

MEE 340



Fluid Mechanics

Virtual Fluids Lab Demonstration
Using FlowLAB by Fluent

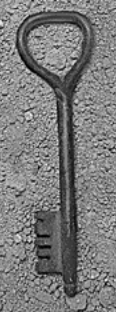
Prepared by Dan Wu

Under the guidance and supervision of Prof. M. Kostic

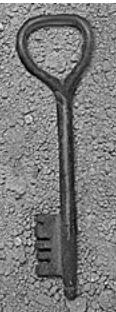


Introduction of Flowlab software

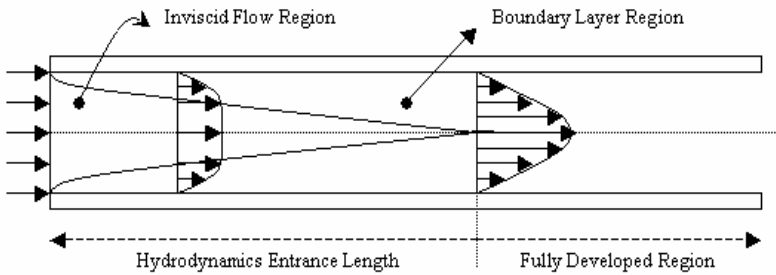
- ◆ The FLOWLAB can help you understand the physical phenomena and reinforce concepts in fluid mechanics.
- ◆ FLOWLAB receives user input by means of its Graphical User Interface (GUI). A part of the FLOWLAB GUI is problem dependent. This is managed by a problem specific template file.
- ◆ FLOWLAB is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

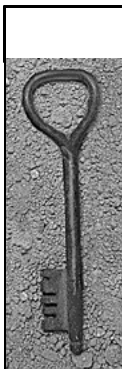


The diagram illustrates the flow of an incompressible viscous fluid through a pipe. It shows the development of the velocity profile from the inlet to the fully developed region. The flow is divided into two main regions: the **Hydrodynamics Entrance Length** and the **Fully Developed Region**. The **Inviscid Flow Region** is the area where the flow is not affected by the pipe walls, and the **Boundary Layer Region** is the area where the flow is affected by the pipe walls. The velocity profile is shown as a series of arrows of increasing length from the inlet to the fully developed region, indicating the development of the velocity profile.



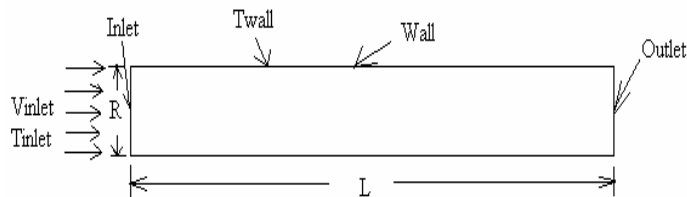
Overview of Case 1

- ◆ 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

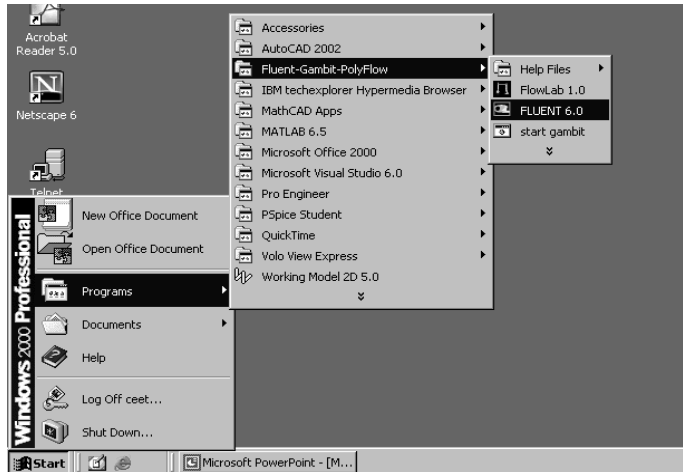


Density
Viscosity
Thermal conductivity
Specific heat



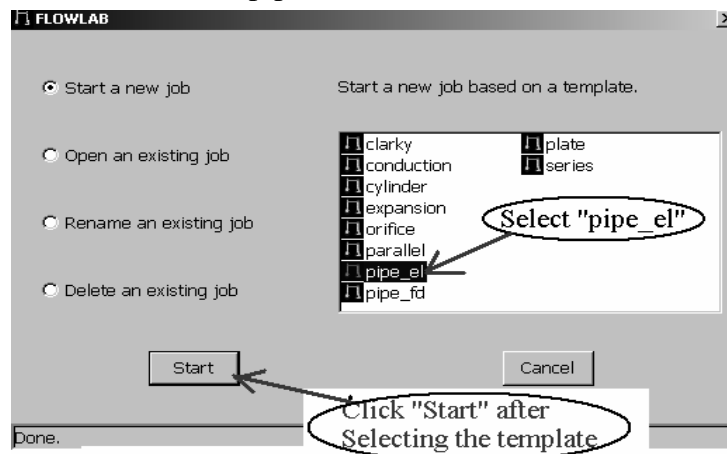
Step 1a: Open a new job

- ◆ 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.

FLOWLAB pipe_el

File Help

Geometry Form

Pipe Radius (R) 0.1 m

Length of the Pipe (L) 10 m

Reset Create Next >

Global Control

Active [Grids] All

Transcript Description

pipe_el.tcas



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

FLOWLAB pipe_el

File Help

Mesh Form

Mesh Density Medium

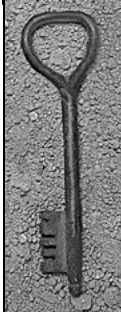
<Back Reset Create Next >

Global Control

Active [Grids] All

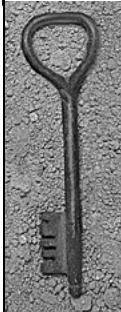
Transcript Description

pipe_el.tcas

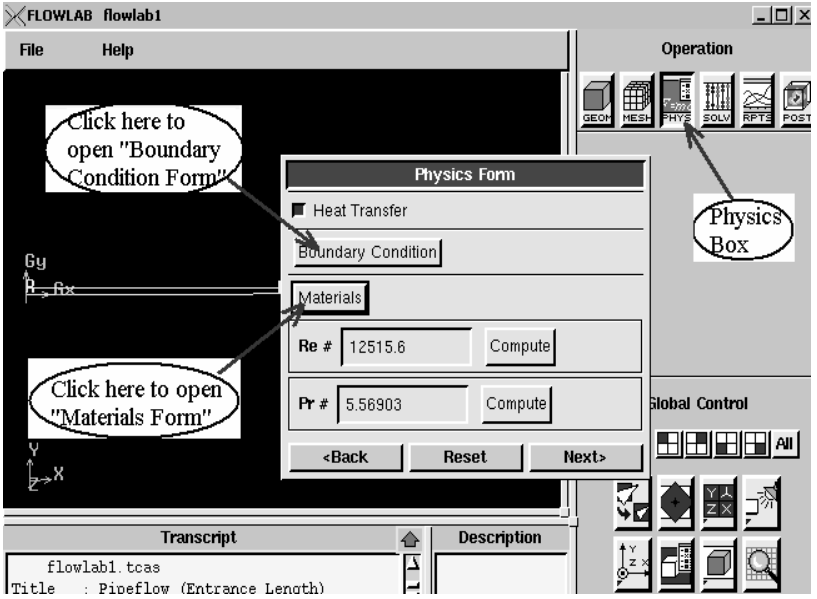


Step 3 Cont'd

- ◆ 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"

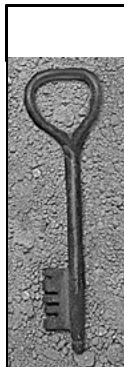
Physics Box


Global Control

Transcript

Description

flowlab1.tcas
Title : Pipeflow (Entrance Length)

	<h2 style="text-align: center;">Step 4 Cont'd</h2> <ul style="list-style-type: none">◆ This step is the modeling step where you can get to specify<ol style="list-style-type: none">1. <i>The boundary conditions</i>2. <i>The material properties</i>
---	---

	<h2 style="text-align: center;">Step 4 Cont'd</h2> <ul style="list-style-type: none">◆ The boundary conditions<p>Under the “Boundary conditions” button you can specify the inlet condition of pipe flow which include <u>inlet temperature and inlet velocity</u>. In this lab you should specify thermal condition by <u>select “temperature” or “flux”</u> button to set the wall temperature of the pipe or the wall heat flux of the pipe.</p>◆ The material conditions<p>Under the “materials” button you can specify the properties of pipe flow which include the <u>density</u>, the <u>viscosity</u>, the <u>specific heat</u> and <u>thermal conductivity</u>.</p>
---	--

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

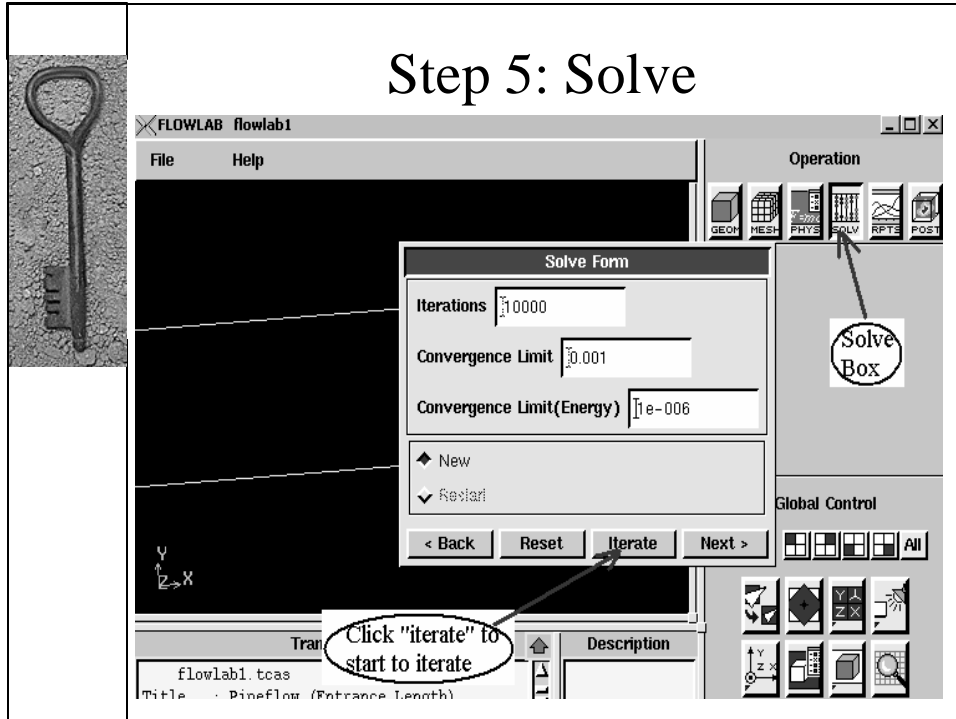
Fluid specific heat: 10J/Kg-C – 100,00J/Kg-C

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

Step 4 Cont'd

The screenshot displays the FLOWLAB software interface. The main window is titled "FLOWLAB flowlab1" and contains several panels. The "Physics Form" panel is active, showing "Heat Transfer" selected. Below it, the "Materials" panel is visible, with fields for Density (1000 kg/m³), Viscosity (0.000799 kg/m-s), Specific Heat (2182 J/kg-C), and Thermal Conductivity (0.6 W/m-C). The "Boundary Condition" panel is also visible, showing "Inlet Velocity" (0.05 m/s), "Inlet Temperature" (293.897 K), and "Thermal Condition" options for Heat Flux (30 W/m²) and Temperature (300 K). A "Transcript" panel at the bottom shows the file name "flowlab1.tcas" and the title "Pipeflow (Entranc...". A callout bubble points to the "OK" button in the Materials panel, with the text "Click 'OK' after the setup of parameters".

Step 5: Solve



The screenshot shows the FLOWLAB software interface. The main window is titled 'FLOWLAB flowlab1'. The 'Operation' toolbar at the top right includes icons for GEOM, MESH, PHYS, SOLV, RPTG, and POST. The 'Solve Form' dialog box is open, displaying the following settings:

- Iterations: 10000
- Convergence Limit: 0.001
- Convergence Limit(Energy): 1e-006

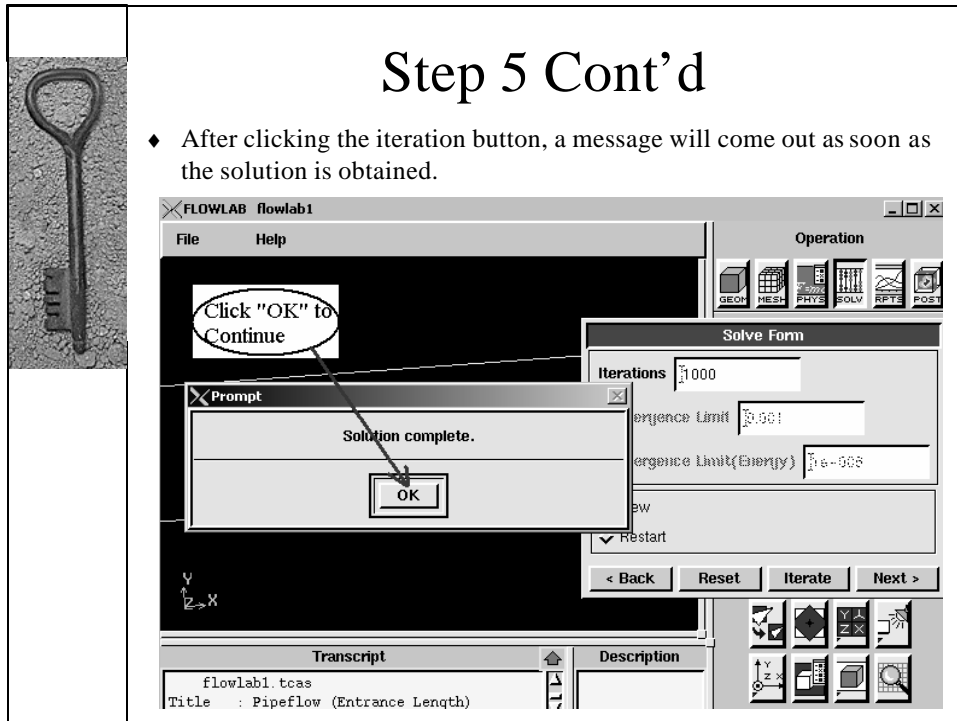
Below these settings are options for 'New' and 'Restart'. At the bottom of the dialog are buttons for '< Back', 'Reset', 'Iterate', and 'Next >'. A callout bubble points to the 'Iterate' button with the text 'Click "iterate" to start to iterate'. The 'Solve Box' icon in the 'Operation' toolbar is also circled.

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

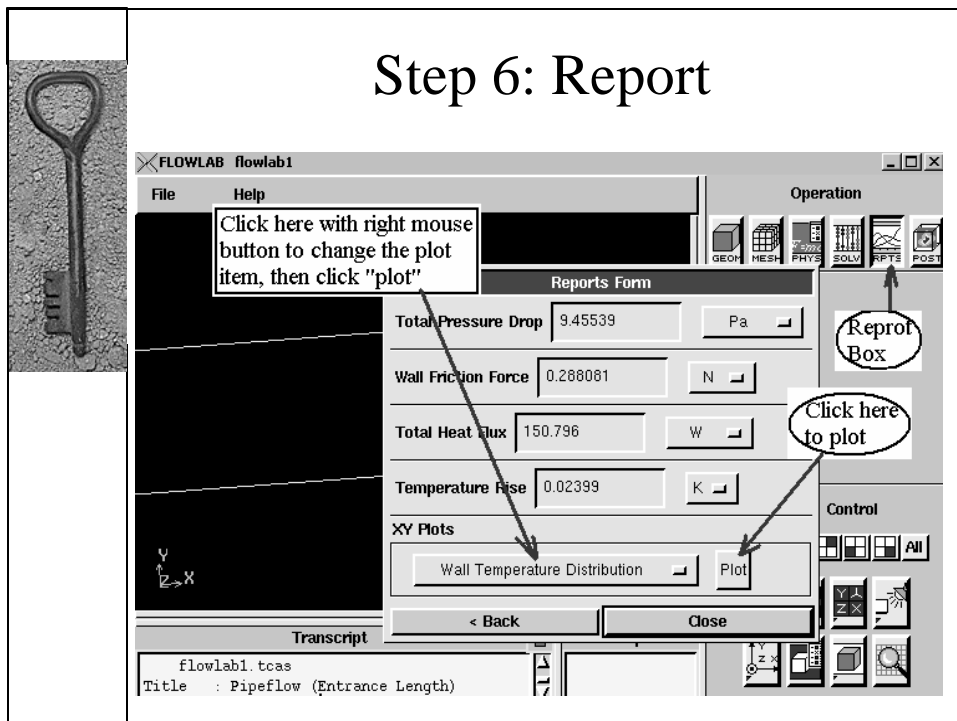
Step 5 Cont'd

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



The screenshot shows the FLOWLAB flowlab1 software interface. A dialog box titled "Prompt" is open, displaying the message "Solution complete." with an "OK" button. A callout bubble points to the "OK" button with the text "Click 'OK' to Continue". In the background, the "Solve Form" panel is visible, showing "Iterations" set to 1000 and "Convergence Limit" set to 0.5%. The "Operation" toolbar includes buttons for GEOM, MESH, PHYS, SOLV, RPT, and POST. The "Transcript" window at the bottom shows the file name "flowlab1.tcas" and the title "Pipeflow (Entrance Length)".

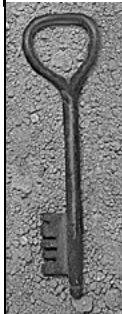
Step 6: Report



The screenshot shows the FLOWLAB flowlab1 software interface with the "Reports Form" dialog box open. The dialog box contains the following data:

Quantity	Value	Unit
Total Pressure Drop	9.45539	Pa
Wall Friction Force	0.288081	N
Total Heat Flux	150.796	W
Temperature Rise	0.02399	K


The "XY Plots" section shows "Wall Temperature Distribution" selected, with a "Plot" button. A callout bubble points to the "Plot" button with the text "Click here to plot". Another callout bubble points to the right side of the dialog box with the text "Click here with right mouse button to change the plot item, then click 'plot'". The "Replot Box" is also visible. The "Operation" toolbar includes buttons for GEOM, MESH, PHYS, SOLV, RPT, and POST. The "Transcript" window at the bottom shows the file name "flowlab1.tcas" and the title "Pipeflow (Entrance Length)".



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

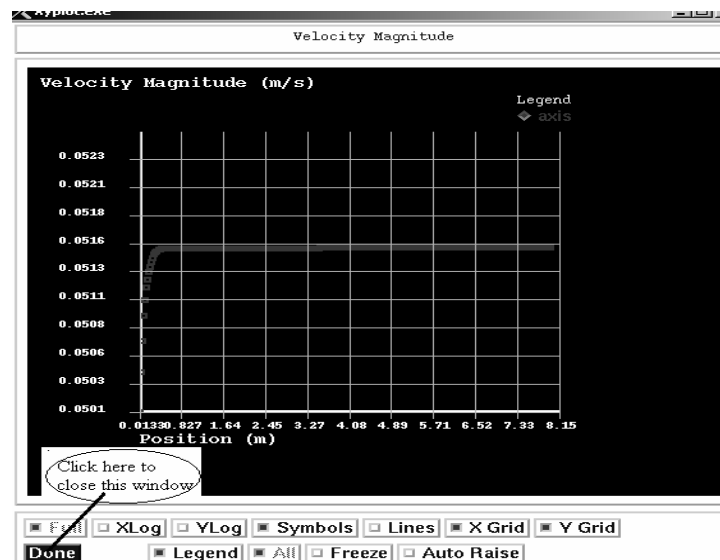
- ◆ Select the plot

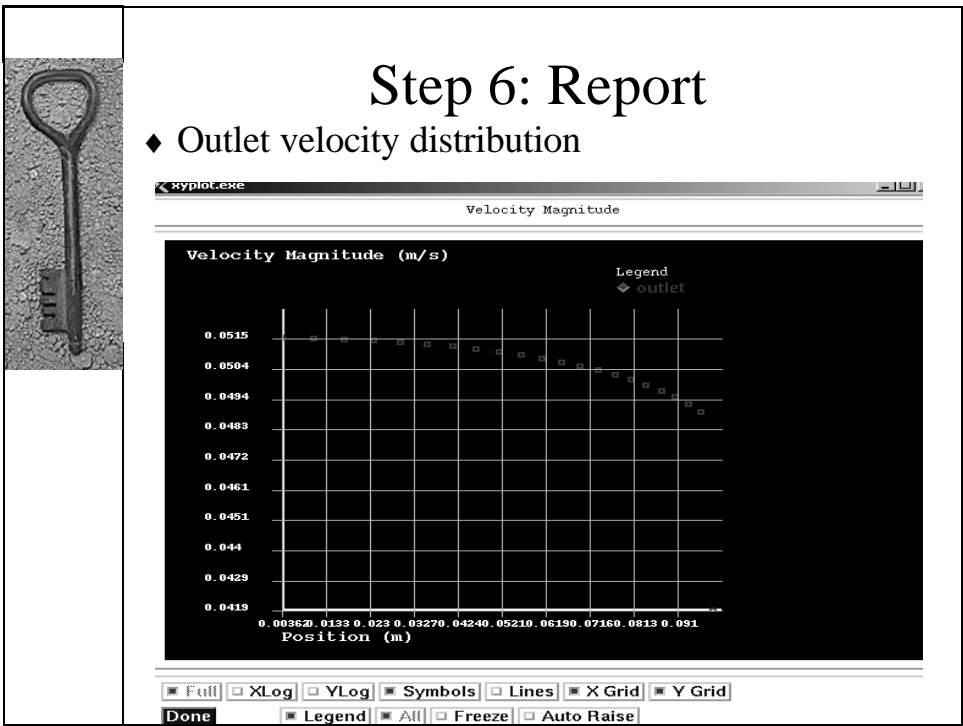
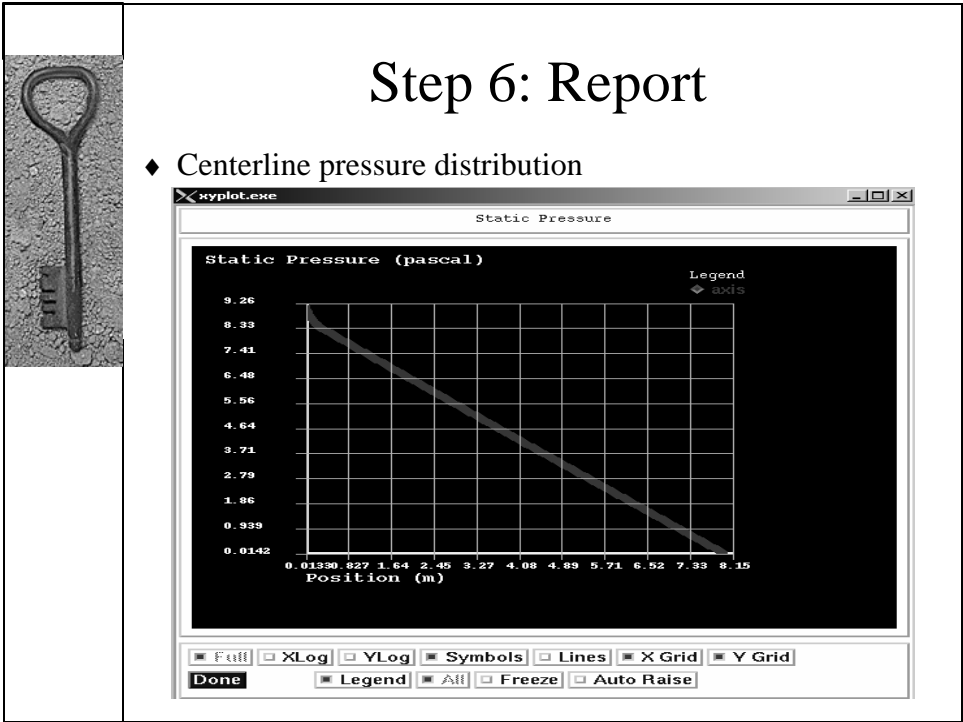
The screenshot shows the FLOWLAB flowlab1 software interface. The 'Reports Form' dialog box is open, displaying various simulation results. A callout bubble with the text 'Select plot item here' points to the 'Centerline Velocity Distribution' option in the list. The list includes: Residual, Centerline Velocity Distribution, Centerline Pressure Distribution, Centerline Temperature Distribution, Wall Friction Factor Distribution, Wall Temperature Distribution, Wall Nusselt No Distribution, Outlet Velocity Distribution, and Outlet Temperature Distribution. The 'Centerline Velocity Distribution' option is highlighted. The 'Plot' button is visible to the right of the list.

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

Step 6: Report

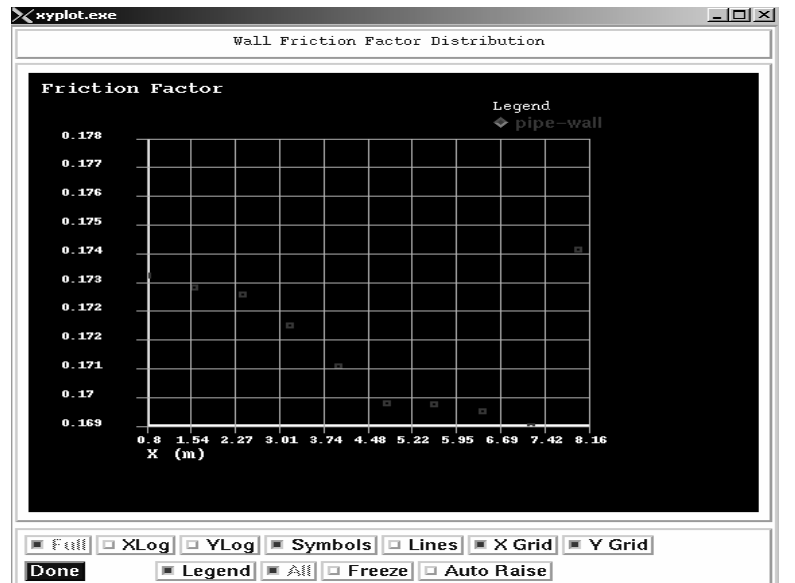
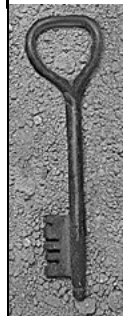
- ◆ Centerline velocity distribution





Step 6: Report

◆ Wall friction factor distribution



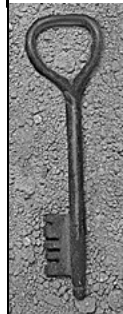
Step 7: Post-processing

◆ 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

Post-processing Box

Highlight "contour" then click "activate"

Click "Activate" to view

Transcript
flowlab1.tcas
Title : Pipeflow (Entrance Length)
Author : Fluent Inc

Description
GRAPHICS
WINDOW- UPPER
LEFT QUADRANT

Step 7: Post-processing Cont'd

◆ Pressure simulation in pipe flow

Highlight here then click "Apply"

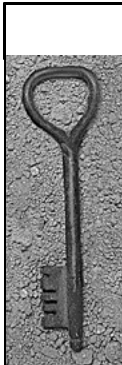
Simulation object

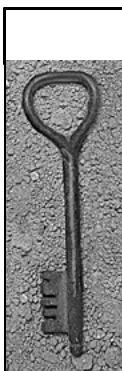
Click "Apply" to create simulation

Click here to create simulation object

Transcript
flowlab1.tcas
Title : Pipeflow (Entrance Length)
Author : Fluent Inc

Description
CREATE/MODIFY
SIMULATION
OBJECT- Create

	<h2 style="text-align: center;">Assignment for the lab</h2> <p>Presentation</p> <ul style="list-style-type: none">◆ 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 <p>Discuss</p> <ul style="list-style-type: none">◆ How does the velocity profile for laminar flow (to be done by you on pipe_fd template) differ with that of turbulent flow?
---	---

	<h2 style="text-align: center;">Thanks!!!</h2> <p>–To my advisor: Prof. M. Kostic for his support</p> <p>–To MEE 340 TA: Srinivasa for his useful discussion</p>
---	--