MEE 340

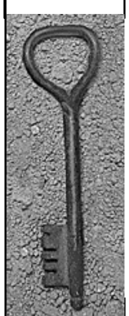
Fluid Mechanics

Virtual Fluids Lab Demonstration

Using FlowLAB by Fluent

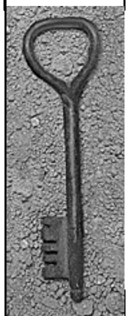
Prepared by Dan Wu

Under the guidance and supervision of Prof. M. Kostic



Introduction of Flowlab software

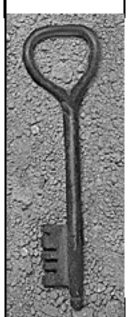
- The FLOWLAB can help you understand the physical phenomena and reinforce concepts in fluid mechanics.
- FLOWLAB receives user input by means of its Graphical User Interface (GUI). A part of the FLOWLAB GUI is problem dependent. This is managed by a problem specific template file.
- FLOWLAB is equipped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



Introduction of FLOWLAB software Cont'd

- What comes along with FLOWLAB?

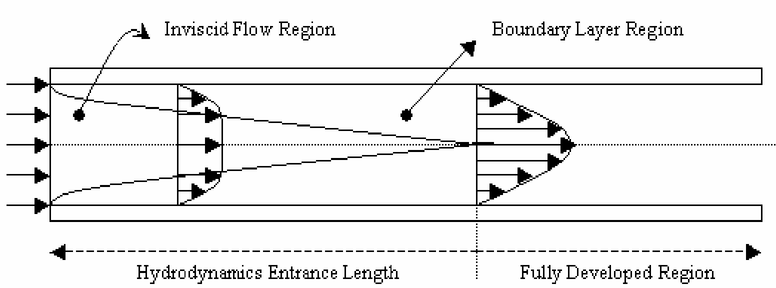
1. A friendly graphical user interface to setup of parameters, the solution control, the convergence monitoring, etc.
2. Graphical representation of results (vector plots, contour plots, path lines etc.)
3. X-Y plots comparison of the numerical results with the theoretical or experimental results

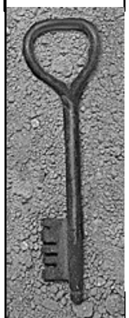


Starting FLOWLAB

- Case 1:

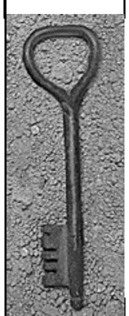
Incompressible Viscous Flow Through Pipes





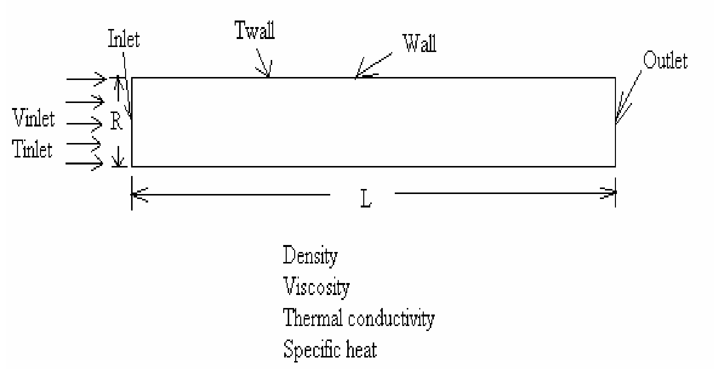
Overview of Case 1

- 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.



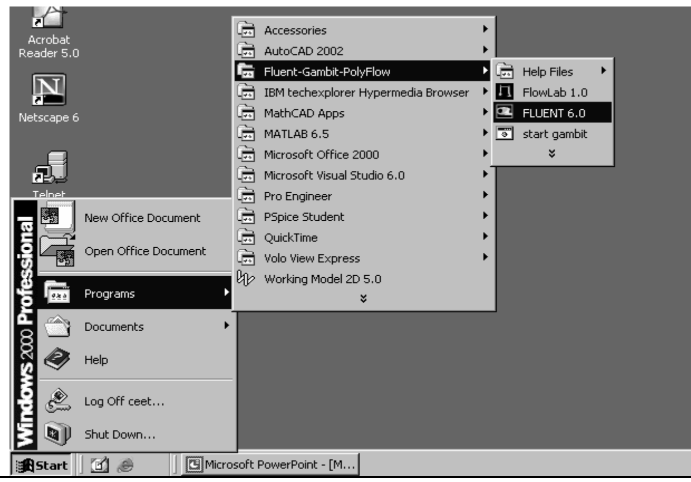
Problem description for Case 1

- 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



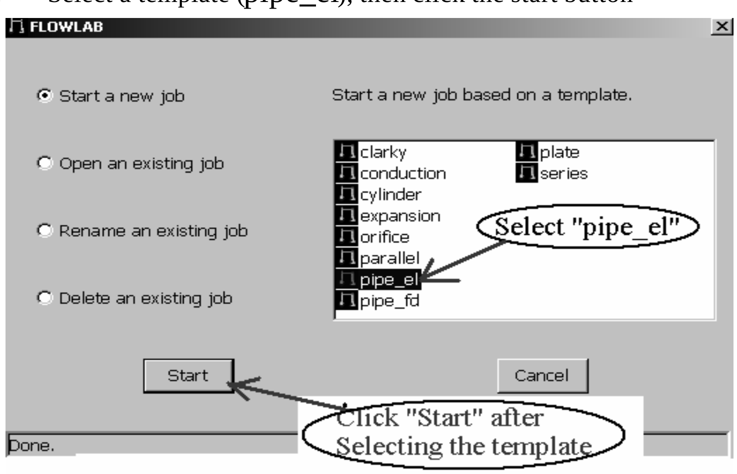
Step 1a: Open a new job

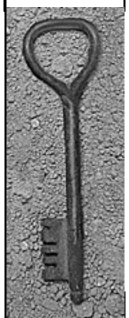
- Click Start>Programs>Fluent-Gambit-PolyFlow>FlowLab 1.0 (this may be different at different computers)



Step 1b: Open a new job

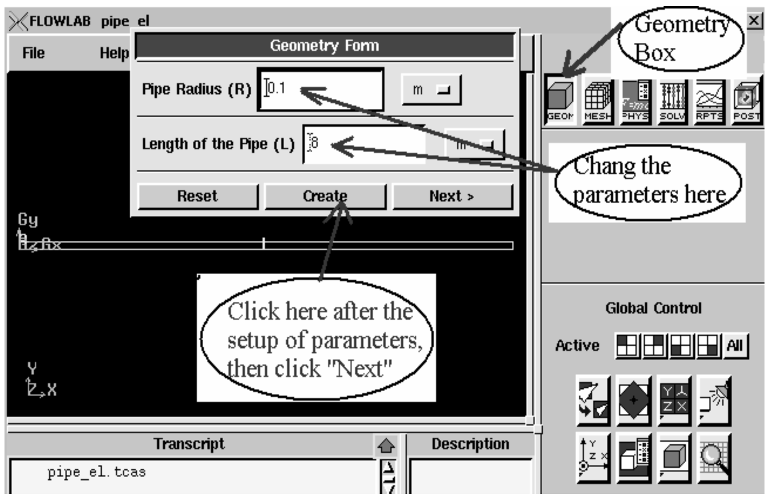
1. Select "Start a new job"
2. Select a template (pipe_el), then click the start button

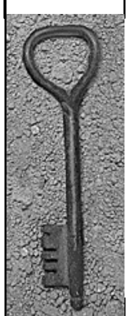




Step 2: Geometry

- 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.

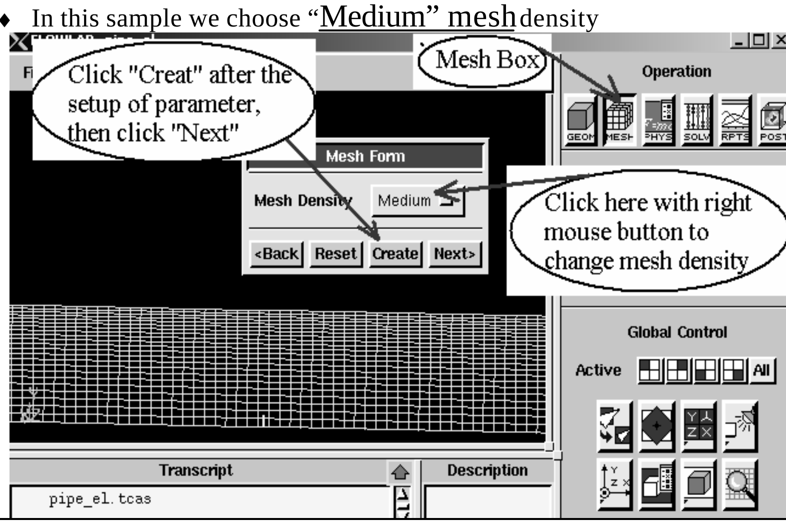


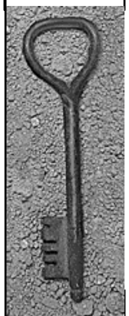


Step 3: Mesh

- There are three choices available for us (for different mesh densities.)

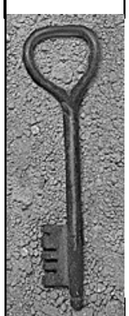
  1. Coarse gird 2. Medium grid 3. Fine grid
- In this sample we choose "Medium" mesh density



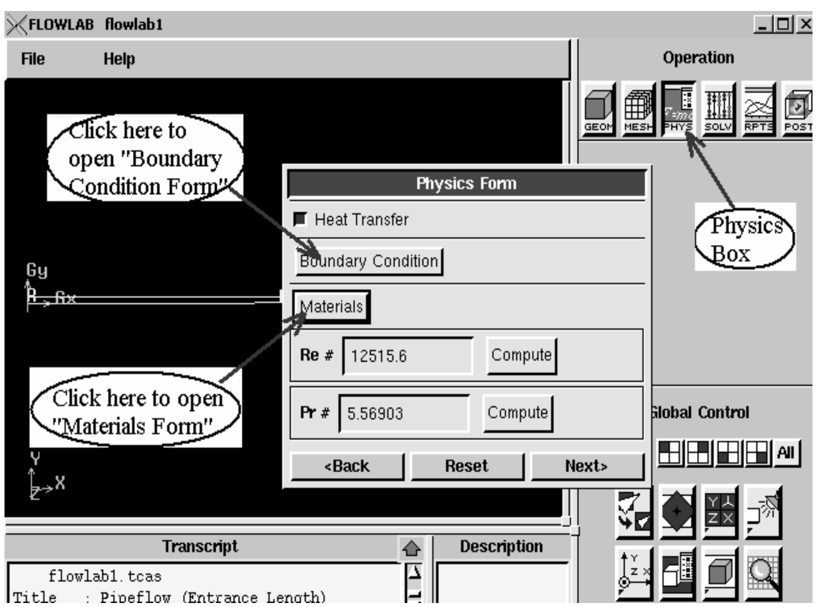


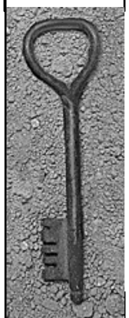
Step 3 Cont'd

- In this step we have <u>generated the grid</u> 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



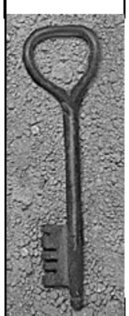


Step 4 Cont'd

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

  1. *The* ***boundary conditions***

  2. The ***material properties***



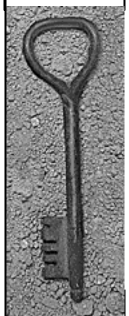
Step 4 Cont'd

- 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/m^2 - 1000W/m^2

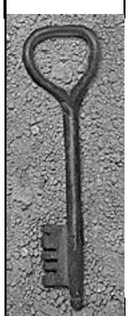
Wall temperature: 100K - 100,00K

Fluid density: 1Kg/m^3 - 2,000Kg/m^3

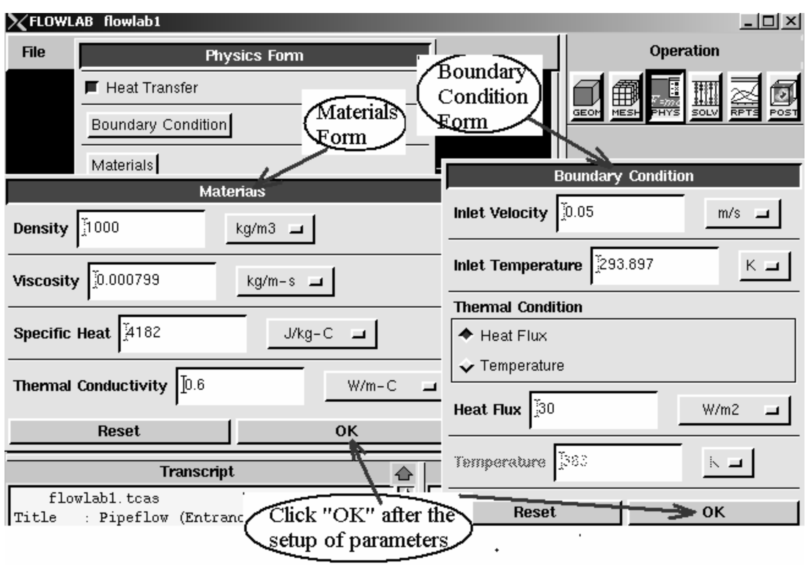
Viscosity: 10^-5 Kg/m-s – 1 Kg/m-s

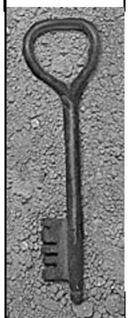
Fluid specific heat: 10J/Kg-C – 100,00J/Kg-C

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

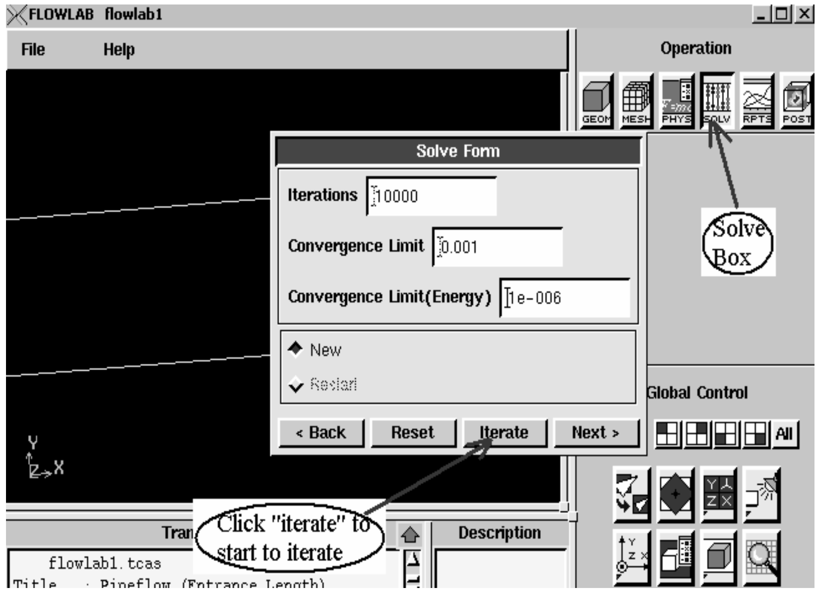


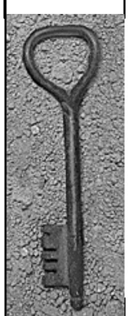
Step 4 Cont’d





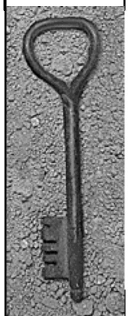
Step 5: Solve





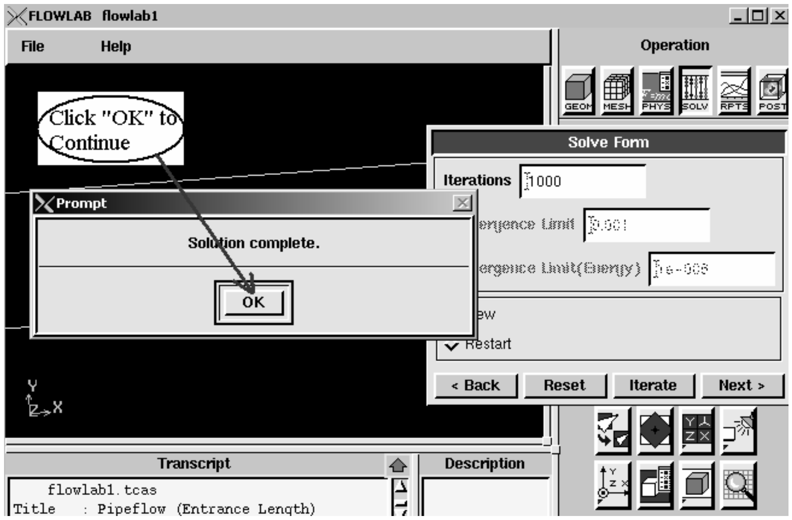
Step 5 Cont'd

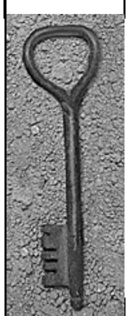
- In this step, the <u>*Navier Stokes equations*</u> are solved throughout the computational domain
- The desired <u>*number of iterations*</u> (1000) and the <u>*convergence limit*</u> (0.00001) for the solver can be specified by the user in the <u>*solve*</u> *form*
- The lower the convergence limit 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



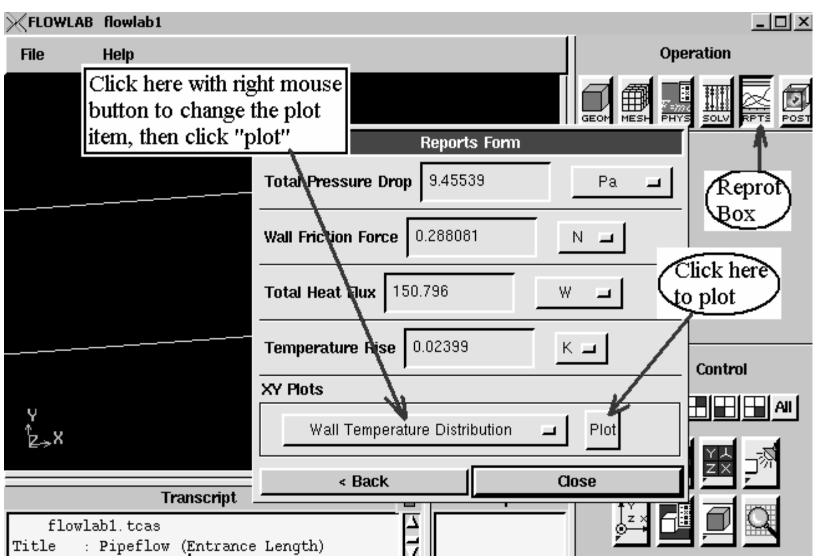
Step 5 Cont'd

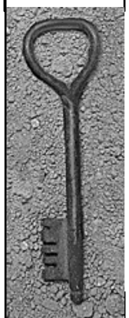
- After clicking the iteration button, a message will come out as soon as the solution is obtained.





Step 6: Report

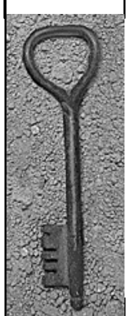




Step 6: Report

- The numerical report:

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



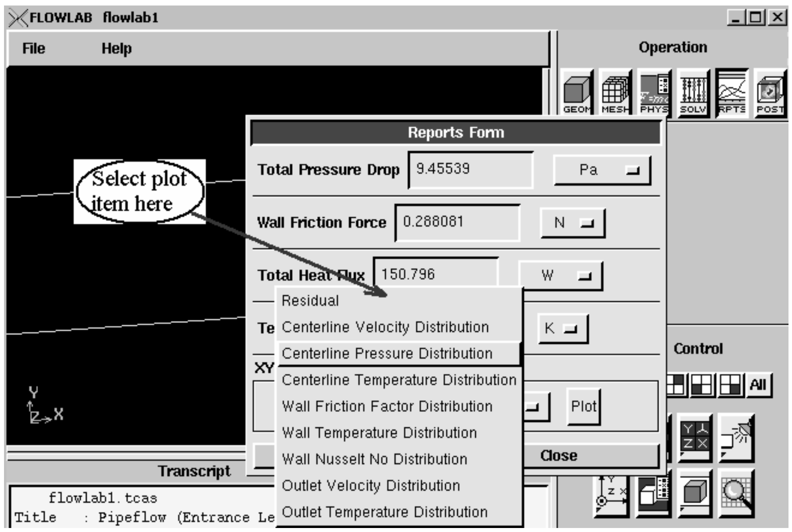
Step 6: Report

- 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

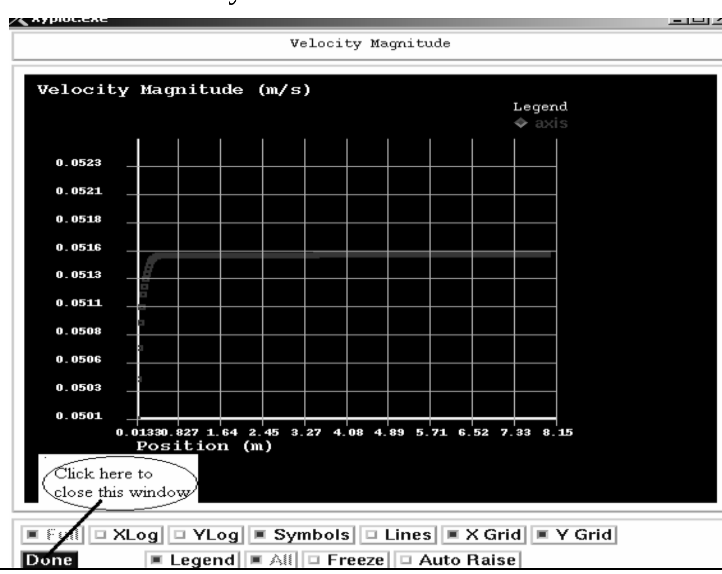
Step 6 Cont'd

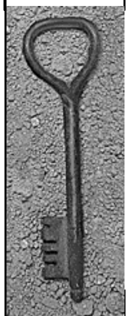
♦ Select the plot



Step 6: Report

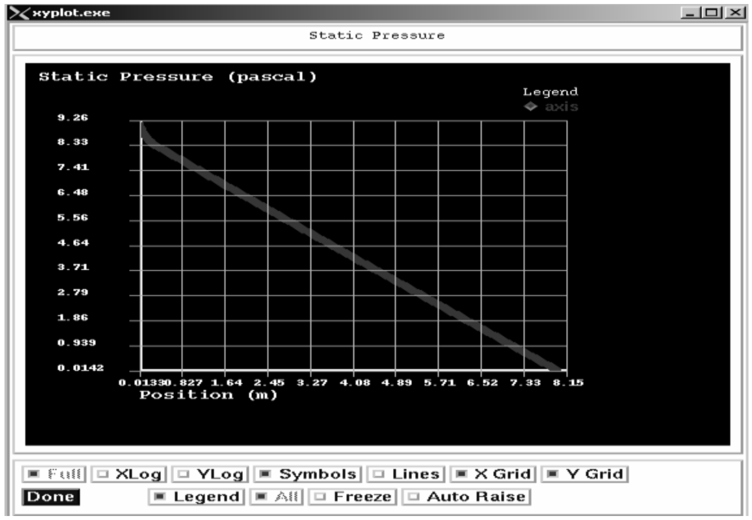
♦ Centerline velocity distribution

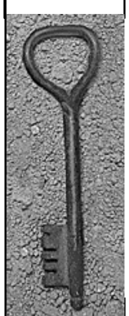




Step 6: Report

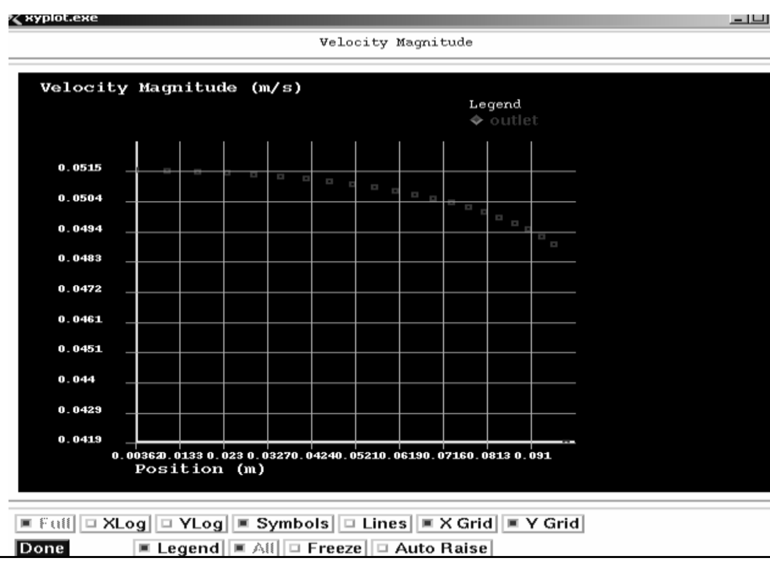
- Centerline pressure distribution

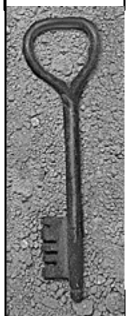




Step 6: Report

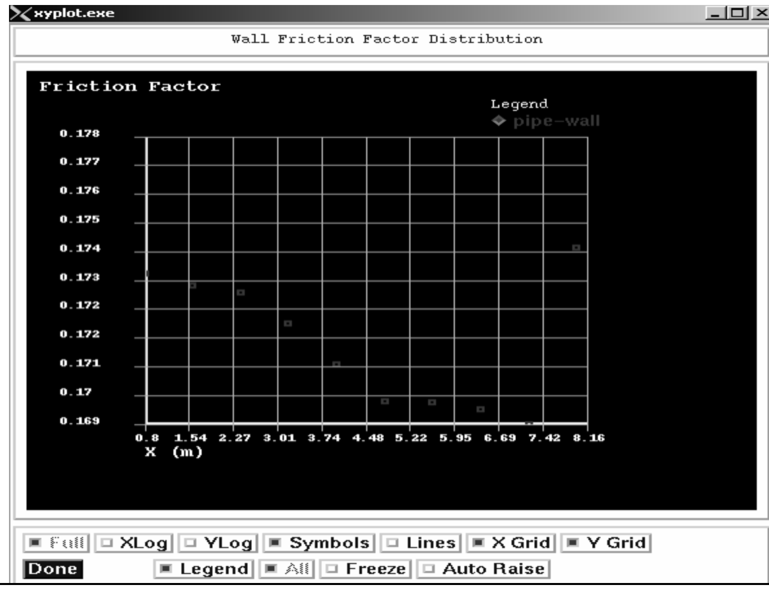
- Outlet velocity distribution

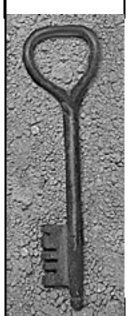




Step 6: Report

♦ Wall friction factor distribution





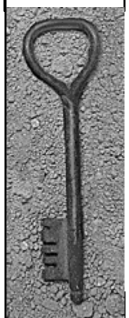
Step 7: Post-processing

♦ Two options are offered in FLOWLAB

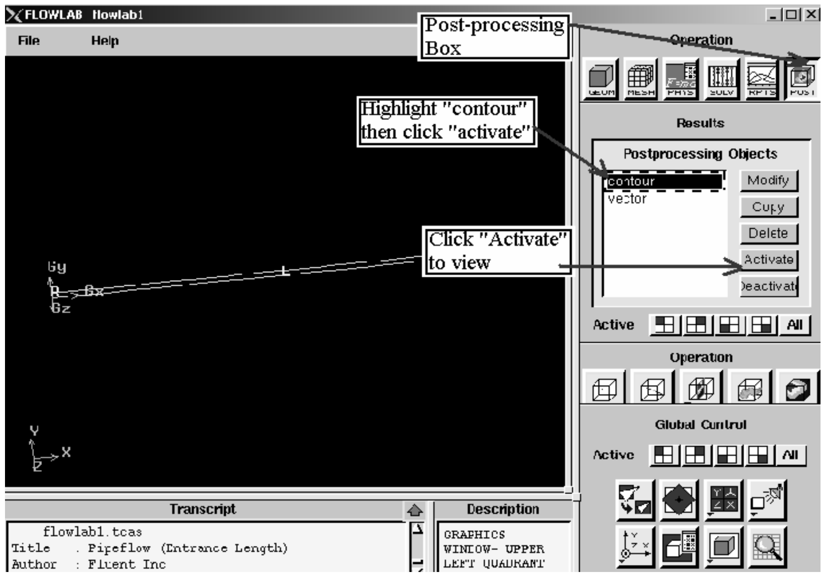
1. Contour
2. Vector

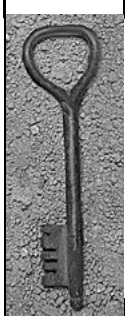
♦ Click and highlight the Contour or Vector, then click “activate” button to view

♦ An example is shown later



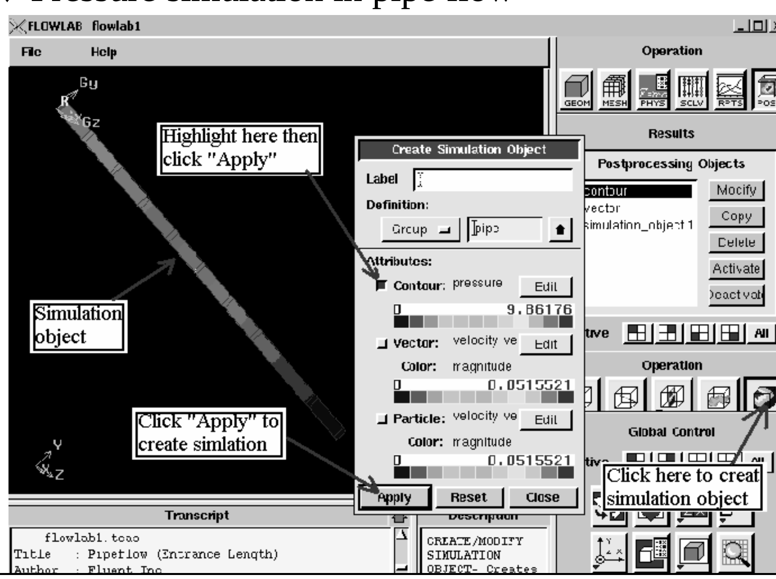
Step 7: Post-processing Cont’d

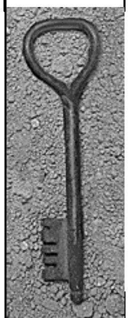




Step 7: Post-processing Cont’d

- Pressure simulation in pipe flow





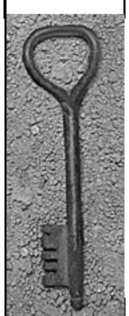
Assignment for the lab

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