Pipe Flow Caclulations using GAMBIT and FLUENT

Use Fluent (2d, steady state) to simulate the flow of air through a pipe 1 cm in diameter and 50 cm in length. The velocity of air at the entry to the pipe is assumed to be constant and equal to 1 m/s.

i.) Determine the wall shear force (wall shear stress integrated over area) and campare wall shear force using the Hagen–Poiseuille (fully developed) flow assumption. You will need to refer a fluid mechanics book for this.

ii.) What is the error in wall shear force introduced by the fully developed flow assumption?

Hints:
The above problem can be easily solved in Fluent by assuming the flow to be axisymmetric and using the axisymmetric solver.

Gambit:

First use **Geometry →Face** command in GAMBIT to declare a rectangle which is 50 units in length and 0.5 units in breadth. Change to **+X +Y** in the **Direction** option.

Use the **Mesh Edge** panel in Gambit to generate a uniform mesh along the edges of the rectangle. Divide the length of the rectangle into 500 equally spaced intervals and the breadth in to 10 equally spaced intervals.

After edge meshing all four sides open the **Mesh Faces** panel and select the rectangle. Use the default face meshing scheme of **Quad** and **Map**. Click **Apply** to mesh the domain. A uniform rectangular mesh will be generated.

In **Zones** declare the **left edge** as **Velocity inlet**, **right** as **Outflow, top** as **Wall** and **bottom** as **Axis** (flow is assumed to be axisymmetric. Don't forget to choose Fluent 5/6 as the solver or else none of the boundary types will be displayed).

Export the 2d mesh.

Fluent:

Open Fluent 2d.

Open the mesh file using **Read → Case → <filename.msh>**

Next check the grid by **Grid → Check.** Scale the grid to cm.

Define the solver to be used by **Define→ Models→ Solver,** and leave the default solver settings to **Segregated.** Change to **Axisymmetric** under **Space.** Leave all other options at their default settings**.**

Declare the boundary conditions from the boundary conditions panel. Enter the velocity magnitude as 1 m/s for the velocity inlet boundary. Leave all other options at their default settings.

Initialize the flow field. Choose **all-zones** in the **Compute from** box in the **Solution Initialization** panel.

Then iterate .The solution should converge in about 33 iterations. (if you had followed all the steps above).