

Turbulent Pipe Flow - Verification & Validation

Author: Rajesh Bhaskaran, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

[Comments](#)

Verification & Validation

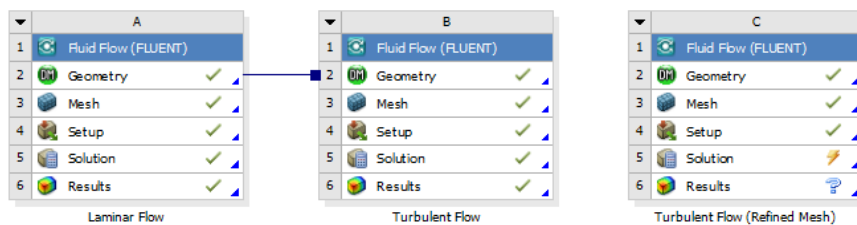


Useful Information

[Click here](#) for the FLUENT 12 version.

In order to assess the numerical accuracy of the results obtained, it is necessary to compare results on different meshes. We'll re-do the calculation on a 100x60 mesh which has twice the number of nodes in the radial direction as the 100x30 mesh.

In Workbench, under **Turbulent Flow** project, right click on **Fluid Flow (FLUENT)** and click duplicate. Rename the duplicate project to **Turbulent Flow Refined Mesh**. You should have three project cells in workbench.



Double click on **Mesh** for **Turbulent Flow Refined Mesh**. The ANSYS Mesher window will open. Under **Outline**, expand mesh tree and click on **Edge Sizing 2**.

Highlight "Edge Sizing 2". Under Details of "Edge Sizing 2", increase **Number of Divisions** to 60. This will refine the mesh in the radial direction at the inlet.

Highlight "Edge Sizing 3". Under Details of "Edge Sizing 3", increase **Number of Divisions** to 60. This will refine the mesh in the radial direction at the outlet.

Click **Update** to generate the new mesh.

Close the ANSYS Mesher and go back to Workbench windows. Under **Turbulent Flow Refined Mesh**, right click on **Fluid Flow (FLUENT)** and click **Update**. Wait for a few minutes for FLUENT to obtain a solution and update all the results.

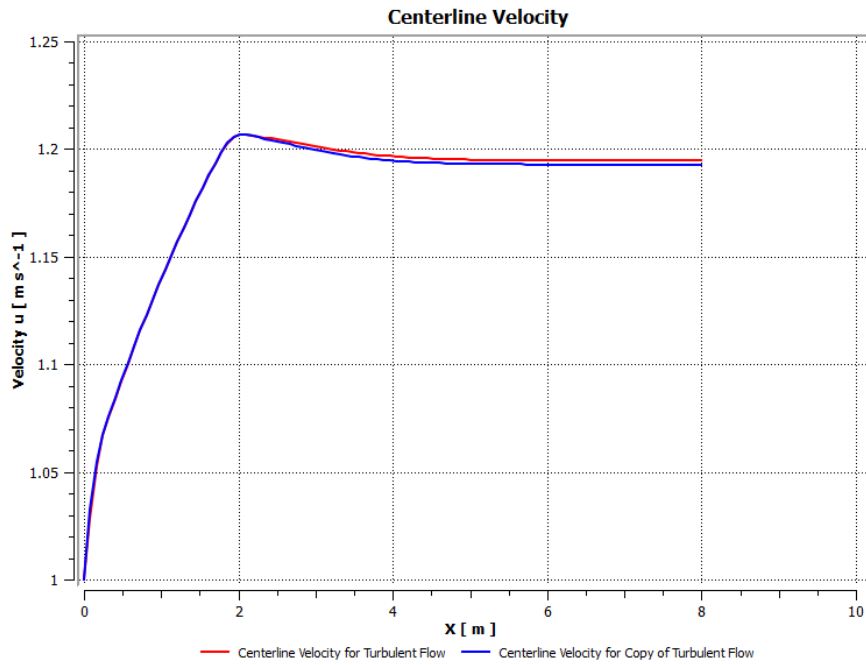
We would want to compare the solution on the two meshes. To do that, drag the **Solution** cell of **Turbulent Flow Refined Mesh** to **Results** cell of **Turbulent Flow**.

Double click the **Results** cell of **Turbulent Flow**, and after CFD Post opens, we can compare our results by simply selecting the desired chart!

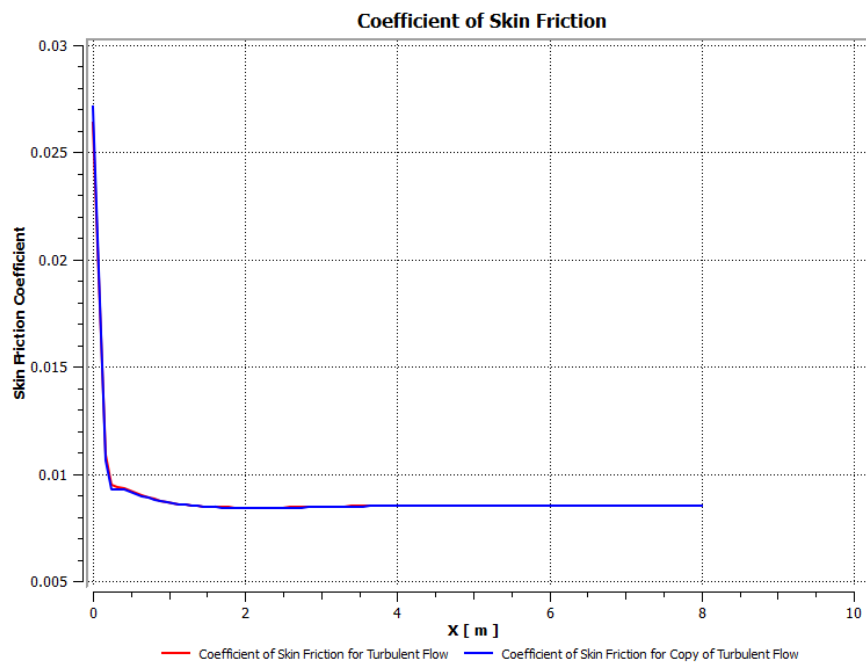
Result Comparison

The following images show comparisons of Centerline Velocity, Coefficient of Skin Friction, Outlet Velocity, and Wall Y-plus.

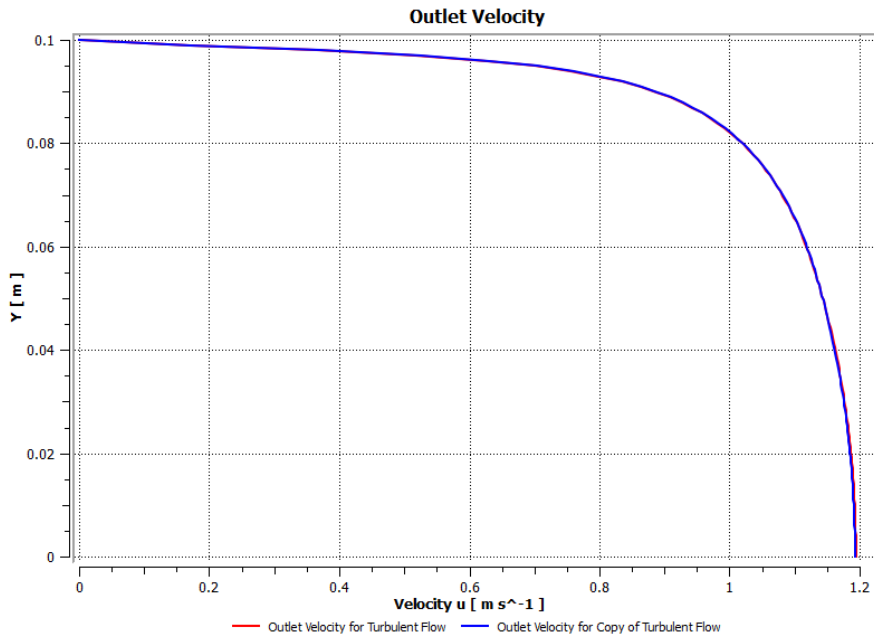
Centerline Velocity



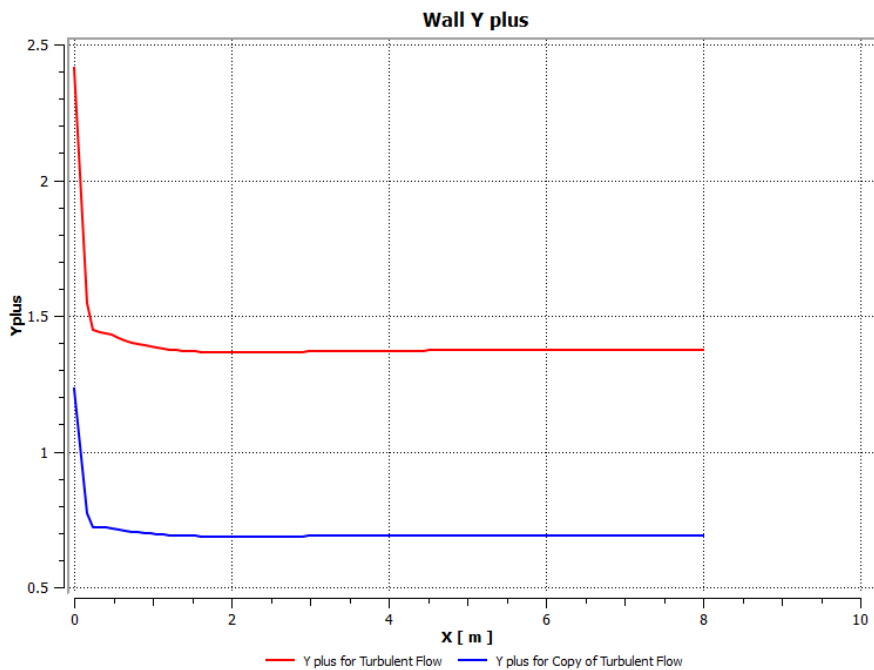
Coefficient of Skin Friction



Outlet Velocity



Wall Y plus



From the first three plots, we can see that the velocity and skin friction coefficient results have remained nearly unchanged. However, the Y-plus results show significant improvement.

You may want to experiment with meshes of other granularities and compare their plots with the plots saved from the 100x30 and 100x60 meshes.

In Problem 1, we will be looking at the effect of coarse meshes with uniform granularity.

[Go to Exercises](#)

[Go to all FLUENT Learning Modules](#)