Old (4/1/2020) Flat Plate Boundary Layer - Numerical Solution

Author: Rajesh Bhaskaran, Cornell University

Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Model Setup
5. Numerical Solution
6. Post Processing
7. Verification & Validation
8. Part II: Flat Plate Convection

Frequently Asked Questions

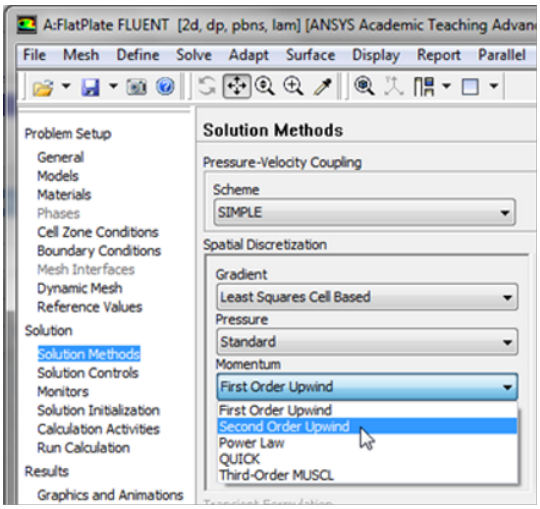
Exercises

Comments

Numerical Solution

Second Order Scheme

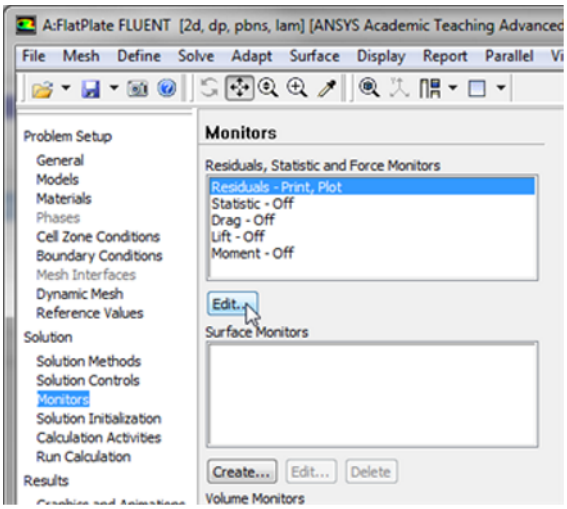
A second-order discretization scheme will be used to approximate the solution. In order to implement the second order scheme click on Solution Methods t hen click on Momentum and select Second Order Upwind as shown in the image below.



https://confluence.cornell.edu/download/attachments/118771076/SecOrder_Full.png

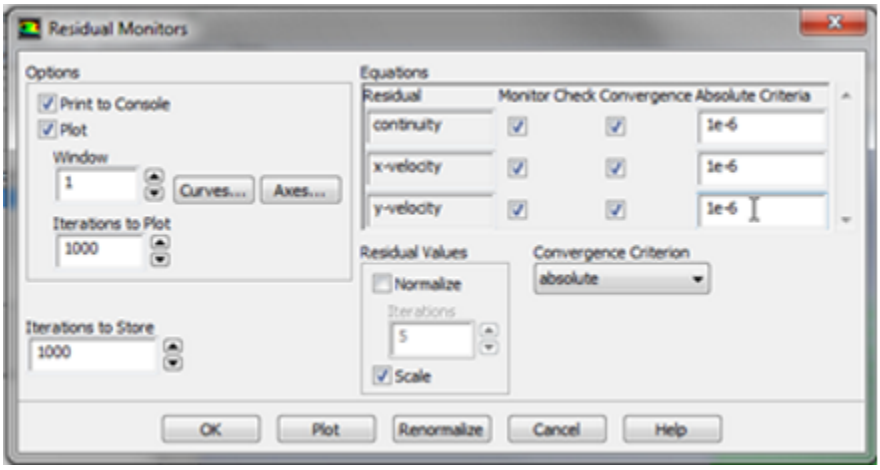
Set Convergence Criteria

FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We'll iterate the solution until the residual for each equation falls below 1e-6. In order to specify the residual criteria (Click) Monitors > Residuals > Edit..., as shown in the image below.



https://confluence.cornell.edu/download/attachments/118771076/EditResid_Full.png

Next, change the residual under Convergence Criterion for continuity, x-velocity,and y-velocity, all to 1e-6, as can be seen below.

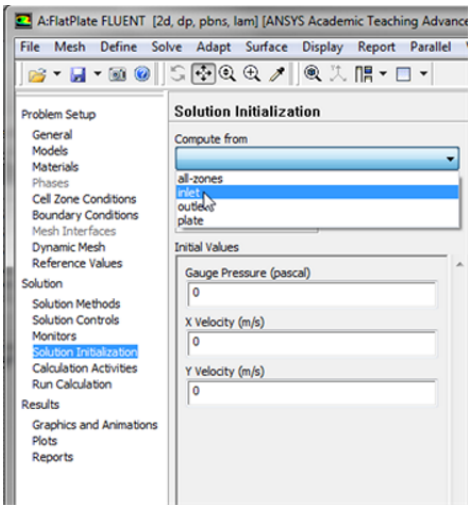


https://confluence.cornell.edu/download/attachments/118771076/ResidMon_Full.png

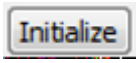
Lastly, click OK to close the Residual Monitors menu.

Set Initial Guess

Here, the flow field will be initialized to the values at the inlet. That is, the initial values of all the cells will be set to 1 m/s and 0 Pa for x velocity and gauge pressure respectively. In order to carry out the initialization click on Solution Initialization then click on Compute from and select inlet as shown below.

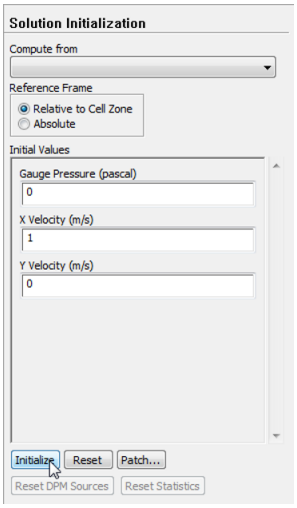


https://confluence.cornell.edu/download/attachments/118771076/CompFromInlet_Full.png

Then, click the Initialize button, 

. This completes the initialization process.

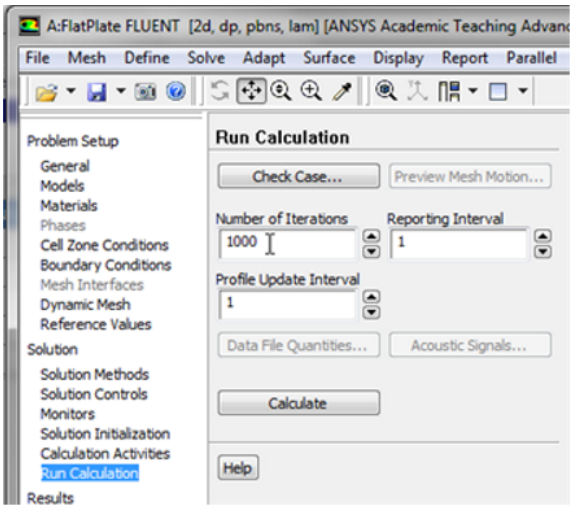
Alternately, you could set the Gauge Pressure to 0 and set the X Velocity to 1 m/s as shown below.



Then, you would need to press the Initialize button to apply the specified initial values to all the cells. Either method will give you the same results.

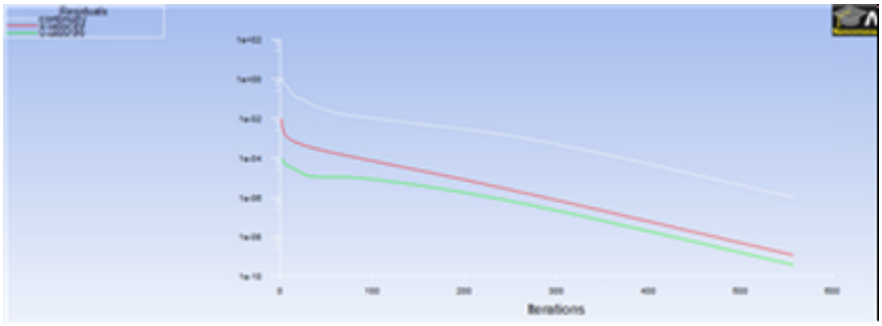
Iterate Until Convergence

Prior, to running the calculation the maximum number of iterations must be set. To specify the maximum number of iterations click on Run Calculation then set the Number of Iterations to 1000, as shown in the image below.



https://confluence.cornell.edu/download/attachments/118771076/1kIter_Full.png

As a safeguard save the project now. Now, click on Calculate two times in order to run the calculation. The residuals for each iteration are printed out as well as plotted in the graphics window as they are calculated. After running the calculation, you should obtain the following residual plot.



https://confluence.cornell.edu/download/attachments/118771076/ResPlot_Full.png

The residuals fall below the specified convergence criterion of 1e-6 in about 557 iterations, as shown below. Actual number of convergence steps may vary slightly.

```
   556 1.0228e-06 1.1556e-09 3.7081e-10  0:00:30  444
!  557 solution is converged
   557 9.9410e-07 1.1281e-09 3.6234e-10  0:00:24  443
```

https://confluence.cornell.edu/download/attachments/118771076/SolConv_Full.png

At this point, save the project once again.

Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules