**Extract BC from EFD Data Reduction Sheet**

* Calculate inlet velocity from venturi meter flow rate measurement
* Outlet pressure is pressure measurement at tap #4

**ANSYS Schematic Layout**

* Drag 1 **Geometry** component
	+ Rename **“pipe”**
* Drag 2 **Mesh** component
	+ Rename 1 **“non-uniform”**
* Drag 2 **Fluent** component
	+ Rename 1 **“turbulent”**
* Create Folder on H:Drive called *CFD Lab and Lab 1*
* Save project file in new folder and call it *Lab 1*

**Geometry Creation**

* Select **Meter**
* Create **New Sketch** on **XY Plane** and **Look At**
* Draw **Rectangle** and dimension as follows:
	+ Length = 6.096m
	+ 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** = 564
	+ **Behavior** = **Hard**
	+ **Bias Type** =**No Bias**
* **Insert** > **Sizing** for left and right
	+ **No of Divisions** = 15
	+ **Behavior** = **Hard**
	+ **Bias Type** =**Fine on wall**
	+ **Bias Factor** =**3.1117**
* **Generate Mesh**
* **Create Named Selection**
	+ *inlet*, *outlet*, *wall*, and *axis*
* **Update Mesh** on **Project Schematic**

**Solution Setup**

* Change **2D Space** to **Axisymmetric**
* Models is **k-e model with default parameters**
* Use average measured temperature to calculate density and viscosity
* Cell zone conditions change to fluid **Air**
* BC: axisymmetric, measured inlet velocity, outlet pressure, and given wall roughness
	+ **Inlet:** measured inlet velocity, turbulent intensity=0.01%, turbulent length scale=0.000294m
	+ **Outlet:** measured pressure at tap4, turbulent intensity=5%, turbulent viscosity ratio=10
	+ **Wall:** no-slip, roughness=2.5e-5m, roughness constant=0.5
	+ **Axis:** symmetry
	+ **Operating conditions:** reference pressure is 97225.9 Pa
* Change **Reference Values** as follows:
	+ **Area** = **0.002154869**
	+ **Density** = **Based on Measured Temperature**
	+ **Length** = **0.05238**
	+ **Temperature** = **Measured**
	+ **Inlet Velocity** = **Based on Measured Temperature**
	+ **Viscosity** = **Based on Measured Temperature**
* **Solution Methods**
	+ **Green Gauss Cell Based**
	+ **Second Order schemes for all equations**
* **Monitors**
	+ All three eqns = **1e-06**
* **Solution Initialization**
	+ **Standard**
	+ **Measured inlet velocity and outlet pressure**
	+ **Turbulent kinetic energy=0.09m^2/s^2**
	+ **Turbulent dissipation rate=16m^2/s^3**
* **Run Calculation**
	+ **Number of Iterations** = **1000**

**Data to save**

* **Residuals image**
* **Centerline pressure with EFD image**
* **Centerline velocity distribution image**
* **Wall shear stress distribution image**
* **Export wall friction factor distribution and calculate shear stress C=8\*t/(rho\*U^2)**
* **Axial velocity with EFD 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**
* **Image of normalized CFD laminar and turbulent velocity profile at the developed region**