# **Sudden Expansion - Numerical Solution**

Authors: Yong Wang & Said Elghobashi, UC Irvine

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

# **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** then click on **Momentum** and select **Second Order Upwind** as shown in the image below.

A:suddenExpansion Flu	ent [axi, dp, pbns, lam] [ANSYS Academic Teaching CF
File Mesh Define So	lve Adapt Surface Display Report Parallel Vie
🕴 🛄 🕴 🚰 🔻 🔛 🔻 🔯	@  \$♀€€ /  € 次     - □ -
Meshing Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Controls Monitors Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Solution Methods Pressure-Velocity Coupling Scheme SIMPLE  Spatial Discretization Gradient Least Squares Cell Based Pressure Second Order Upwind First Order Upwind First Order Upwind First Order Upwind Power Law QUICK Third-Order MUSCL Transient Formulation  Non-Iterative Time Advancement Frozen Flux Formulation Paseudo Transient High Order Term Relaxation Default Report Poor Quality Elements Help

### **Set Initial Guess**

Here, the flow field will be initialized to the values at the inlet. In order to carry out the initialization click on Solution Initialization, Standard Initialization, and click on Compute from and select inlet as shown below.

Then, click the *Initialize* button, Initialize. This completes the initialization.

## 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.

A:suddenExpansion Flue	ent [axi, dp, pbns, lam] [ANSYS Academic Teaching
File Mesh Define So	lve Adapt Surface Display Report Parallel
i 💼 i 📂 🕶 🔛 🔻 🞯	Ø ∰S ⊕ Q € ↗ ∅ Q 次 Ⅲ + □ +
Meshing	Monitors
Mesh Generation	Residuals, Statistic and Force Monitors
Solution Setup	Residuals - Print, Plot
General	Stausuc - Off
Materials	
Phases	
Cell Zone Conditions	
Mesh Interfaces	
Dynamic Mesh	
Reference Values	
Solution	
Solution Methods Solution Controls	
Monitors	
Solution Initialization	Create Edit Delete
Run Calculation	Volume Monitors
Results	
Graphics and Animations	
Plots	
Reports	
	Create Edit Delete
	Convergence Monitors
	Convergence Manager
	Hala

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

Residual Monitors	_				×
Options           Image: Option Print to Console           Image: Option Print to Console	Equations Residual continuity	Monitor C	heck Convergence	Absolute Criteria	*
Window 1 Curves Axes	x-velocity y-velocity		V V	1e-06	
	Residual Values           Normalize         Iterations		Iterations	Convergence Criterion absolute	
Iterations to Store	Scale	al Scale	5		
OK Plot Renormalize Cancel Help					

Lastly, click OK to close the Residual Monitors menu.

#### **Execute Calculation**

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

A:suddenExpansion Flu File Mesh Define Sol	ent [axi, dp, pbns, lam] [ANSYS Academic Teaching Cl Ive Adapt Surface Display Report Parallel Vi
i 📖 i 📂 - 🔛 - 🞯	@  \$₩Q.Q.∥!@,%    -□-
Meshing Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Run Calculation         Check Case       Preview Mesh Motion         Number of Iterations       Reporting Interval         1000       Image: Comparison of the second se

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.



The residuals fall below the specified convergence criterion of 1e-6 in about 473 iterations, as shown below. Actual number of convergence steps may vary slightly.

```
472 7.1711e-07 1.0157e-06 1.2395e-07 0:00:38 528

473 solution is converged

473 7.0347e-07 9.9975e-07 1.2188e-07 0:00:30 527
```

At this point, save the project once again.

Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules