* 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**