Flat Plate Boundary Layer - Verification & Validation

Author: Rajesh Bhaskaran, Cornell University

Verification & Validation

> ℹ **MAE 3240/4230/5230 Spring 2020:** Please read through this section for your understanding. You <u>DO NOT</u> need to complete this section in ANSYS for the homework (feel free to give it a try though if you are curious).
>
> **MAE 3240 students need to complete the next section: Part II, Flat Plate Convection**

It is **very important** that you take the time to check the validity of your solution. This section leads you through some of the steps you can take to validate your solution.

Refine Mesh

Let's repeat the solution on a finer mesh. For the finer mesh, we will increase the total number of elements(cells) by a factor of four. In order to accomplish this, we double the number of divisions on each section. Instead of modifying the project that was just created, we will duplicate it and modify the duplicate. In the *Workbench Project Page* right click on *Mesh* then click *Duplicate* as shown below.
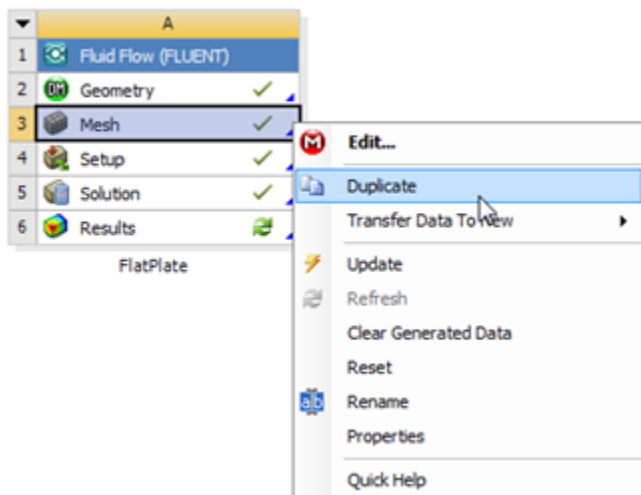
Higher Resolution Image

Rename the duplicate project to *FlatPlate (mesh 2)*. You should have the following two projects in your *Workbench Project Page*.

FlatPlate
FlatPlate (mesh2)
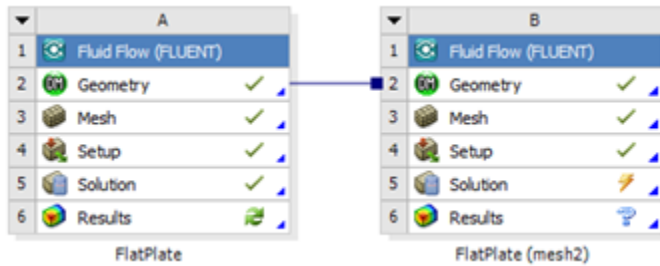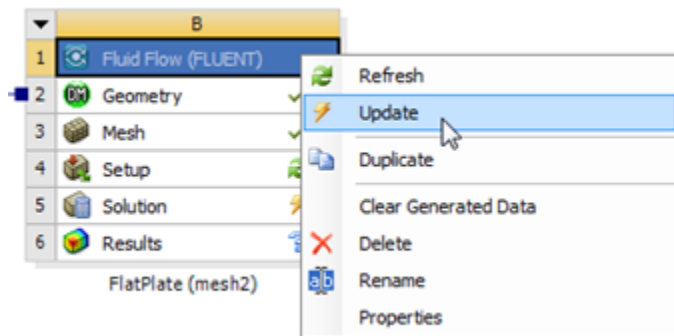
Next, double click on the *Mesh* cell of the *FlatPlate (mesh 2)* project. A new ANSYS Mesher window will open. Under *Outline*, expand *Mesh* and click on *Edge Sizing*. Under Details of "Edge Sizing", enter 100 for *Number of Divisions*. Next set the number of divisions for *Edge Sizing 2* and *Edge Sizing 3* to 120. Then, click *Update* to create the new mesh. The new mesh should now have 12000 elements (100 x 1200). A quick glance of the mesh statistics reveals that there is indeed 12000 elements.

| Statistics	
Nodes	12221
Elements	12000
Mesh Metric	None

Higher Resolution Image

Compute the Solution

Close the ANSYS Mesher to go back to the *Workbench Project Page*. Under *FlatPlate (mesh 2)*, right click on *Fluid Flow (FLUENT)* and click on *Update*, as shown below.

FlatPlate (mesh2)

Now, wait a few minutes for FLUENT to obtain the solution for the refined mesh. After FLUENT obtains the solution, save your project.

Convergence

In order to launch FLUENT double click on the **Solution** of the "FlatPlate (mesh 2)" project in the **Workbench Project Page**. The new mesh has significantly more cells, thus it is likely that the solution did not converge to the tolerances we have previously set. Therefore, we will iterate the solution further, to make sure that the solution converges. In order to do so click on **Run Calculation**, set **Number of Iterations** to 1000 and click **Calculate**, as shown below.
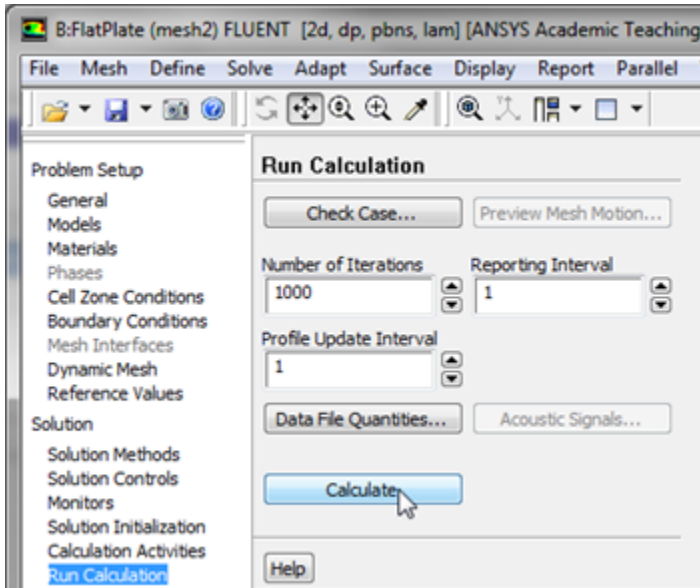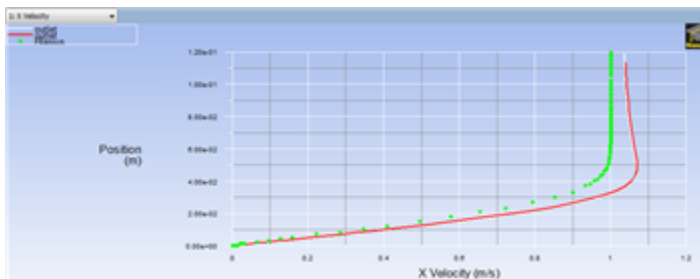
Once you rerun the calculation, you will quickly see that the solution did not converge for the finer mesh within 1000 iterations. The solution should converge by the 1784th iteration as shown below.

```
! 1784 solution is converged
  1784 9.9753e-07 1.7521e-09 3.9381e-10  0:00:11  216
```

Outlet Velocity Profile

Now, the variation of the x component of the velocity will be plotted with the results of the original mesh to determine whether the solution is mesh converged. Set up a plot for the variation of the x component of the velocity along the outlet as was done in the solution section. Then load the *XVelOutlet. xy* file into the plot and generate the plot. You should obtain the following image.

As one can see from the image above, the numerical solution does not vary much at all between the two meshes. Thus, it has been confirmed that the solution is mesh converged.

Go To Step 8: Flat Plate Convection

Go to all FLUENT Learning Modules