Turbulent Pipe Flow (LES) - Geometry

Author: Ranjith Tirunagari, 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

Geometry

For users of ANSYS 15.0, please check this link for procedures for turning on the Auto Constraint feature before creating sketches in DesignModeler.

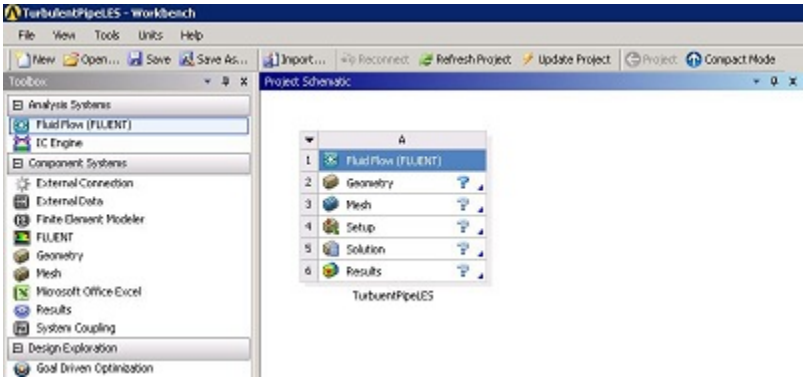
Saving

It would be of best interest, to save the project at this point. Click on the "Save As.." button, 

, which is located on the top of the ***Workbench Project Page*** . Save the project as "TurbulentPipeFlow" in your working directory. When you save in ANSYS, a file and a folder will be created. For instance if you save as "TurbulentPipeLES", a "TurbulentPipeLES.wbpj" file and a folder called "TurbulentPipeLES_files" will appear. In order to reopen the ANSYS files in the future you will need both the ".wbpj" file and the folder. If you do not have BOTH, you will not be able to access your project.

Fluid Flow (FLUENT) Project Selection

Left click (and hold) on ***Fluid Flow (FLUENT)*** , and drag the icon into the empty space in the ***Project Schematic*** . Your ANSYS window should now look comparable to the image below.

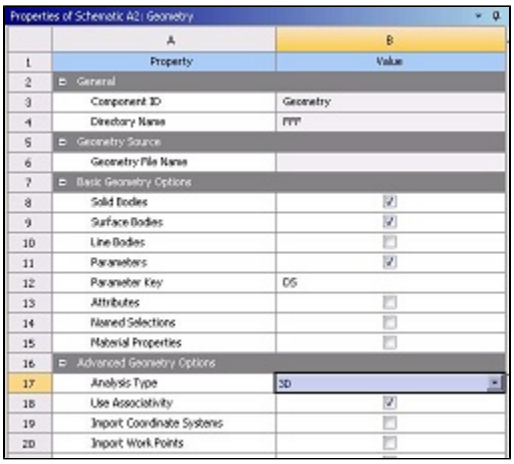


Higher Resolution Image

Rename the project to TurbulentPipeLES. We will work through each step from top down to obtain the solution to the problem.

Analysis Type

As noted earlier, the computational domain for this simulation will be a full 3-dimensional pipe. In the ***Project Schematic*** of the Workbench window, right click on ***Geometry*** and select ***Properties*** . The properties menu will then appear to the right of the Workbench window. Under ***Advance Geometry Options*** , make sure that the ***Analysis Type*** is 3D as shown in the image below.



Higher Resolution Image

Launch Design Modeler

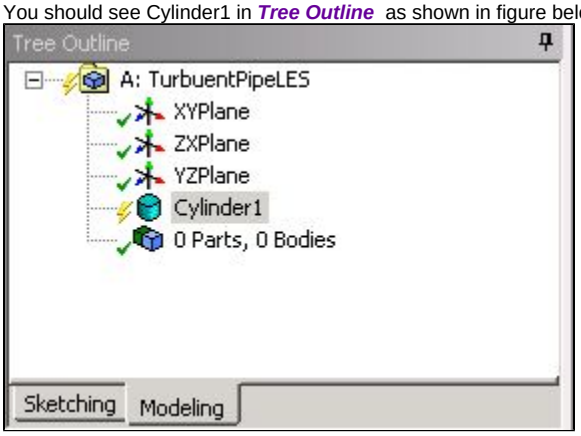
In the ***Project Schematic*** , double click on ***Geometry*** to start preparing the geometry.
At this point, a new window, ANSYS Design Modeler will be opened. You will be asked to select desired length unit. Use the default meter unit and click ***OK*** .

Creating Cylinder

From the ***Create*** menu, choose ***Primitives*** and then ***Cylinder*** .

Create> Primitives> Cylinder

You should see Cylinder1 in ***Tree Outline*** as shown in figure below.



Now we have to define the dimensions for the cylinder according to the problem specifications. Let us assume that the axis of the cylinder is in x-direction.

In the Details View table, change Base Plane to YZPlane, Axis X Component to 0.0635 m and Radius (>0) to 0.00635m. Click ***Generate*** , this will create a cylinder with YZ plane as the base plane and X-axis as the axis of the cylinder. The cylinder has a diameter of 0.0127 m and is 0.0635 m long. Inputs in the ***Details View*** table are shown below.

Details View

Details of Cylinder1	
Cylinder	Cylinder1
Base Plane	YZPlane
Operation	Add Material
Origin Definition	Coordinates
FD3, Origin X Coordinate	0 m
FD4, Origin Y Coordinate	0 m
FD5, Origin Z Coordinate	0 m
Axis Definition	Components
FD6, Axis X Component	0.0635 m
FD7, Axis Y Component	0 m
FD8, Axis Z Component	0 m
FD10, Radius (>0)	0.00635 m
As Thin/Surface?	No

.

At this point, you can close the ***Design Modeler*** and go back to ***Workbench Project Page*** . Save your work thus far in the ***Workbench Project Page*** .

Go to Step 3: Mesh

Go to all FLUENT Learning Modules