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

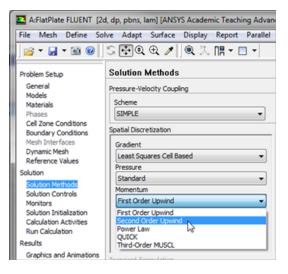
Author: Rajesh Bhaskaran, Cornell University

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.

I A:FlatPlate FLUENT [2d, dp, pbns, lam] [ANSYS Academic Teaching Advanced File Mesh Define Solve Adapt Surface Display Report Parallel V ] @ ▼			
Problem Setup General Motels Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Monitors Residuals, Statistic and Force Monitors Residuals - Print, Plot Statistic - Off Drag - Off Lift - Off Moment - Off Edit. Surface Monitors		
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results	Create Edit Delete Volume Monitors		

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.

Options	Equations				_
Print to Console	Residual	Monitor C	heck Converge	nce Absolute Criteria	
V Plot	continuity	V		1e-6	
1 Curves	x-velocity		V	1e-6	
Iterations to Plot	y-velocity	V	V	1e-6 ]	1.
1000	Residual Values	s Con	vergence Crite	rion	
	Normalize	abs	olute	•	
Iterations to Store	Iterations				
1000	Scale	Ŷ			

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.

	Ive Adapt Surface Display Report Paralle
📸 • 🖬 • 🗟 🔘	중 🚱 🍳 🥒 🔍 🍭 洗 🖷 • 🗖 •
Problem Setup	Solution Initialization
General	Compute from
Models	
Materials	al anna
Phases	al-zones inlet
Cell Zone Conditions	outleas
Boundary Conditions	plate
Mesh Interfaces	Initial Values
Dynamic Mesh Reference Values	Initial values
	Gauge Pressure (pascal)
Solution	0
Solution Methods	
Solution Controls	X Velocity (m/s)
Monitors Solution Initialization	0
Calculation Activities	and a day
Run Calculation	Y Velocity (m/s)
Results	0
Graphics and Animations Plots	
Reports	

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.

Compute from	
	•
Reference Frame	
<ul> <li>Relative to Cell Zone</li> <li>Absolute</li> </ul>	
nitial Values	_
Gauge Pressure (pascal)	^
0	
X Velocity (m/s)	
1	
Y Velocity (m/s)	
0	
1	

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.

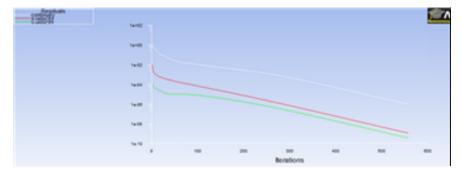
### Iterate Until Convergence

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.

A:FlatPlate FLUENT [2	d, dp, pbns, Iam] [ANSYS Academic Teaching Advanc			
File Mesh Define Solve Adapt Surface Display Report Parallel				
▼ 🗟 🔘   🔄 🔂 🤁 🗶 🥒     ● 🏷 🔚 ▼ 🗖 ▼ 🔜				
Problem Setup	Run Calculation			
General Models	Check Case Preview Mesh Motion			
Materials Phases Cell Zone Conditions	Number of Iterations Reporting Interval			
Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Profile Update Interval			
Solution	Data File Quantities Acoustic Signals			
Solution Methods Solution Controls Monitors Solution Initialization	Calculate			
Calculation Activities Run Calculation	Help			
Results				

https://confluence.cornell.edu/download/attachments/118771076/1klter\_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