

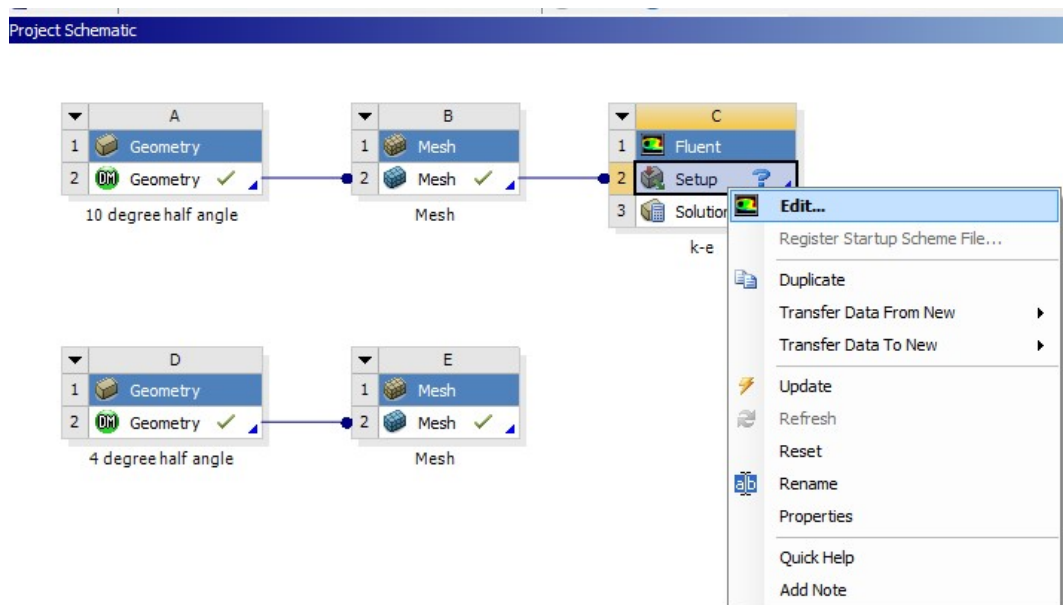
# Simulation of Turbulent Flow in an Asymmetric Diffuser

## 1. Purpose

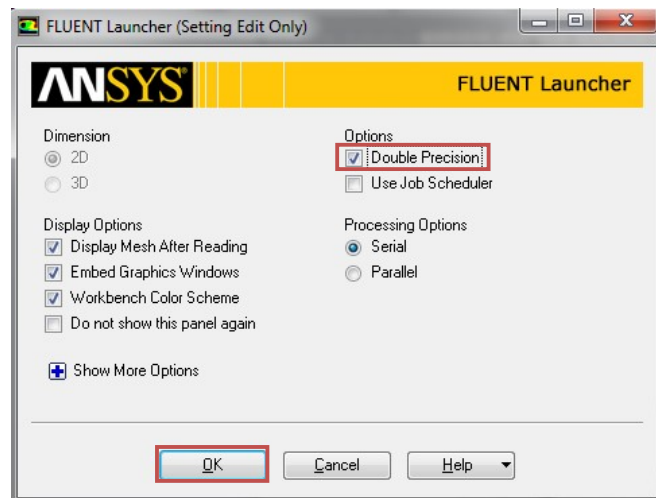
The Purpose of CFD Lab 2 is to simulate **turbulent** flows inside a diffuser following an interactive step-by-step approach and conduct verifications. Students will have “hands-on” experiences using ANSYS to conduct **validation of velocity, turbulent kinetic energy, and skin friction factor. Effect of turbulent models will be investigated, with/without separations.** Students will manually generate meshes, solve the problem and use post-processing tools (contours, velocity vectors, and streamlines) to visualize the flow field. Students will analyze the differences between CFD and EFD and present results in a CFD Lab report.

## 6. Setup

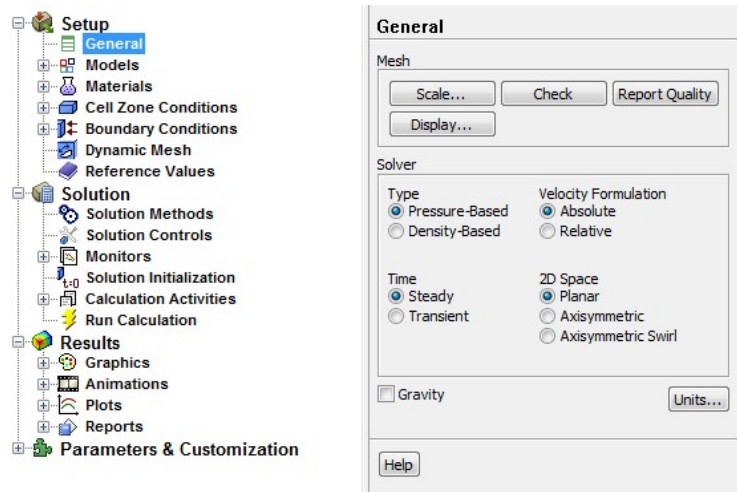
6.1 Right click **Setup** and click **Edit**.



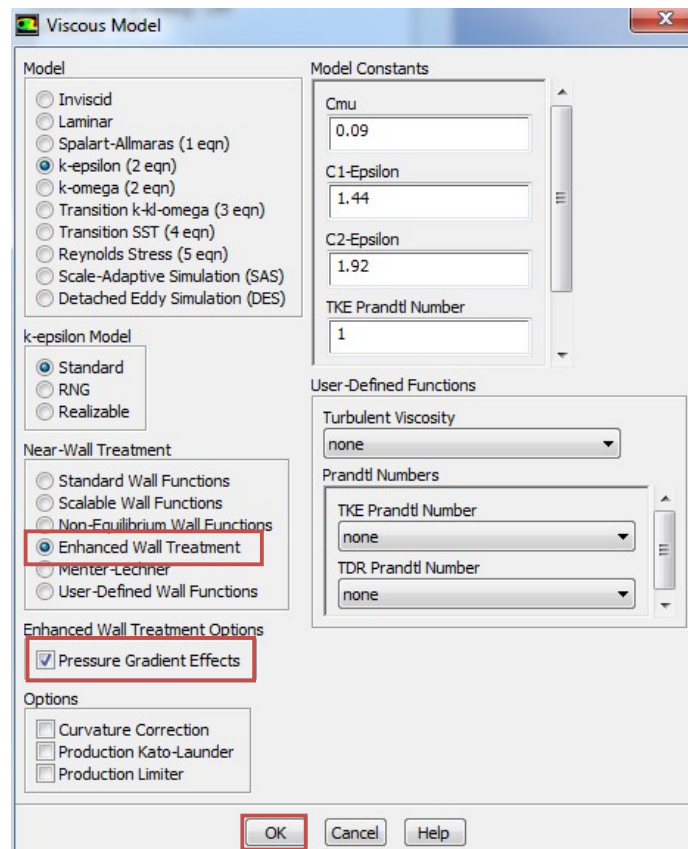
6.2 Check **Double Precision** and select **OK**.



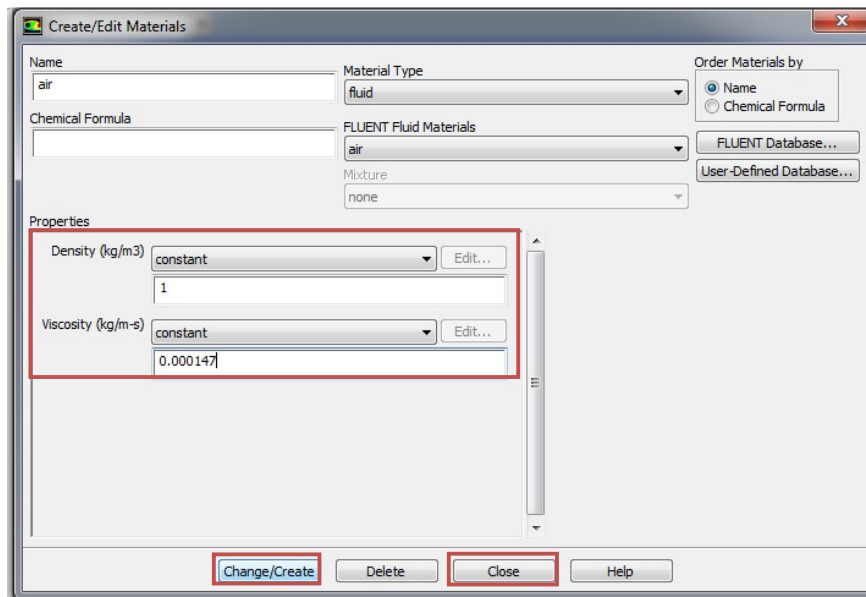
6.3 **Setup > General.** Set the parameters as per below. (Note: You may ignore the warning for the aspect ratio)



6.4 **Setup > Models > Viscous > Edit...** Select parameters as per below and click **OK**.

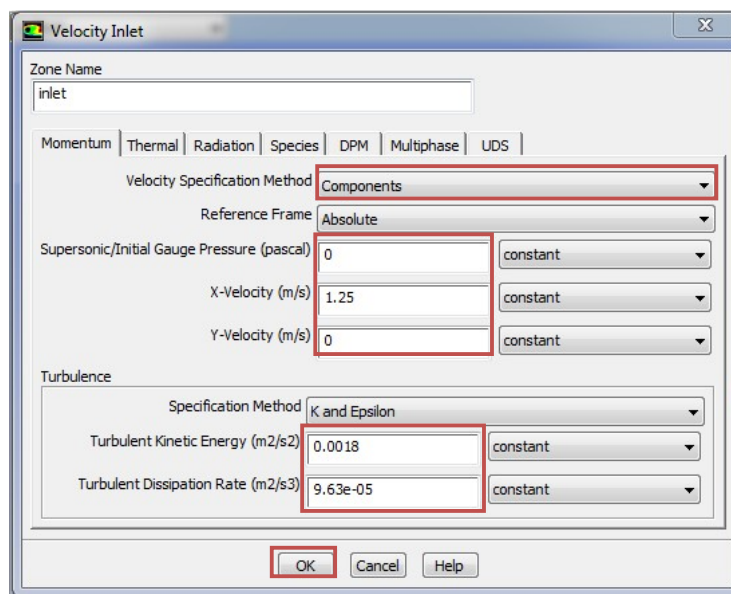


- 6.5 **Setup > Materials > Fluid > air > Create/Edit....** Change the fluid properties and then click **Change/Create** then click **Close**.



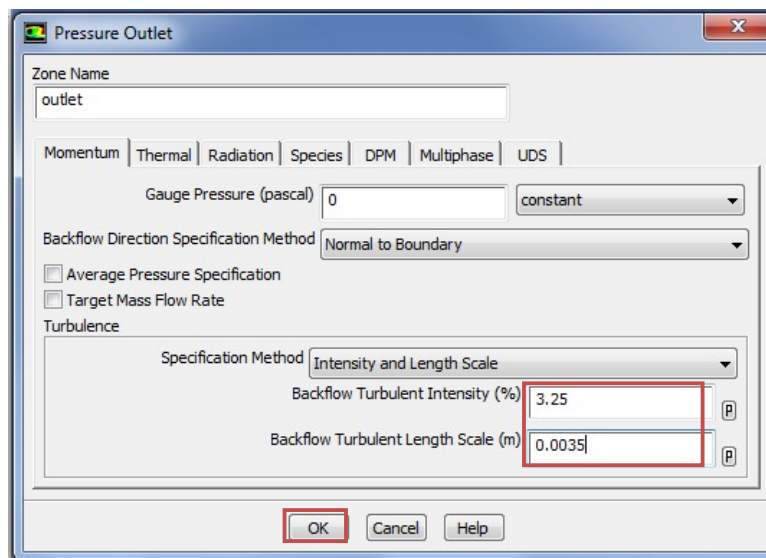
- 6.6 **Setup > Boundary Conditions > Zone > inlet > Edit...** Change parameters for inlet velocity. Use the table below for as per below and click **OK**.

Inlet Boundary Condition					
Variable	u (m/s)	v (m/s)	P (Pa)	k (m <sup>2</sup> /s <sup>2</sup> )	e(m <sup>2</sup> /s <sup>3</sup> )
Magnitude	1.25	0	-	0.0018	9.63e-05
Zero Gradient	-	-	Y	-	-



6.7 **Setup > Boundary Conditions > Zone > outlet > Edit...** Change parameters as per below and click **OK**.

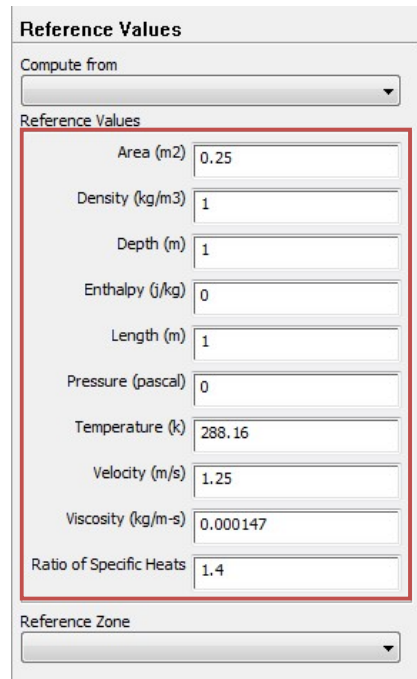
Outlet Boundary Condition					
Variable	u (m/s)	v (m/s)	P (Pa)	Intensity (%)	Length scale (m)
Magnitude	-	-	0	3.25	0.0035
Zero Gradient	Y	Y	-	-	-



6.8 Make sure boundary condition type is wall for top and bottom walls.

Airfoil Boundary Condition					
Variable	u (m/s)	v (m/s)	P (Pa)	k ( $\text{m}^2/\text{s}^2$ )	e ( $\text{m}^2/\text{s}^3$ )
Magnitude	0	0	-	0	0
Zero Gradient	-	-	Y	-	-

6.9 **Setup > Reference Values.** Change reference values as per below.



**Reference Values**

Compute from:

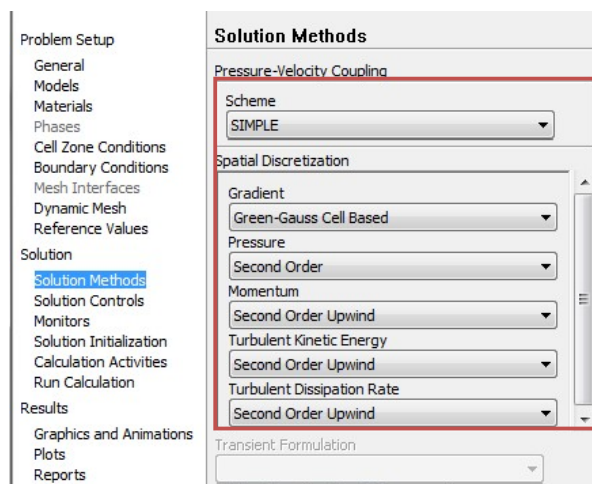
**Reference Values**

Area (m <sup>2</sup> )	0.25
Density (kg/m <sup>3</sup> )	1
Depth (m)	1
Enthalpy (J/kg)	0
Length (m)	1
Pressure (pascal)	0
Temperature (K)	288.16
Velocity (m/s)	1.25
Viscosity (kg/m·s)	0.000147
Ratio of Specific Heats	1.4

Reference Zone:

## 7. Solve

7.1 **Solution > Solution Methods.** Change the solution methods as per below.



**Solution Methods**

Pressure-Velocity Coupling

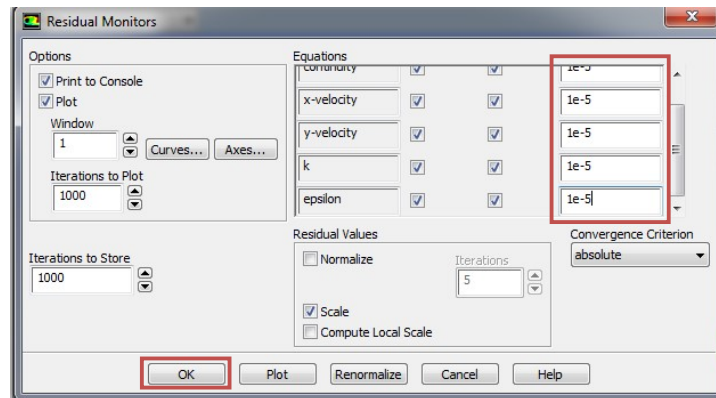
Scheme:

Spatial Discretization

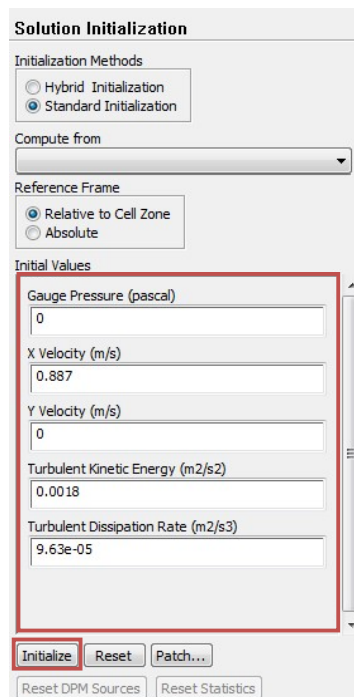
Gradient	Green-Gauss Cell Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Turbulent Dissipation Rate	Second Order Upwind

Transient Formulation:

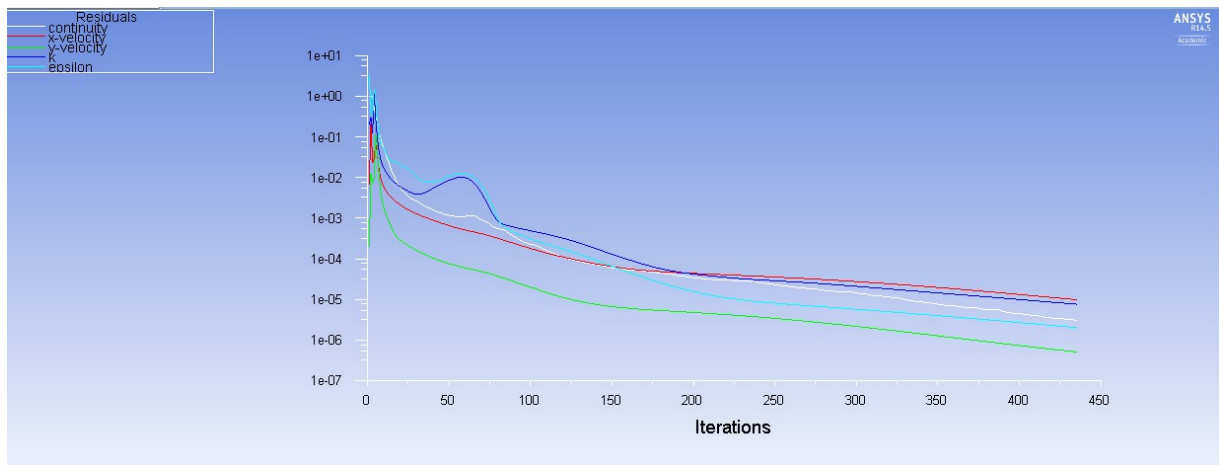
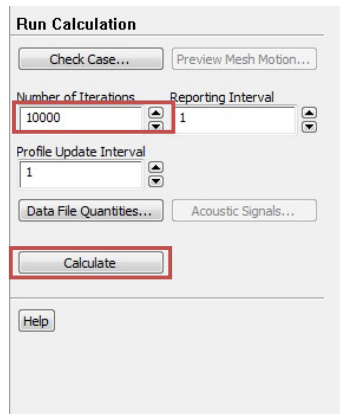
- 7.2 **Solution > Monitors > Residuals - Print, Plot > Edit...** Change convergence criteria to **1e-05** and click **OK**.



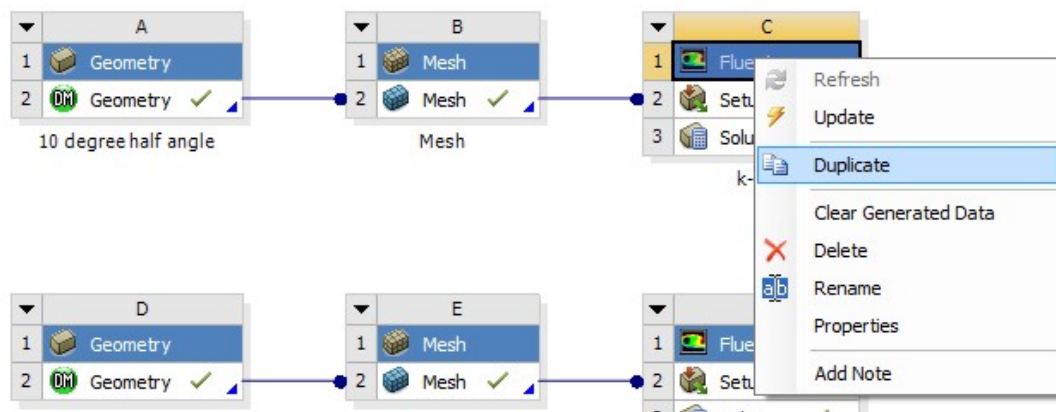
- 7.3 **Solution > Solution Initialization.** Change parameters as per below and click **Initialize**.



- 7.4 **Solution > Run Calculation.** Change **Number of Iterations** to **10,000** and click **Calculate**.

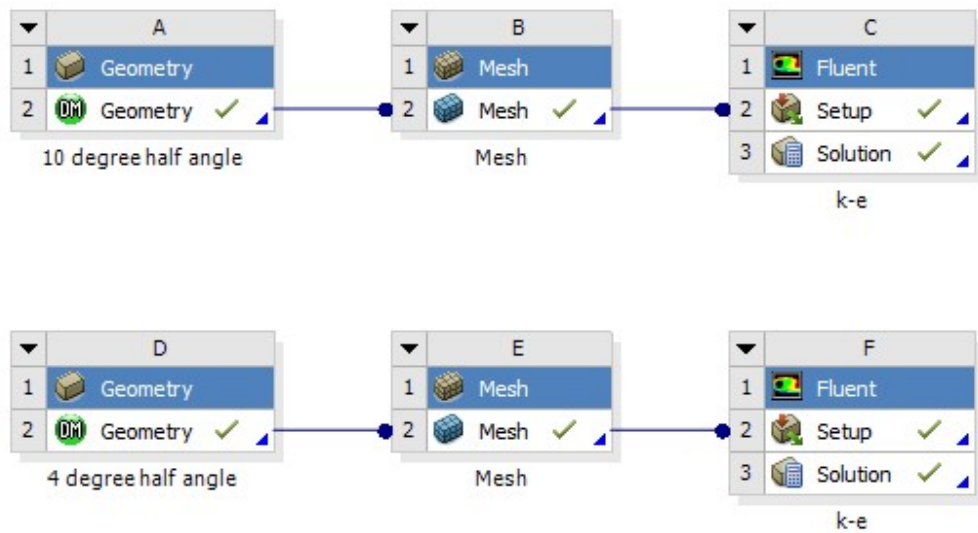


- 7.5 Save your project and quit ANSYS fluent.
- 7.6 Duplicate the k-e setup for 10 degree half angle case to 4 degree angle case as per below then run the case.



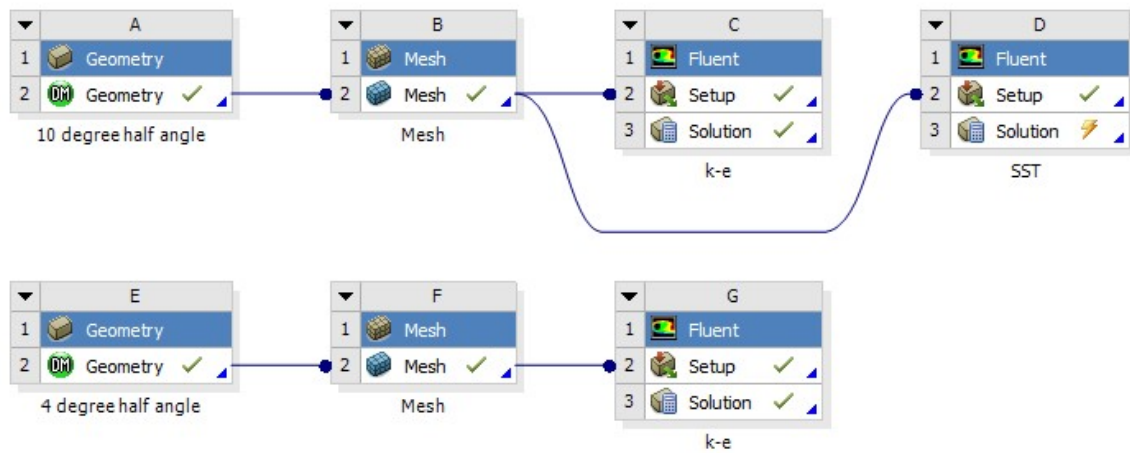


## Project Schematic

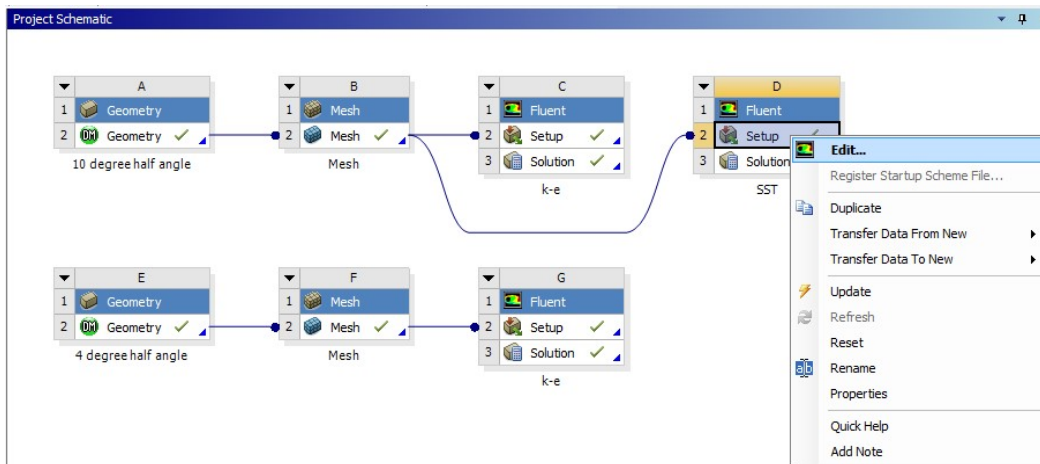


## 7.7 Duplicate the k-e setup for 10 degree half angle and rename it SST

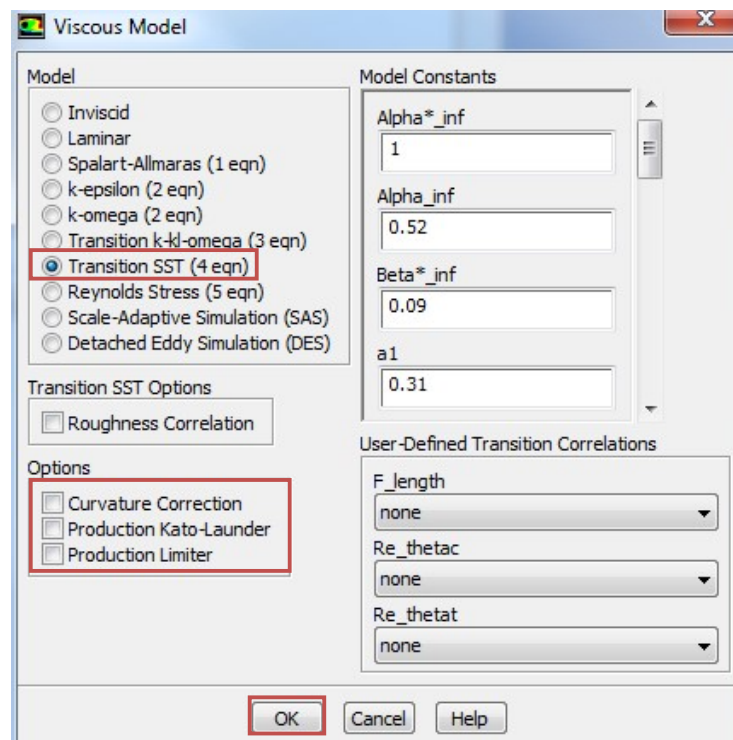
## Project Schematic



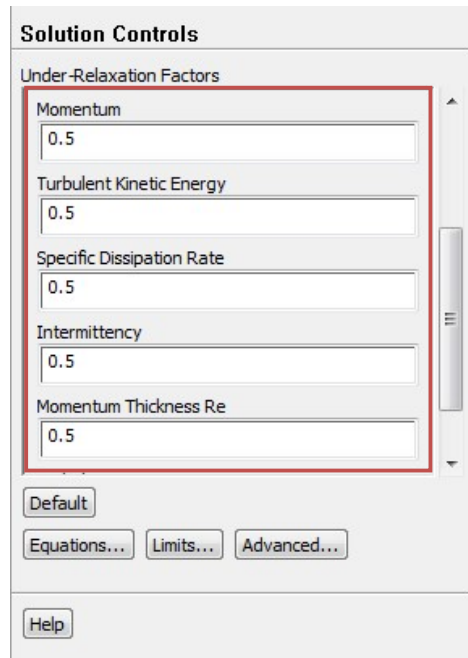
### 7.8 Right click and select **Edit...**



### 7.9 **Setup > Models > Viscous > Edit.** Select SST model and use the default parameters as per below then click ok.



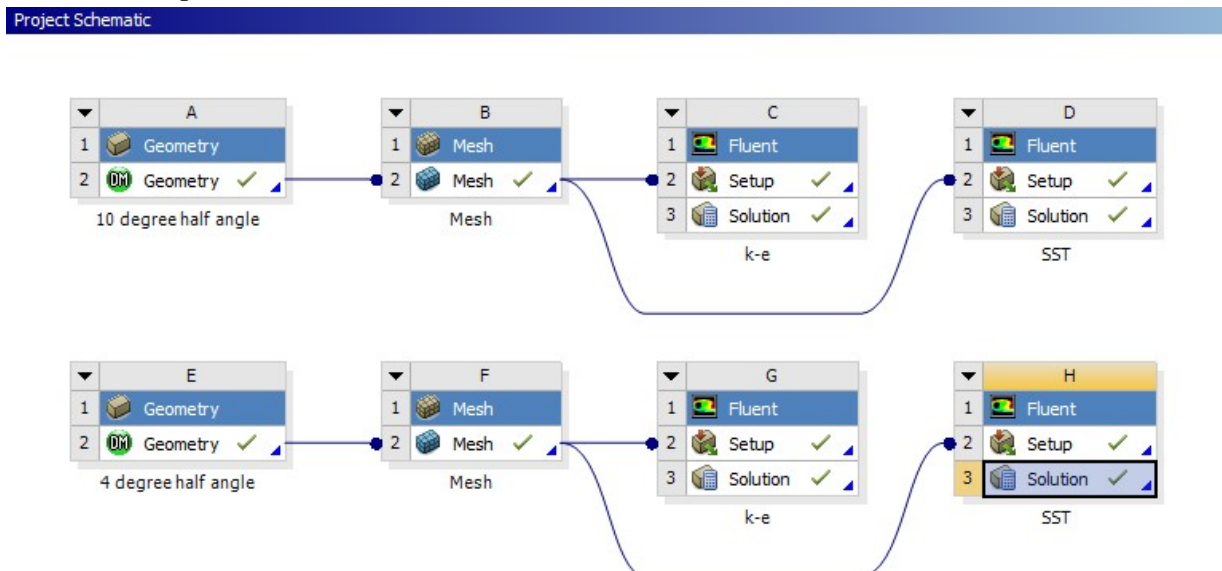
7.10 **Solution > Solution Controls.** Change **Under-Relaxation Factors** as per below.



7.11 **Solution > Solution Initialization > Initialize.**

7.12 **Solution > Run Calculation > Calculate.**

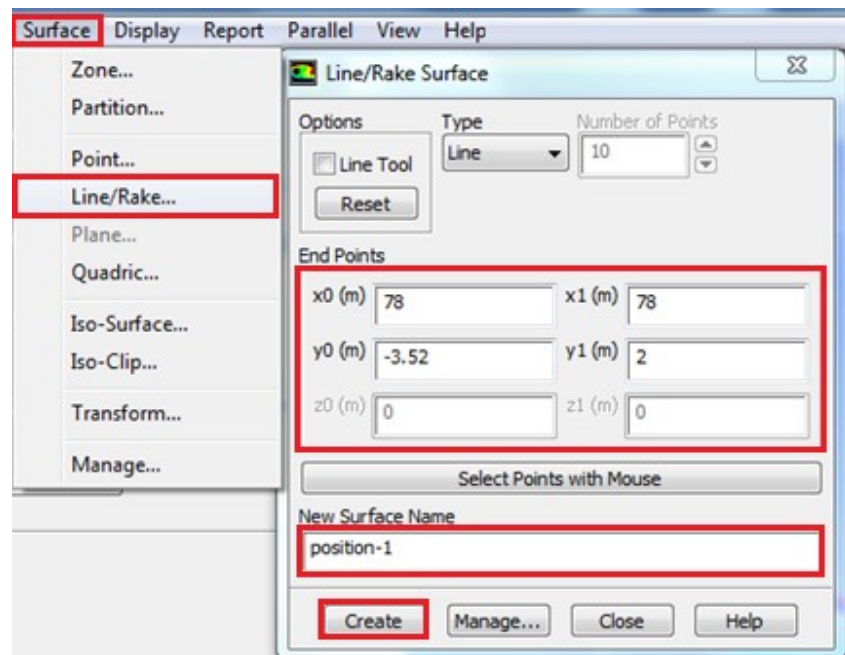
7.13 Duplicate SST fluent setup for the 4 degree half angle case and run the simulation as per below.



## 8. Results

8.1 Create lines for plotting modified TKE and modified U.

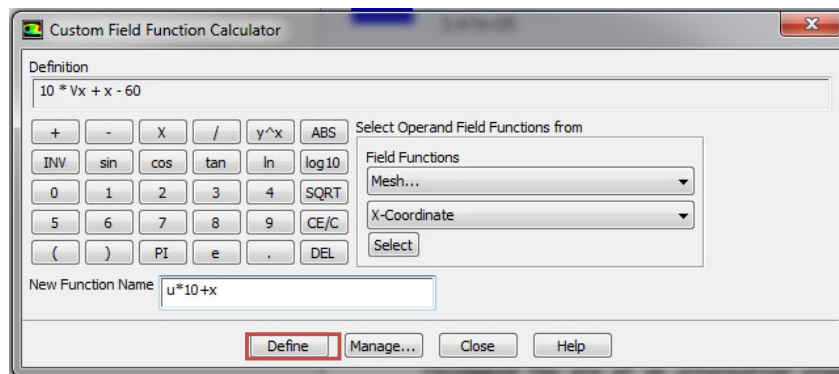
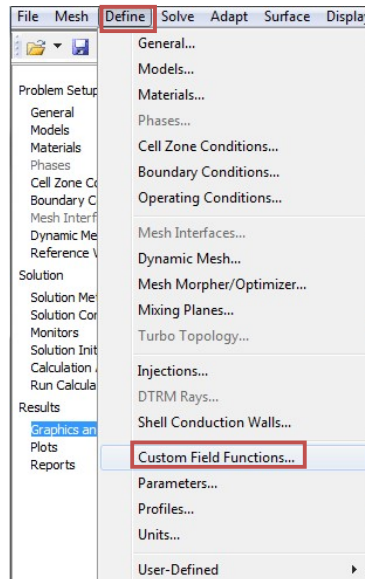
**Surface > Line/Rake.** Create 7 Lines at the given location on the table.



Surface Name	x0	y0	x1	y1
Position-1	78	-3.52	78	2
Position-2	82	-4.23	82	2
Position-3	86	-4.9371	86	2
Position-4	98	-7.053	98	2
Position-5	102	-7.4	102	2
Position-6	110	-7.4	110	2
Position-7	118.5	-7.4	118.5	2

## 8.2 Define custom field functions for modified velocity, modified TKE and skin friction coefficient.

**Define > Custom Field Function.** Write the equation shown below and click **Define**. You will need to use the Field function and the buttons to enter the parameters. Definitions of the variables and custom field function that need to be defined are shown on table below.

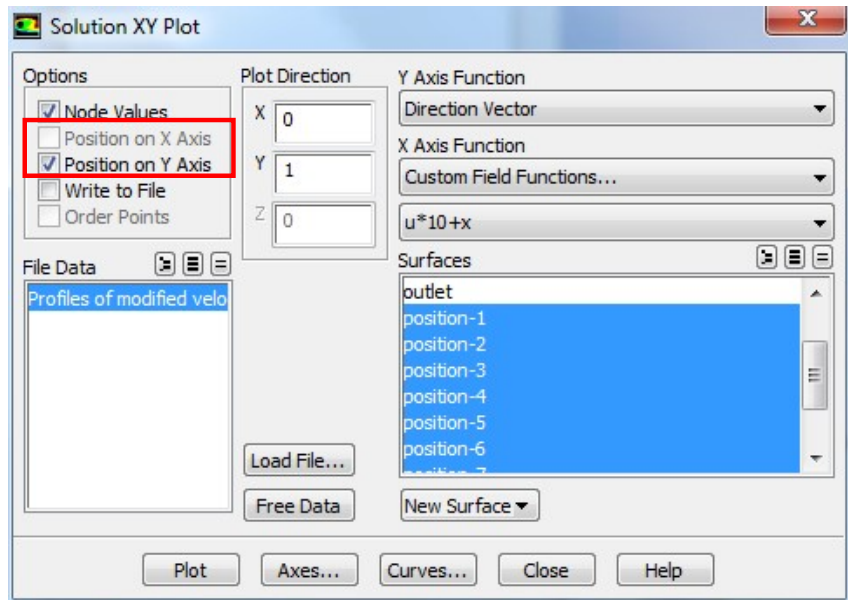


Function Name	Definition
u*10+x (Modified U)	10*V <sub>x</sub> +x-60
k*500+x (Modified TKE)	500*turb-kinetic-energy+x-60
skinfriction-coefficient	x-wall-shear * 2 / density / 1.25 ^ 2

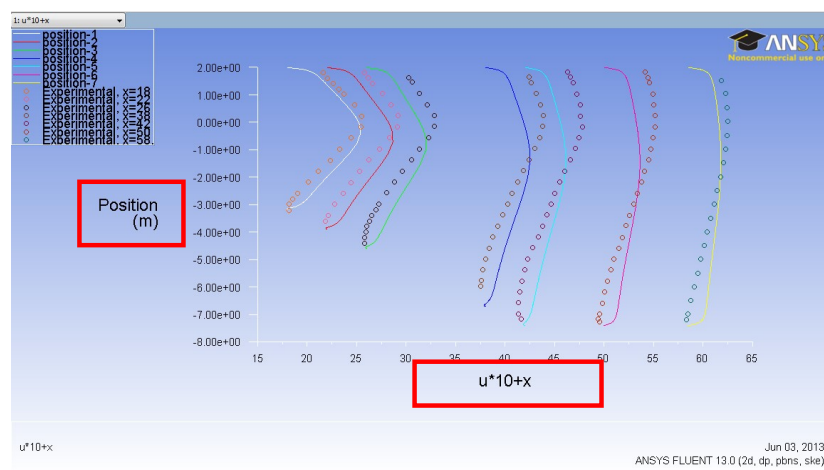
You can find the functions you created under XY plot and create plots.

Note : Make sure about options as shown below.

Turn off “Position on X Axis” and Turn on “Position on Y Axis”



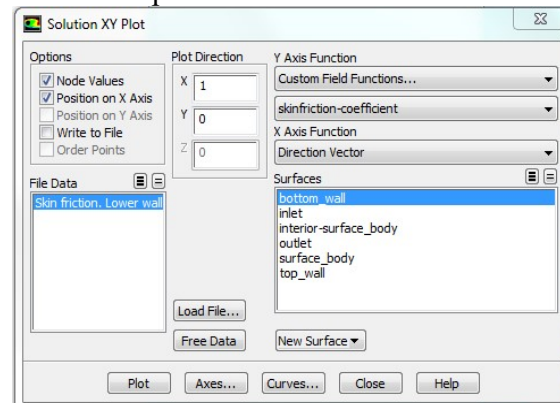
You can compare EFD and CFD using the custom field functions on the lines you created as per below. Be careful about the axis location as shown below



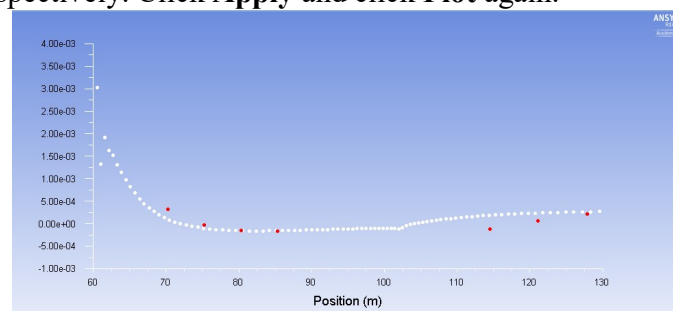
### 8.3 Plotting skin friction

**Results > Plots > XY Plot > Set Up... > Load File...** Select the Skin Friction bot wall.xy file from the class website and click **OK**.

Change the parameters as per below and click **Plot**.



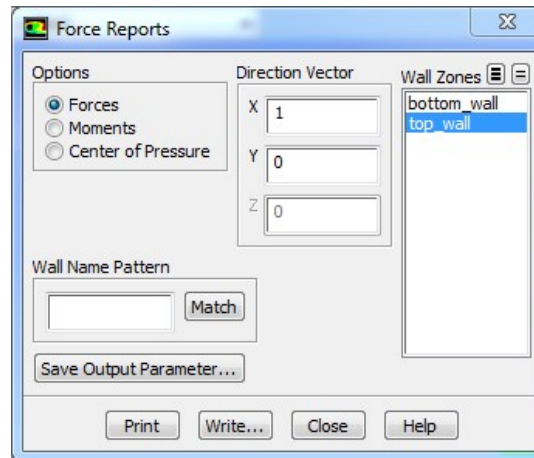
You can change the axis by clicking **Curves** under XY plot. Change the x-axis min and max to **60** and **130** respectively. Change the y-axis max and min to **4e-03** and **-1e-03** respectively. Click **Apply** and click **Plot** again.



### 8.4 Total friction

**Results > Reports > Forces > Set Up....** Select the zone where you want to calculate the total force then select print. This will print a report as per below

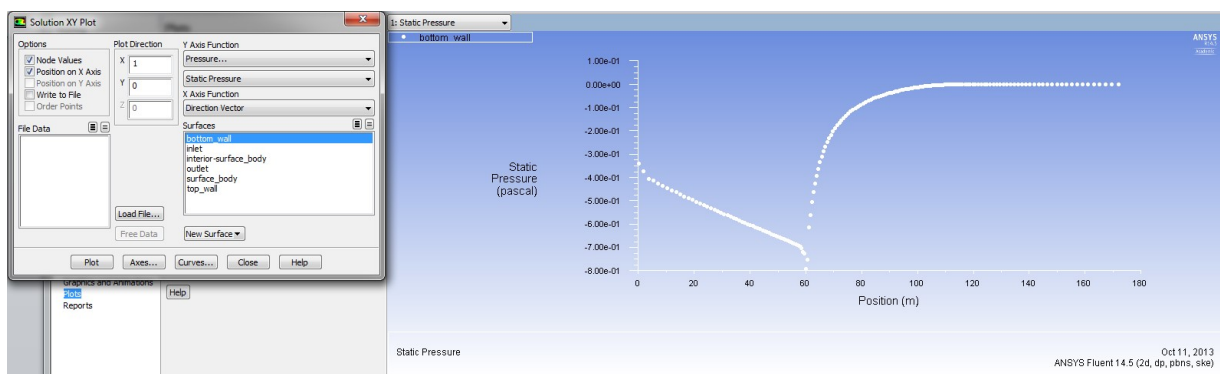




Forces						
Zone	Forces (n)			Viscous		Total
top_wall	Pressure (0 -38.91522 0)			(0.35687363 0 0)		(0.35687363 -38.91522 0)
Net	(0 -38.91522 0)			(0.35687363 0 0)		(0.35687363 -38.91522 0)
Forces - Direction Vector (1 0 0)						
Zone	Forces (n)			Coefficients		
top_wall	Pressure 0	Viscous 0.35687363	Total 0.35687363	Pressure 0	Viscous 1.827193	Total 1.827193
Net	0	0.35687363	0.35687363	0	1.827193	1.827193

### 8.5 Finding the pressure difference between inlet and outlet.

You can simply write pressure at bottom wall to a file and take the difference of pressure at inlet and outlet.



### 8.6 Plotting contours, velocity vectors and streamlines.

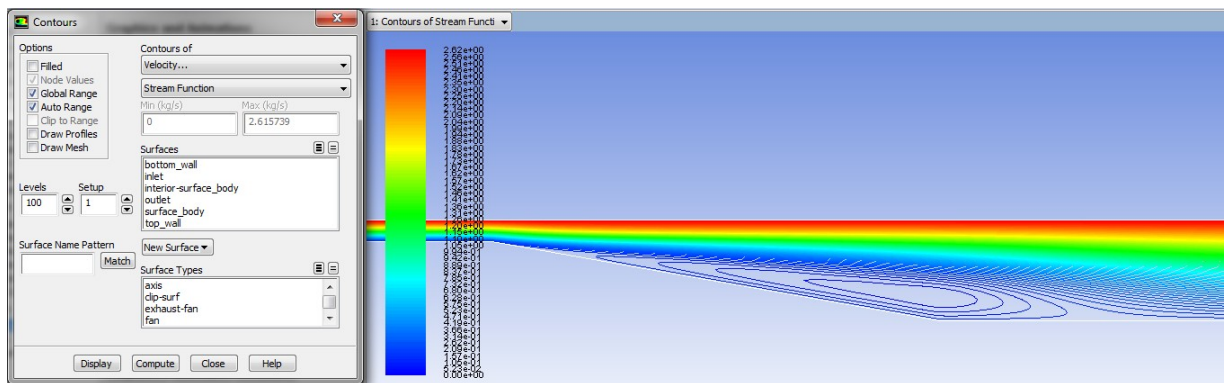
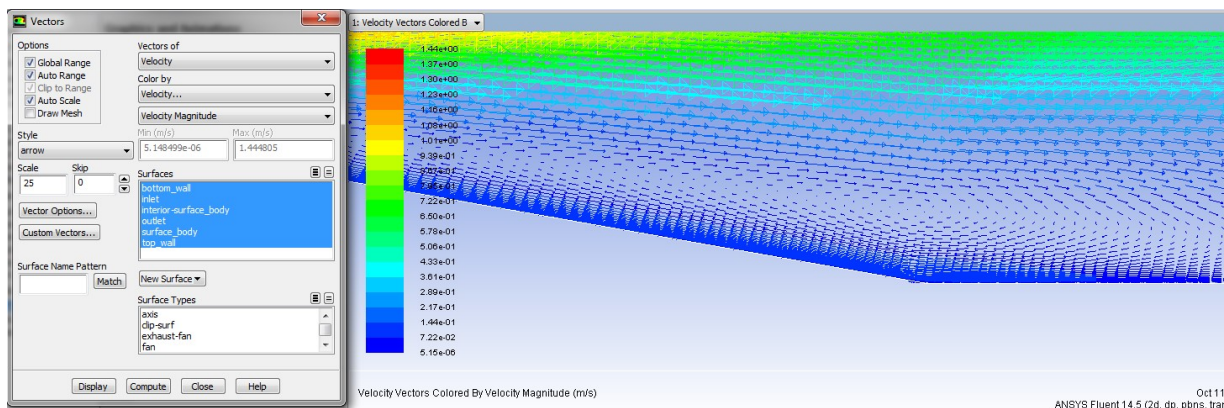
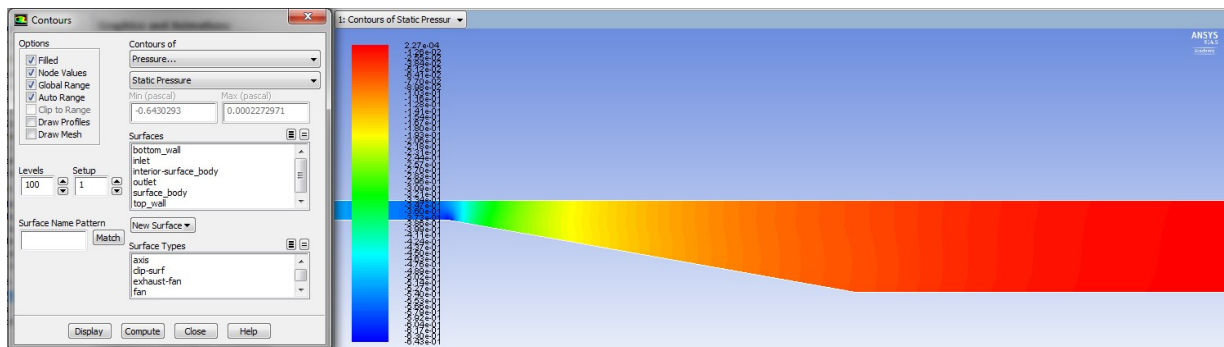
Plot streams, velocity vectors and pressure distributions.

**Results > Graphics and Animations > Graphics > Contours > Setup.** Click **Display**

**Results > Graphics and Animations > Graphics > Vectors > Set Up.** Click **Display**

You can change the scales and levels for vectors and streamlines respectively to show the separation region. Few examples are shown below.





## 9. Exercises

You need to complete the following assignments and present results in your lab reports following the lab report instructions.

### Simulation of Turbulent Flow in an Asymmetric Diffuser

#### 1. Simulation of turbulent diffuser flows without separation (4 degree):

1.1. Run simulations for 4 degree half angle diffuser with k- $\epsilon$  model.

1.2. Run simulations for 4 degree half angle diffuser with SST model.

1.3. Questions:

- Do you observe separations in 1.1 or 1.2? (use streamlines)
- What are the differences between 1.1 and 1.2 regarding modified u, modified TKE, and the variables in the following table?

Turbulent model	Total pressure difference between inlet and outlet (Pa)	Total friction force on the upper wall (N)
SST		
k-e		
Relative error (%)		

- **Figures to be saved** (for both 1.1 and 1.2): 1. XY plots for residual history, 2. Modified u vs. x, 3. Modified TKE vs. x, 4. Contours of pressure 5. Contours of axial velocity, 6. Velocity vectors and streamlines
- **Data to be saved:** the above table with values.

#### 2. Simulation of turbulent diffuser flows with separation (10 degree):

2.1. Run simulations for 10 degree half angle diffuser with k- $\epsilon$  model.

2.2. Run simulations for 10 degree half angle diffuser with SST model.

2.3. Questions:

- Do you observe separations in 2 or 2.1? (using streamlines)
- Comparing with EFD data, what are the differences between 2.1 and 2.2 on the following aspects: (1). Modified velocity, (2). Modified TKE, (3). Skin friction factor on top and bottom walls, (4). Variables in the following table.

Turbulent models	Total pressure difference between inlet and outlet (Pa)	Total friction force on the upper wall (N)
SST		
k-e		
Relative error (%)		

- If any separation shown, where is the separation point on the diffuser bottom wall ( $x=?$ ) and where does the flow reattach to the diffuser bottom wall again ( $x=?$ ) (hint: use wall friction factor)
- Do you find any separation on the top wall?
- **Figures to be saved** (for both 2.1 and 2.2): 1. Residual history, 2. Modified  $u$  vs.  $x$  with EFD data, 3. Modified TKE vs.  $x$  with EFD data, 4. Skin friction factor distributions on top and bottom walls with EFD data, 5. Contour of pressure, 6. Contour of axial velocity, 7. Velocity vectors and streamlines with appropriate scales showing the separation region if the simulation shows separated flows.
- **Data to be saved:** The above table with values.

### 3. Questions need to be answered in CFD Lab3 report:

- 1.2. By analyzing the results from exercise 1 and exercise 2, what can be concluded about the capability of  $k-\epsilon$  and SST models to simulate turbulent flows inside a diffuser with and without separations?