- Start **Workbench** from **Start Menu**

ANSYS Schematic Layout

- Drag 1 **Geometry** component
  - Rename **"pipe"**
- Drag 2 **Mesh** component
  - Rename 1 **"Uniform"**
- Drag 2 **Fluent** component
  - Rename 1 **"Laminar"**
- Create Folder on H:Drive called *CFD Pre-Lab and Lab 1*
- Save project file in new folder and call it *CFD Pre-Lab and Lab 1 Pipe Flow*

Geometry Creation

- Check Default Unit **Meter**
- **Sketching > Constraints > Auto Constraints**: Check Cursor
- Create **New Sketch** on **XY Plane** and **Look At**
- Draw **Rectangle** and dimension as follows:
  - Length = 7.62m
  - Radius = 0.02619m
- **Concept** > **Surface from Sketch** > select *Sketch 1* click **Apply** and **Generate**
- **File** > **Save Project**

Mesh Generation

- **Insert** > **Mapped Face Meshing**
- **Insert** > **Sizing** for top and bottom
  - **No of Divisions** = 453 **Behavior** = **Hard** **No Bias**
- **Insert** > **Sizing** for left and right
  - **No of Divisions** = 45 **Behavior** = **Hard** **No Bias**
- **Generate Mesh**
- **Create Named Selection**
  - *inlet*, *outlet*, *wall*, and *axis*
- **Update Mesh** on **Project Schematic**

Solution Setup

- Change **2D Space** to **Axisymmetric**
- Leave models as **Laminar**
- Change material properties as follows
  - **Density** = **1.17**
  - **Viscosity** = **1.872e-05**
- Cell zone conditions change to fluid **Air**
- Change boundary conditions as follows
  - **Inlet Velocity** = **0.2**
  - **Outlet Pressure** = **0**
  - **Wall** stays the same
  - **Operating Conditions** = **97225.9**
- Change **Reference Values** as follows:
  - **Area** = **0.002154869**
  - **Density** = **1.17**
  - **Length** = **0.05238**
  - **Temperature** = **298.16**
  - **Inlet Velocity** = **0.2**
  - **Viscosity** = **1.872e-05**
- **Solution Methods**
  - **Green Gauss Cell Based**
  - **Second Order**
  - **Second Order Upwind**
- **Monitors**
  - All three eqns = **1e-06**
- **Solution Initialization**
  - **Standard**
  - **Axial Velocity** = **0.2**
- **Run Calculation**
  - **Number of Iterations** = **1000**

Data to save

- **Residuals image**
- **Centerline pressure distribution image**
- **Centerline velocity distribution image**
- **Wall friction factor distribution image**
- **Export wall friction factor distribution and calculate shear stress C=8*t/(rho*U^2) AFD value is 0.097747231**
- **Axial velocity with AFD image**

Surface Name	X0	Y0	X1	Y1
x=10d	0.5238	0	0.5238	0.02619
x=20d	1.0476	0	1.0476	0.02619
x=40d	2.0952	0	2.0952	0.02619
x=60d	3.1428	0	3.1428	0.02619
x=100d	5.238	0	5.238	0.02619

- **Export velocity profile at x=100d and normalize it then save image**
- **Contours of radial velocity**
- **Velocity vector at region where flow is becoming fully developed**