature and velocity, etc.



Problem description for Case 1

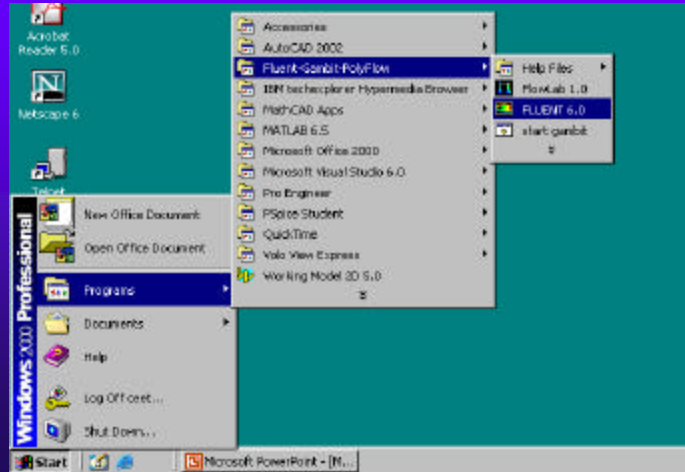
- ◆ Because of symmetry of the circular pipe geometry, only a portion of the domain needs to be modeled. The computational domain is shown in the following figure





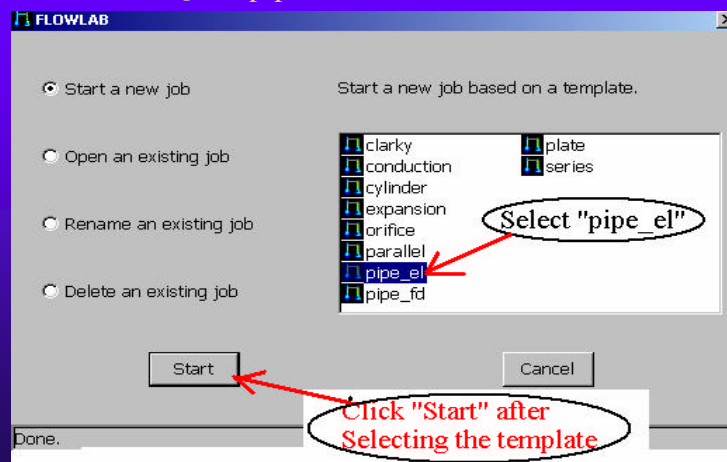
Step 1a: Open a new job

- ◆ Click Start>Programs>Fluent-Gambit-PolyFlow>FlowLab 1.0 (this may be different at different computers)



Step 1b: Open a new job

1. Select "Start a new job"
2. Select a template (`pipe_el`), then click the start button





Step 2: Geometry

- ◆ The input required to be given by users are pipe radius (Min=0.05m, Max=0.5m) and pipe length (Min=0.4m, Max=50m). In this sample use defaults please.

Click here after the setup of parameters, then click "Next"

Change the parameters here



Step 3: Mesh

- ◆ There are three choices available for us (for different mesh densities.)
 1. Coarse grid
 2. Medium grid
 3. Fine grid
- ◆ In this sample we choose "Medium" mesh density

Click "Creat" after the setup of parameter, then click "Next"

Click here with right mouse button to change mesh density

Step 3 Cont'd

- ◆ In this step we have generated the grid for the purpose of discretization, to translate the physical space into computational space
- ◆ The coarser the grid the faster the solutions converge. But for more accurate solutions use fine grid

Step 4: Physics

Click here to open "Boundary Condition Form"

Click here to open "Materials Form"

Physics Box

Global Control

Transcript

Description

flowlab1.tcas
Title : Pipeflow (Entrance Length)



Step 4 Cont'd

- ◆ This step is the modeling step where you can get to specify

1. *The **boundary conditions***

2. *The **material properties***



Step 4 Cont'd

- ◆ **The boundary conditions**

Under the “Boundary conditions” button you can specify the inlet condition of pipe flow which include inlet temperature and inlet velocity. In this lab you should specify thermal condition by select “temperature” or “flux” button to set the wall temperature of the pipe or the wall heat flux of the pipe.

- ◆ **The material conditions**

Under the “materials” button you can specify the properties of pipe flow which include the density, the viscosity, the specific heat and thermal conductivity.

Step 4 Cont'd

◆ The range of parameters

Inlet velocity: 0.01m/s - 10m/s

Inlet temperature: 100K - 100,00K

Wall heat flux: 1W/m² - 1000W/m²

Wall temperature: 100K - 100,00K

Fluid density: 1Kg/m³ - 2,000Kg/m³

Viscosity: 10⁻⁵ Kg/m-s – 1 Kg/m-s

Fluid specific heat: 10J/Kg-C – 100,00J/Kg-C

Thermal conductivity: 0.01W/m-C – 100,00W/m-C

Step 4 Cont'd

The screenshot displays the FLOWLAB software interface. The 'Materials Form' is visible with the following parameters:

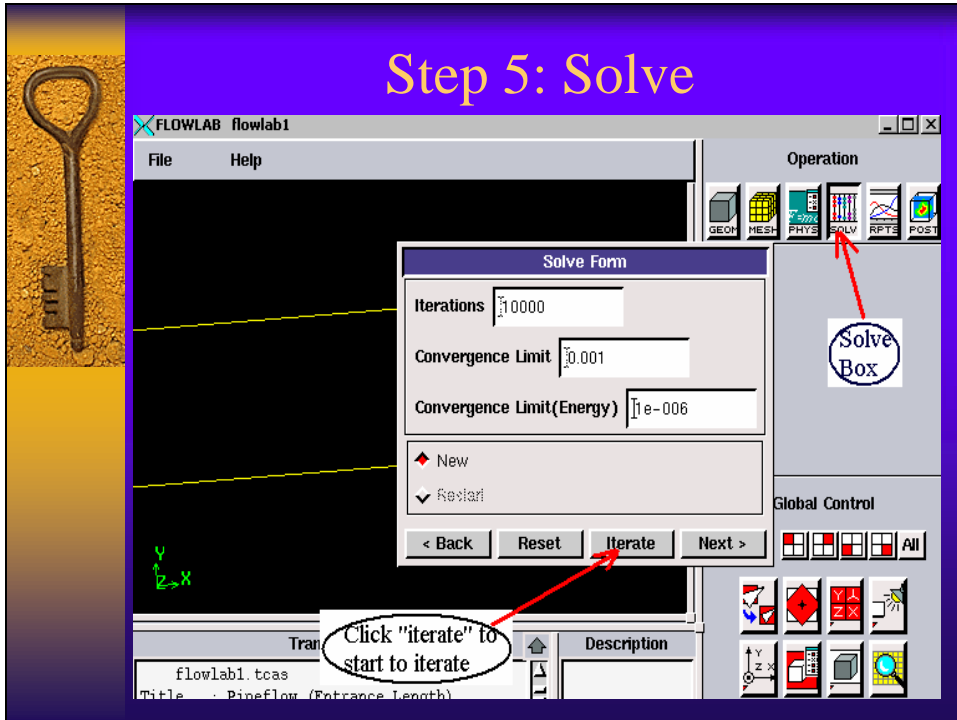
- Density: 1000 kg/m³
- Viscosity: 0.000799 kg/m-s
- Specific Heat: 2102 J/kg-C
- Thermal Conductivity: 0.5 W/m-C

The 'Boundary Condition Form' is also visible with the following parameters:

- Inlet Velocity: 0.05 m/s
- Inlet Temperature: 293.097 K
- Thermal Condition: Heat Flux (selected)
- Heat Flux: 30 W/m²
- Exposure: 0.67 K

Red arrows point to the 'Materials Form' and 'Boundary Condition Form' labels, and another red arrow points to the 'OK' button in the Transcript area. A text box at the bottom of the screenshot reads: "Click 'OK' after the setup of parameters".

Step 5: Solve



FLOWLAB flowlab1

File Help

Operation

GEOM MESH PHYS SOLV RPTG POST

Solve Form

Iterations 10000

Convergence Limit 0.001

Convergence Limit(Energy) 1e-006

◆ New

▼ Restart

< Back Reset Iterate Next >

Global Control

All

Click "iterate" to start to iterate

Solve Box

flowlab1.tcas

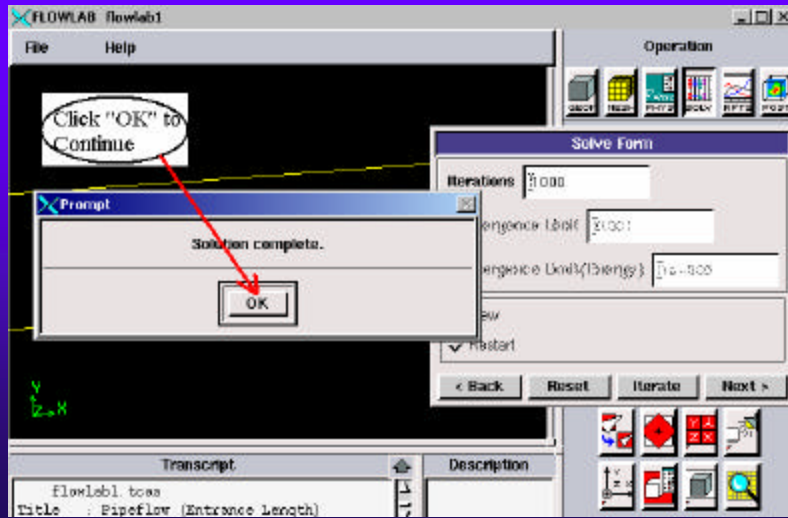
Title : Pipeflow (Entrance Length)

Step 5 Cont'd

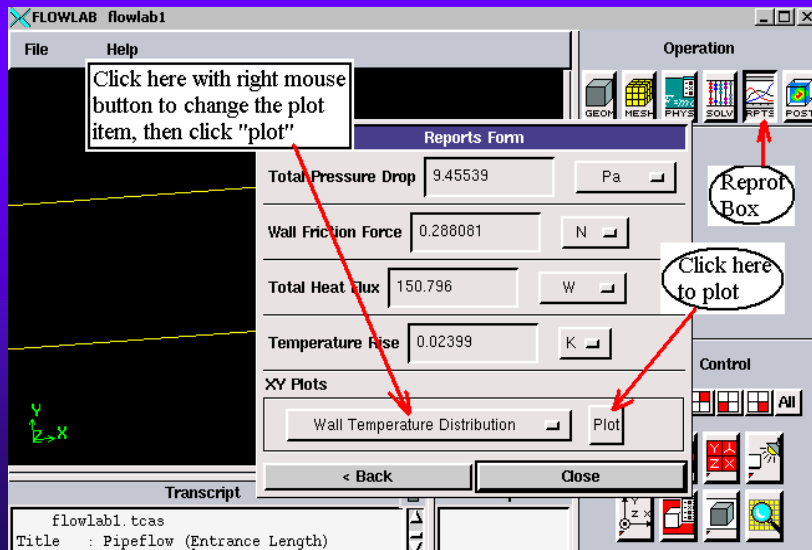
- ◆ In this step, the *Navier Stokes equations* are solved throughout the computational domain
- ◆ The desired *number of iterations* (1000) and the *convergence limit* (0.00001) for the solver can be specified by the user in the *solve form*
- ◆ The lower the convergence limit the smaller the error
- ◆ Change the convergence limit bigger (0.001) if the solve process is too long
- ◆ The solver keeps iterating till the convergence limit is reached

Step 5 Cont'd

- After clicking the iteration button, a message will come out as soon as the solution is obtained.



Step 6: Report





Step 6: Report

◆ The numerical report:

1. Total pressure drop
2. Wall friction factor
3. Total heat flux
4. Temperature rise



Step 6: Report

◆ The plot report

1. Residuals
2. Centerline velocity distribution
3. Centerline pressure distribution
4. Centerline temperature distribution
5. Wall friction factor distribution
6. Wall temperature distribution
7. Wall Nusselt number no distribution
8. Outlet velocity distribution
9. Outlet temperature distribution

Step 6 Cont'd

- ◆ Select the plot

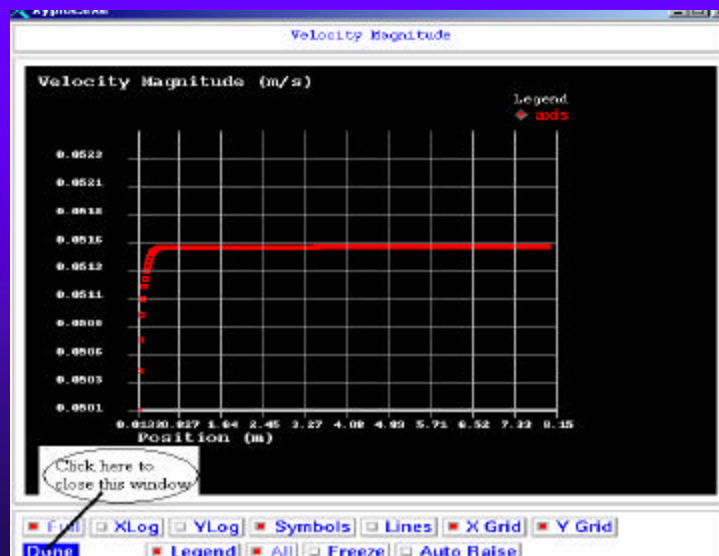
The screenshot shows the FLOWLAB software interface. The main window is titled 'flowlab1'. A 'Reports Form' dialog box is open, displaying various simulation results. A red arrow points to the 'Centerline Velocity Distribution' option in the list. A callout box with the text 'Select plot item here' is positioned over the dialog box. The 'Transcript' window at the bottom shows the file name 'flowlab1.tcas' and the title 'Pipeflow (Entrance Le...'. The 'Reports Form' contains the following data:

Item	Value	Unit
Total Pressure Drop	9.45539	Pa
Wall Friction Force	0.288081	N
Total Heat Flux	150.796	W

The 'Centerline Velocity Distribution' option is selected in the list. Other options include Residual, Centerline Pressure Distribution, Centerline Temperature Distribution, Wall Friction Factor Distribution, Wall Temperature Distribution, Wall Nusselt No Distribution, Outlet Velocity Distribution, and Outlet Temperature Distribution.

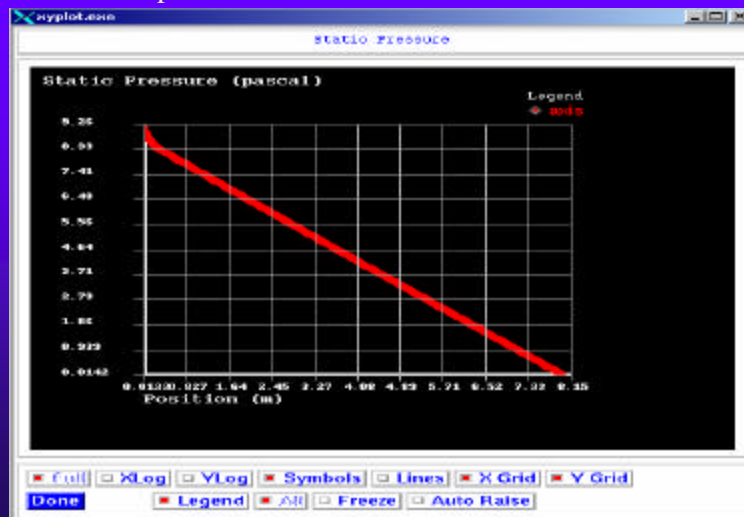
Step 6: Report

- ◆ Centerline velocity distribution



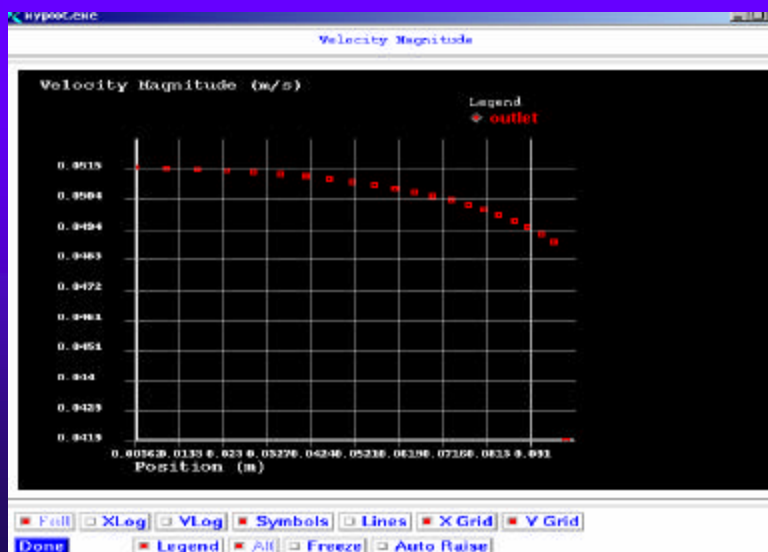
Step 6: Report

- ◆ Centerline pressure distribution



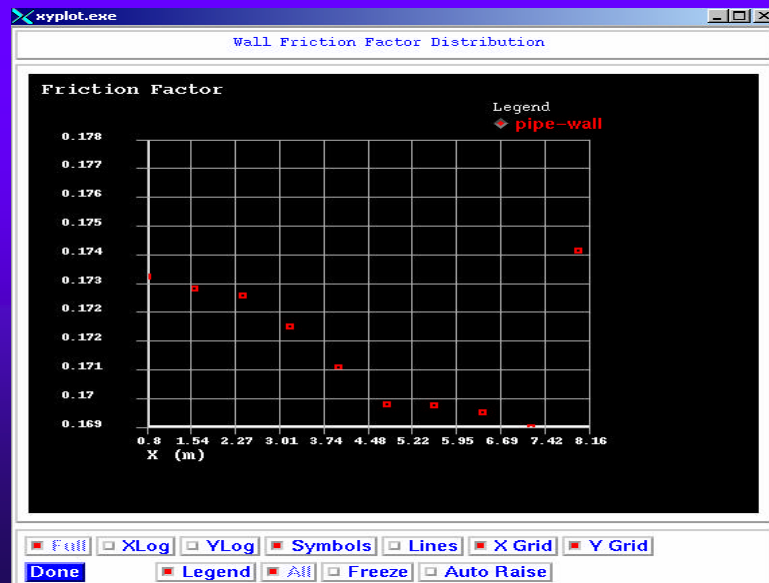
Step 6: Report

- ◆ Outlet velocity distribution



Step 6: Report

- ◆ Wall friction factor distribution



Step 7: Post-processing

- ◆ Two options are offered in FLOWLAB
 1. Contour
 2. Vector
- ◆ Click and highlight the Contour or Vector, then click “activate” button to view
- ◆ An example is shown later

Step 7: Post-processing Cont'd

The screenshot shows the FLOWLAB software interface. A yellow contour plot is visible in the main window. A callout box labeled "Post-processing Box" points to the right-hand side of the interface. Another callout box labeled "Highlight 'contour' then click 'activate'" points to the "contour" entry in the "Postprocessing Objects" list. A third callout box labeled "Click 'Activate' to view" points to the "Activate" button in the "Postprocessing Objects" panel. The "Postprocessing Objects" panel also shows "Vector" as an active object. The "Operation" and "Global Control" panels are visible at the bottom right.

Step 7: Post-processing Cont'd

◆ Pressure simulation in pipe flow

The screenshot shows the FLOWLAB software interface with a "Create Simulation Object" dialog box open. The dialog box has a "Label" field, a "Definition" field with "Pipe" selected, and "Attributes" section. The "Attributes" section has "Contour: pressure" checked, with a color scale from 0 to 9.86176. The "Vector" and "Particle" options are unchecked. The "Apply" button is highlighted. A callout box labeled "Highlight here then click 'Apply'" points to the "Apply" button. Another callout box labeled "Click 'Apply' to create simulation" points to the "Apply" button. A third callout box labeled "Click here to create simulation object" points to the "POST" button in the "Operation" panel. The "Postprocessing Objects" panel shows "contour" and "simulation_object:1" as active objects. The "Operation" and "Global Control" panels are visible at the bottom right.



Assignment for the lab

Presentation

- ◆ The plot of residuals' history on pipe_el template
- ◆ Contour of pressure (shown before in this sample) and velocity vector (to be done by you) plots of the developing region of the flow on pipe_el template

Discuss

- ◆ How does the velocity profile for laminar flow (to be done by you on pipe_fd template) differ with that of turbulent flow?



Thanks!!!

- To my advisor: Prof. M. Kostic for his support
- To MEE 340 TA: Srinivasa for his useful discussion