MAE – 571 APPLICATIONS OF CFD

SPRING 2018

FINAL PROJECT REPORT

CFD Simulation of Flow around ONERA M6 Wing

By

Romesh Satish Prasad

678204795

Background-

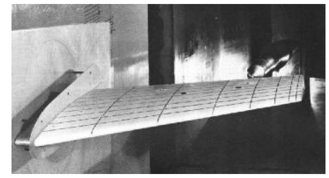


Figure 1: M6 Wing

This report talks about the Onera M6 wing which was designed by the ONERA Aerodynamics Department. The wing was well instrumented and tested in a wind tunnel at various transonic Mach Numbers for angle-of-attack range up to 6 degrees. The Reynolds numbers were about 12 million based on the mean aerodynamic chord. The wind tunnel tests are documented by Schmitt and Charpin in the AGARD Report AR-138 [1]. Because of its simple geometry combined with complexities of transonic flow (i.e. local supersonic flow, shocks, and turbulent boundary layers separation), the wing became a classic CFD validation case for external flows.

Objectives-

For this project I am simulating flow around a scaled-down version of M6 wing using Computational Fluid Dynamics. I am going to compare computational results with the experimental data available from NASA simulation. The NASA simulation results are for pressure coefficients at z/b locations of 0.2, 0.44 and 0.65 along the wing and also the simulation results for lift and drag coefficients.

1- Mesh Independence Study- I will be making three kind of mesh i.e. coarse mesh (300-400K), middle mesh (700-800K) and a refined mesh (1.2 million and above) for any one turbulence model of my choice (which in my case is Spalart Allmaras).I will be calculating Drag and lift for whole wing for all the above case mentioned and comparing them with the NASA results. Additionally I am going to do static pressure contour, Mach number contour for different z/b locations. Along with this I am going to plot coefficient of pressure for different z/b locations and comparing them with the NASA simulation results.
2- Turbulence Model Sensitivity Study- I will be investigating the effects of three different turbulence model (in my case it is Spalart-Allmaras, k-epsilon, k-omega) and running simulation for the finest mesh from 1. I will be comparing the lift and drag for different turbulence model and comparing them with the NASA simulation results. To understand which model agrees better with the NASA data. Again I will be plotting contours of Mach number, Static pressure along different z/b locations in the plane so that we can find if there is any shock wave. Plotting pressure coefficient for different locations z/b along the plane and comparing it with the NASA simulation results.
3- For all the above mentioned case there will also be a plotting of y+ and also calculation of average y+ for each turbulence model.

Geometry:-

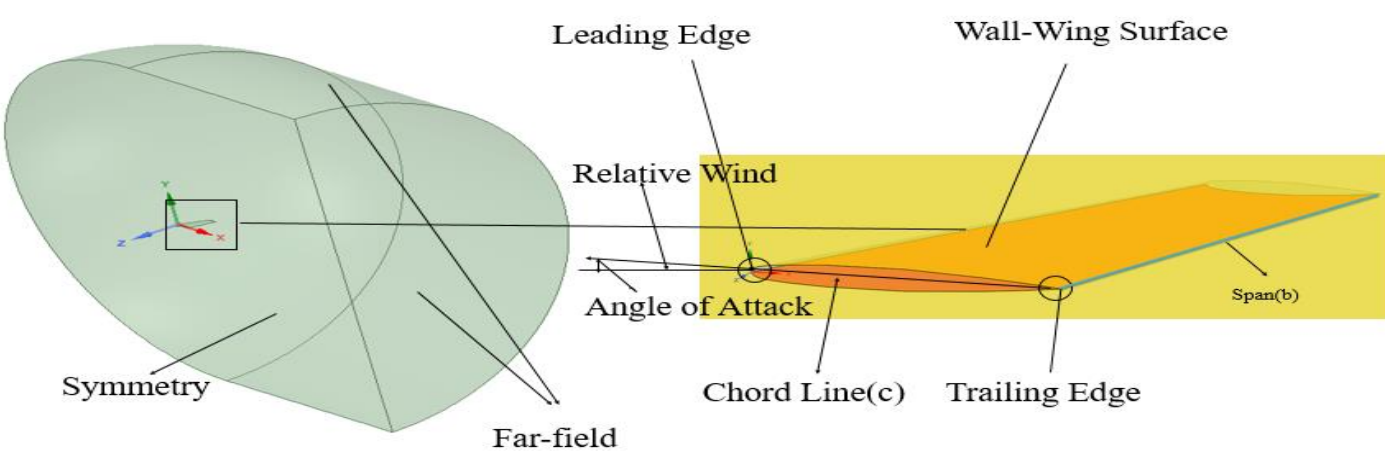


Figure 2: Geometry of M6 wing in Flow-field

Boundary Conditions:-

Zone	Type
Symmetry	Symmetry
Far-field	Pressure-far-field
Wall-wing-surfaces	wall
Wall-wing-tips	wall

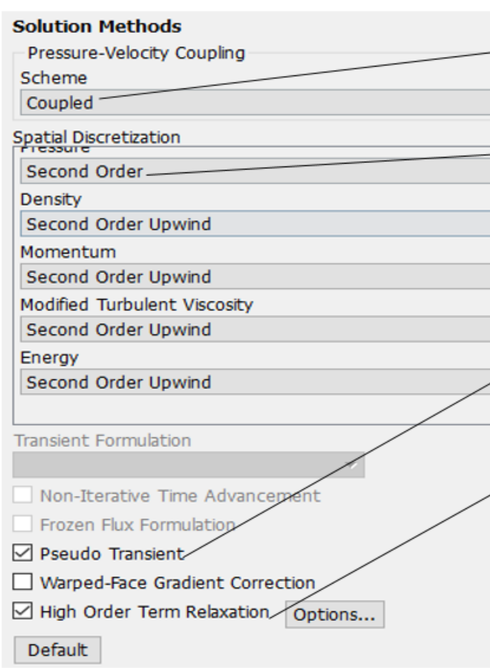
Table1: Zone and type of conditions assigned

Flow properties and reference values:-

Mach Number	0.84
Static pressure (psi)	45.8290
Total temperature (R)	460
Dynamic viscosity (lbm/ft-s)	1.09329e-5
Angle of attack	3.06
Reynolds Number (based on mean aerodynamic chord)	11.72e+6
Reference area (square feet)	1.076
Reference length (ft)	1
Reference density (lbm/ft3)	0.2689053

Table-2- Flow properties and reference values

Solver Setting



For many general fluid-flow problems, convergence speed can be improved by using the Coupled solver

Second-order discretization provides optimum accuracy.

The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior.

The relaxation of high order terms will help to improve the solution behavior of flow simulations when higher order spatial discretization's are used (higher than first).

Figure 3- Details of solver used and explanation

Mesh Dependency Study

Scope	Element size			Behavior			Sphere Center			Sphere Radius			Element Size			Inflation Transition			Max-Layers		
	C	M	F	C	M	F	C	M	F	C	M	F	C	M	F	C	M	F	C	M	F
Fairfield	2.2	1.5	1	H	H	H	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-
Wall Wing Surface	0.01	0.009	0.009	H	H	H	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-
Body	-	-	-	-	-	-	CS	CS	CS	2	2	2	0.20	0.15	0.1	-	-	-	-	-	-
Body	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	S	S	S	5	10	12

Table-3- Details of all the parameters used for meshing. Units are feet.

Here
C-Coarse Mesh.
M- Medium Mesh
F- Fine Mesh
H-Hard
CS- Co-ordinate system X=0.337, Y-0, Z=-0.5

Coarse Mesh – (469488 elements)

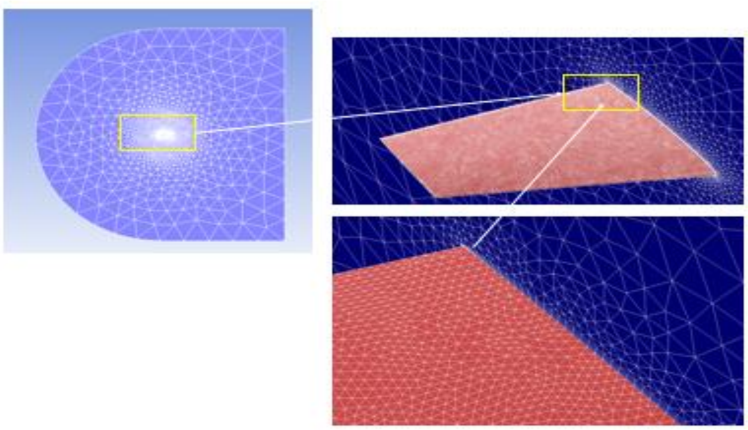


Figure 4- Coarse Mesh for M6 wing

The maximum skewness of our mesh is coming out to be 0.97 which is less than 1 making it equally distributed.

Turbulence Model – Spalart Allmaras
It is a one equation turbulence model that has been developed specifically for aerodynamic flows.
It is based on kinematic eddy viscosity and mixing length
Since it deals with only one equation it very fast for non-linear converging.
After running the simulation for 100 iterations we see that our lift and drag converges around 60 iterations.

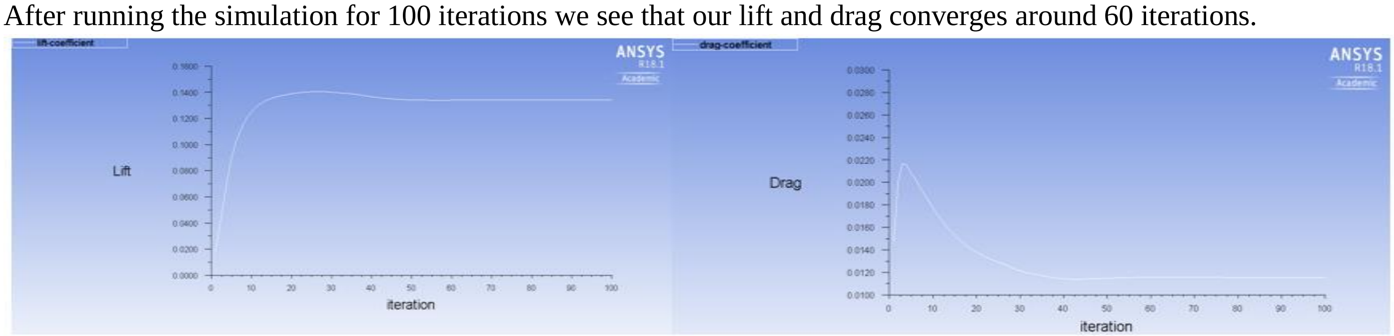


Figure 5- Convergence for lift and drag

Location	0.2	0.44	0.65
Average Coefficient of pressure	-1.872E-07	-7.56949e-006	-1.29486e-005

Table4- Average Coefficient of Pressure for Z/B locations on Coarse

This value suggests that as you move away from the trailing edge the coefficient of pressure decreases.

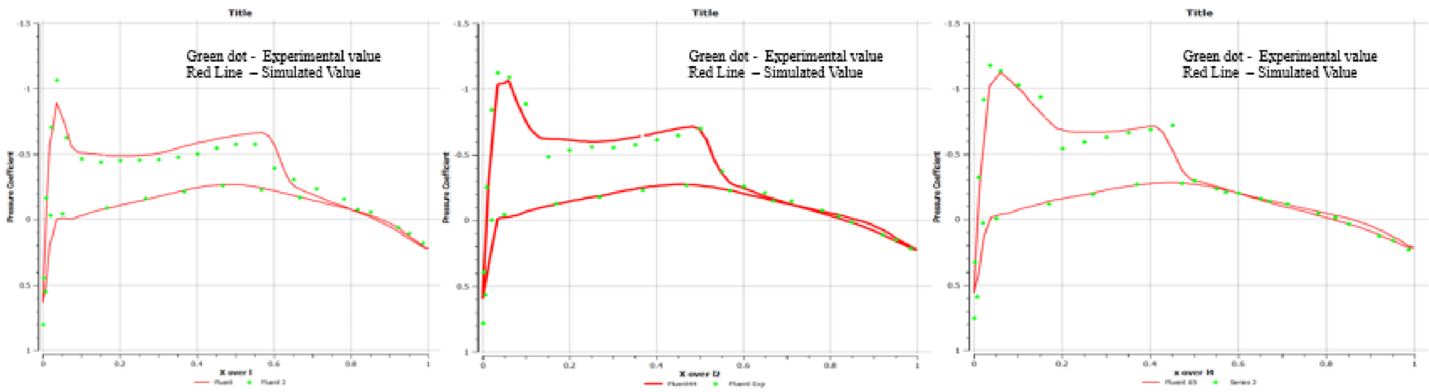


Figure 6:- Pressure Coefficient vs X/C for different Z/b locations 0.2, 0.44, 0.65 from left to right for Coarse Mesh

Medium Mesh – (714349 elements)

Using the condition mentioned in the Table 3 the mesh is further refined. The skewness for the medium mesh is 0.96 which again make it uniformly distributed along the domain

.

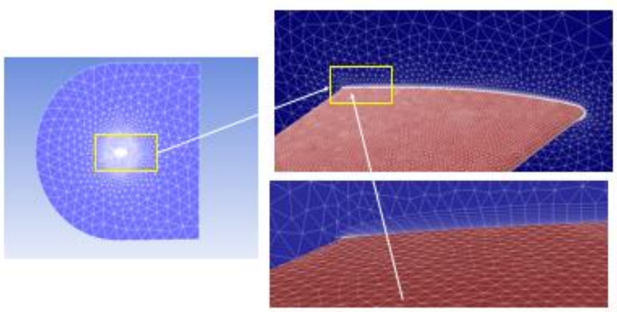


Figure 7: Mesh for the M6 Wing

Location	0.2	0.44	0.65
Average Coefficient of pressure	2.44881e-005	1.3715e-005	-7.97054e-007

Table 5: Average Coefficient of Pressure for Z/B locations on Medium Mesh

From Table 5 we can infer that as the flow goes away from the wing surface the average pressure coefficient decreases.

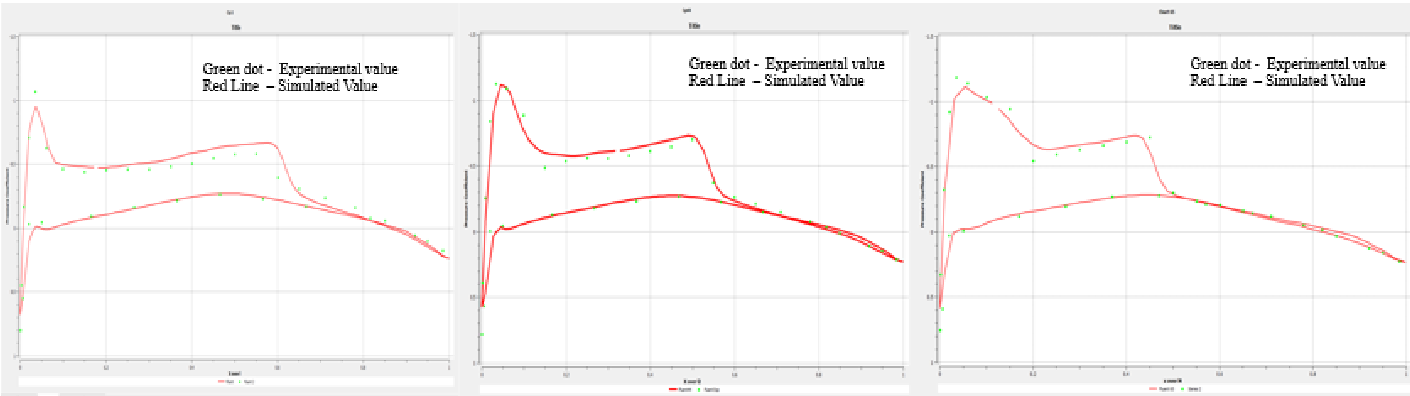


Figure 8:- Pressure Coefficient vs X/C for different Z/b locations 0.2, 0.44, 0.65 from left to right for Medium Mesh

Fine Mesh – 2.2 Million elements

Using the condition mentioned in the Table 2 the mesh is further refined. The skewness for the fine mesh is 0.99 which again make it uniformly distributed along the domain.

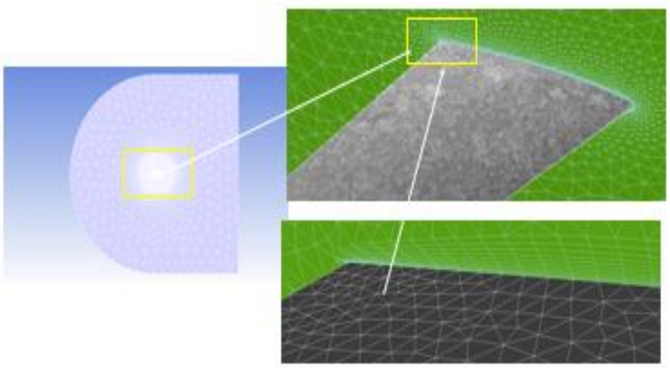


Figure 9: Fine Mesh for the M6 Wing

Location	0.2	0.44	0.65
Average Coefficient of pressure	-3.95597e-007	-1.26e-06	-3.08721e-005

Table 6: Average Coefficient of Pressure for Z/B locations on Fine Mesh

From Table 6 we can infer that as the flow goes away from the wing surface the average pressure coefficient decreases. Also the mesh look well refined near the airfoil making the simulation more accurate. It took approximately 16 minutes to converge the results for 2.2 million elements.

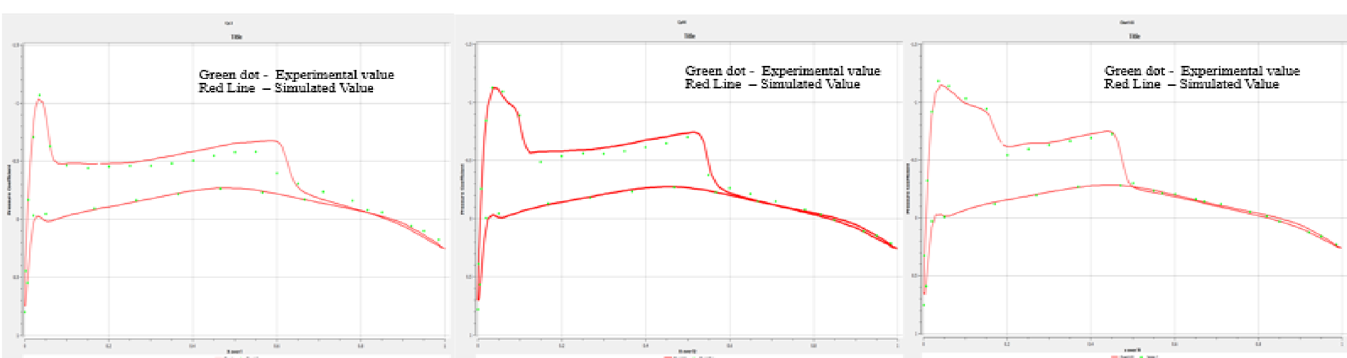


Figure 10:- Pressure Coefficient vs X/C for different Z/b locations 0.2, 0.44, 0.65 from left to right for Fine Mesh

Results for mesh independence study

Drag and Lift Coefficient:

	Drag	Lift	Error Drag	Error Lift
NASA	0.0088	0.141	0	0
Coarse Mesh	0.011538	1.34E-01	31.11	4.858
Medium Mesh	0.010911	0.13688	23.98864	2.9219
Fine Mesh	0.010282	0.14097	16.84091	0.021277

Table 7: Coefficient of Lift and Drag for Coarse, Medium and Fine Meshes

For Fine Mesh, Coefficient of lift has an error of 0.02% and Drag has an error of 16.84%. Which is very less when compared to coarse mesh and medium mesh. As element size increases error and drag percentage decreases.

Pressure Coefficients:

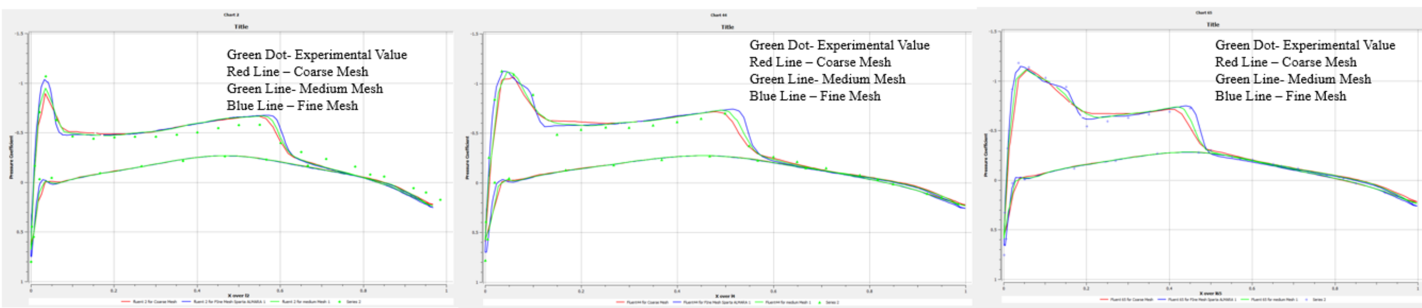


Figure 11- Pressure Coefficient vs X/C for different Z/b locations 0.2, 0.44, 0.65 from left to right for Fine Mesh

From figure 11 we can see that blue line which is fine mesh agrees with the experimental value which is shown by dotted line, most often than not compared to the Coarse mesh and Medium mesh.

Y+ on the Airfoil Surface for different meshes

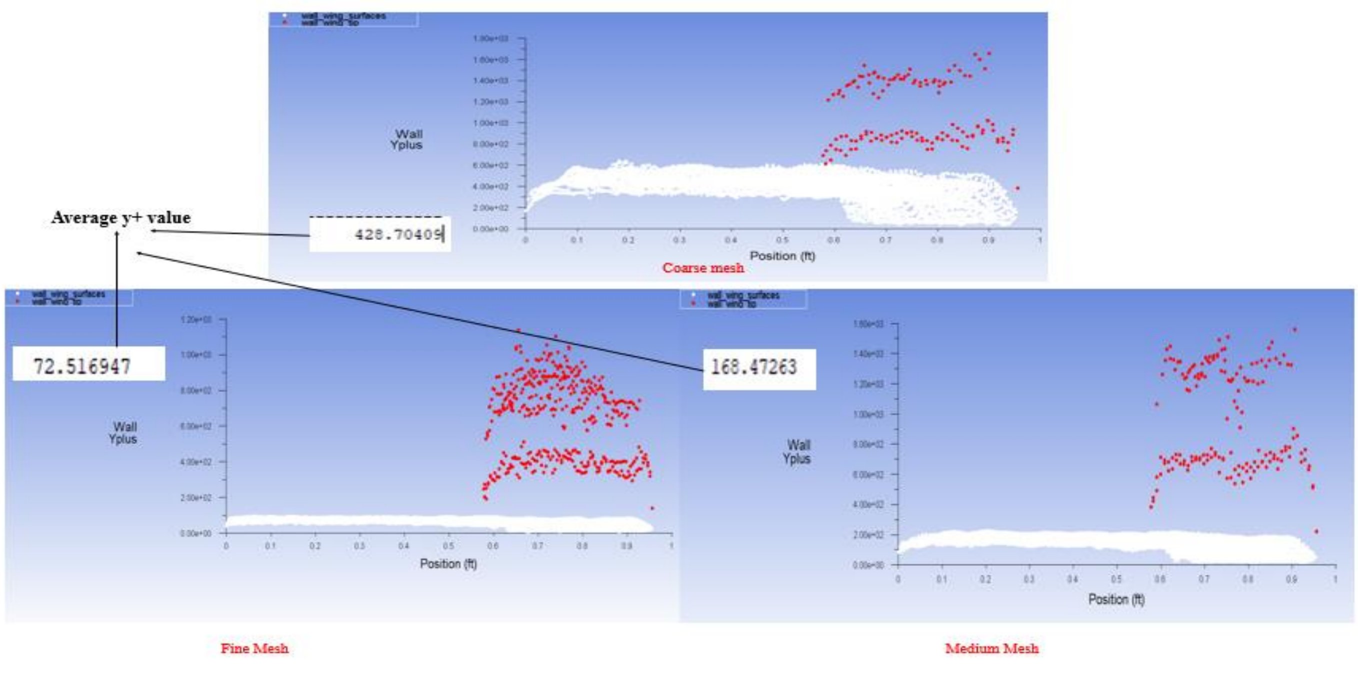


Figure 12- Wall y-plus for coarse, medium and fine mesh

From the above plots we can see that the Average values of Y+ keep decreasing with the increasing number of cell count**.** The values in the higher range depict the y+ for the tip whereas the values in the lower range depict the wall-wing surface.

Contour of static pressure at different z/b locations.

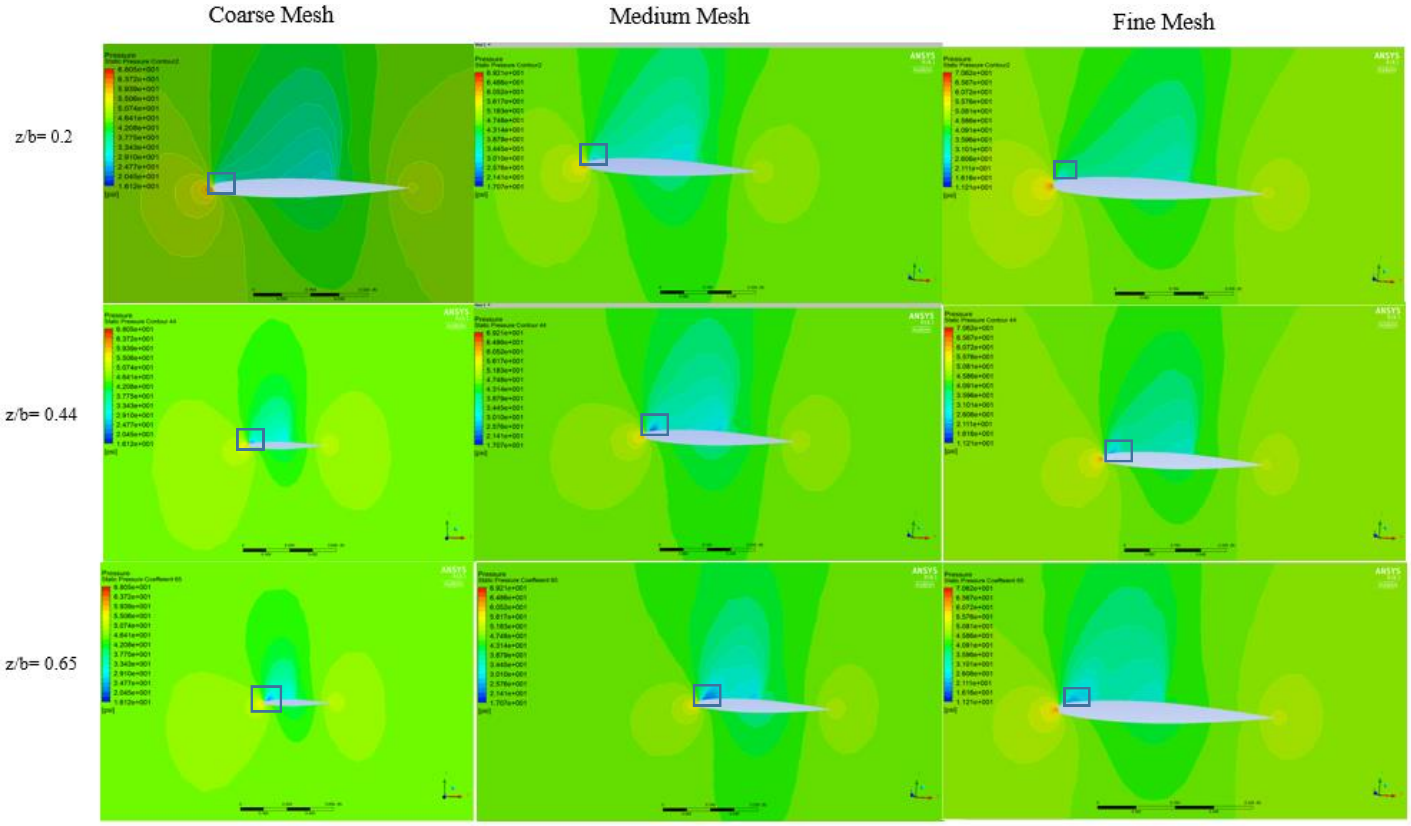


Figure 13- Pressure Contour for different meshes at different z/b locations.

From figure 13 we can infer that static pressure goes on decreasing on top of the leading edge for the following z/b locations which is shown by square box. After comparing the contour plots of Static pressure on the wing surface for the three meshes, it can be observed that the pressure distribution changes with increase in number of cell count. The trend being, the boundary layer separation occurs late going through the surface of the wing from Coarse to Finest

Contours of Mach number at different z/b locations.

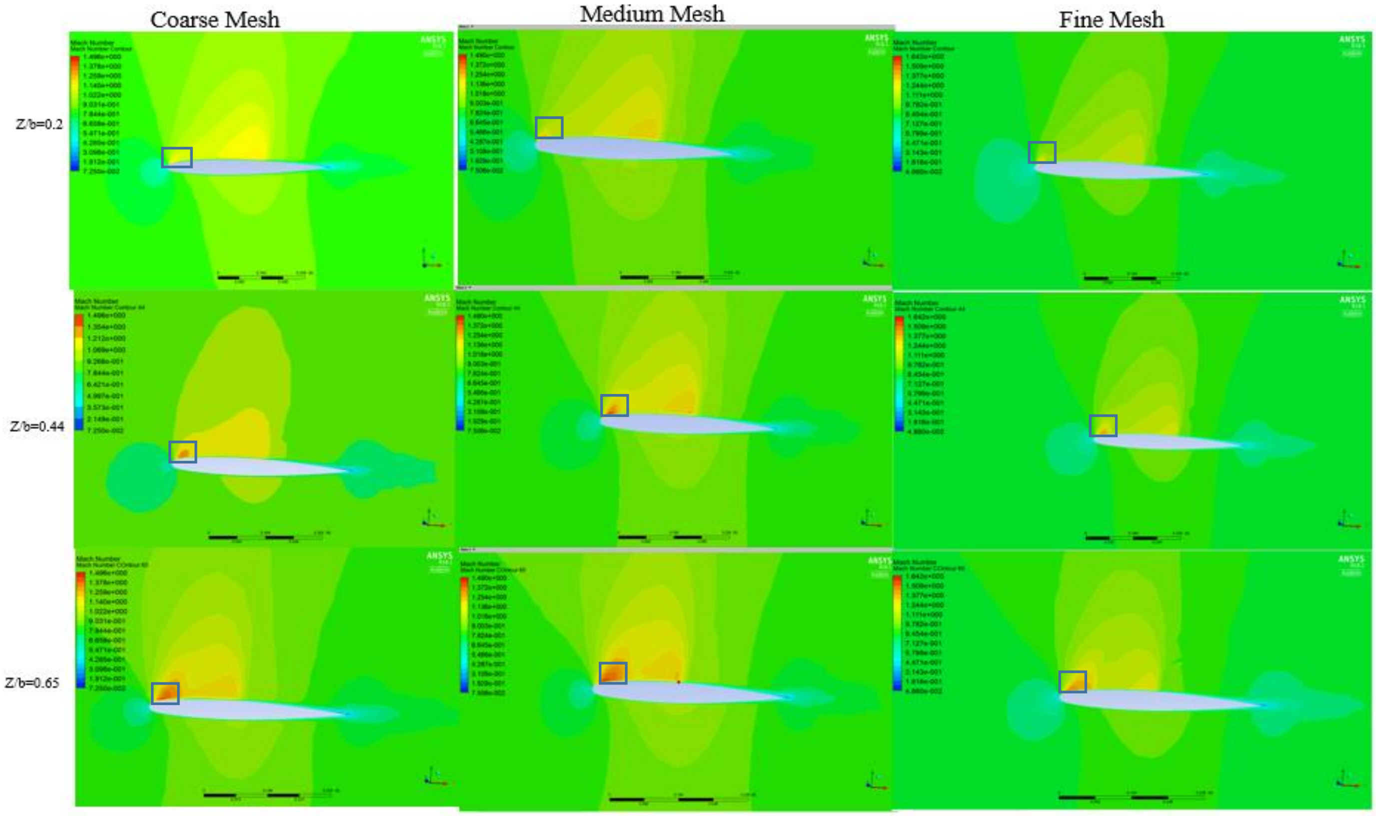


Figure 14- Mach Contour for different meshes at different z/b locations.

From this Figure we can see that there is a shock wave generated on top of the wing surface as we move away from the leading edge. Also, we can see that for fine mesh at Z/b =0.65 the shock wave is less than that of coarse and medium on the same location. So, we can infer that if we increase the number of walls near a boundary the chances of getting shock decreases.

Contours of Static Pressure on wall_wing_surfaces

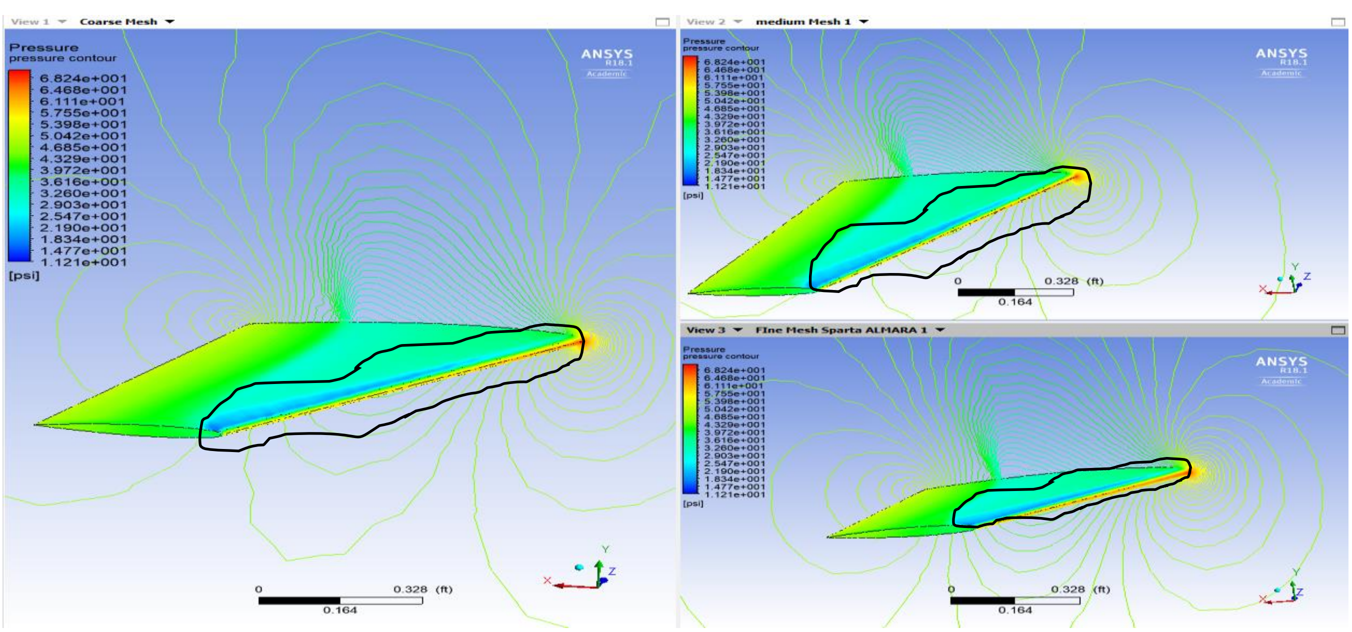


Figure 15- Static Pressure on wing surfaces for different meshes.

From above figure we can clearly see the shock generated by the span of the wing as the pressure changes drastically from high number to a low number. Also we can see that with more fine mesh the shock wave are clearly visible. The freeform shape in the figure shows the shockwave.

Turbulence Model Sensitivity Study

For the Turbulence Model Sensitivity Study we go with three turbulence model for the finest mesh which in my case is 2.2 million elements. Apart from Spalart-Allmaras the other two turbulence model that I have used are:

k-epsilon:

We solve for two variables , the turbulent kinetic energy (k) and rate of dissipation of kinetic energy(epsilon).
This technique gives good convergence.
Equation for epsilon is postulated so it has a difficulty solving the epsilon equation

k-omega:

It is a one equation turbulence model that has been developed specifically for aerodynamic flows .
It is based on kinematic eddy viscosity and mixing length
Since it deals with only one equation it very fast for non linear converging

Lift and Drag Coefficients

	CD	CL	%error, CD	%error, CL
NASA CFD	0.0088	0.141	0	0
Spalart -Allmaras	0.010282	0.14097	16.840	0.02
K-epsilon	0.010516	0.14229	19.5	0.914
K-omega	0.010392	0.14095	18.09091	0.035461

Table 8: Coefficient of Lift and Drag for different turbulence model used

From the comparision above we can see that error percentage for coefficient of drag and lift for spalart – allmaras is very less when compared to other turbulence models. Even though the drag is high I believe if we can increase the number of mesh near the walls we can achieve better results for drag.

Pressure Coefficients:

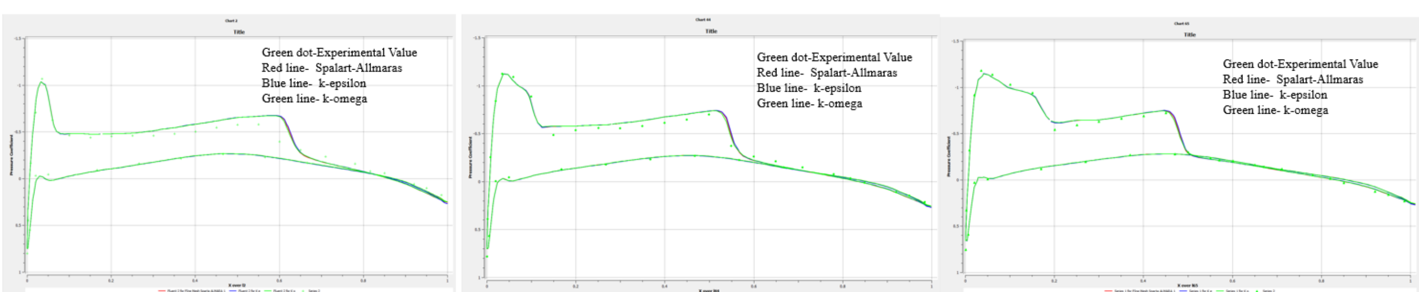


Figure 16--Pressure Coefficient vs X/C for different Z/b locations 0.2, 0.44, 0.65 from left to right for different turbulence model

From the above figure we can say that all the model is relatively good as they overlap each other and at the same time it overlaps the experimental value. Based on this it is hard to select a particular model and say it fits right for our model. So on the basis of Coefficient of lift and drag we select Spalart- Allmaras model.

Y+ on the Airfoil Surface for different turbulence model

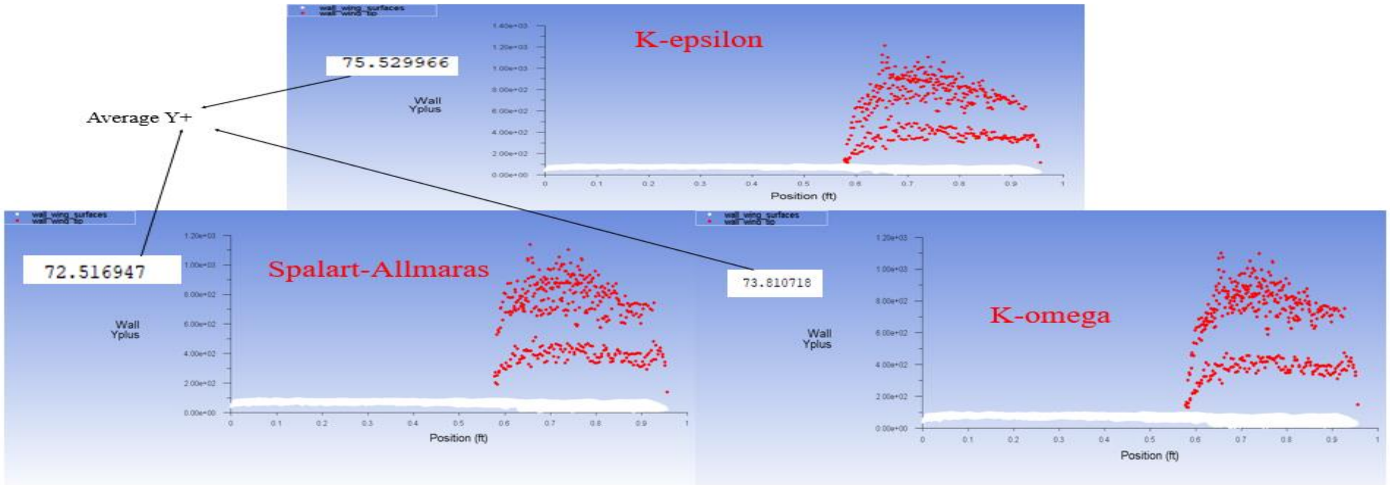


Figure 17 Wall y-plus for different turbulence model

From the above plots we can see that the Average values of Y+ are the lowest for the Spalart Allmaras Turbulence Model. Therefore giving further evidence to go with Spalart- Allmaras model. Although the ideal value of Y+ should be closer to one, but for our simulation these values lie in the expected range as for our models it ranges from 30 to 200.

Contour plots of Static Pressure at different z/b location.

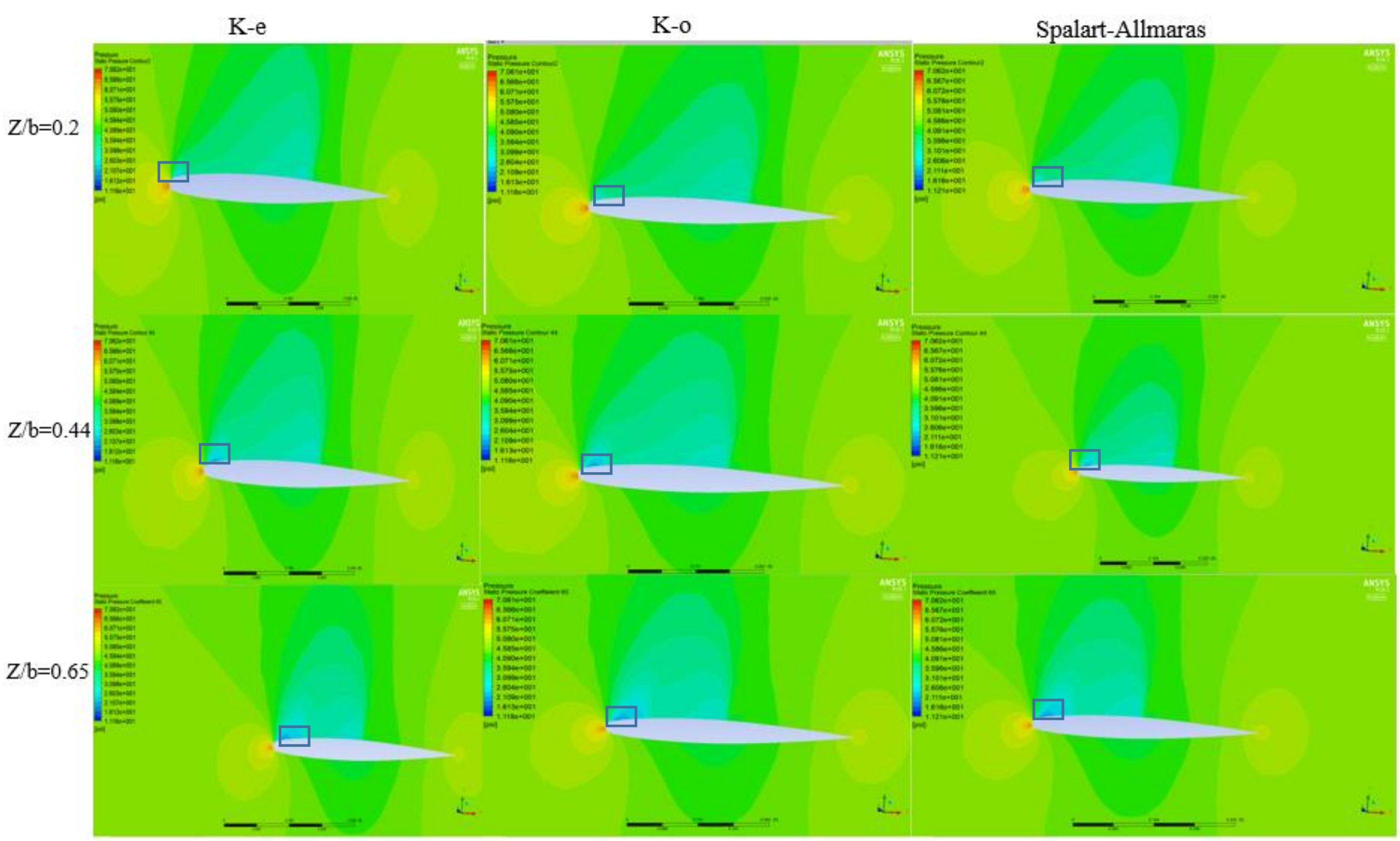


Figure 18- Pressure Contour for different turbulence models at different z/b locations.

The square box indicates the part of the wing surface where the pressure coefficient changes for different z/b locations.

Contour plot of Mach for different z/b locations

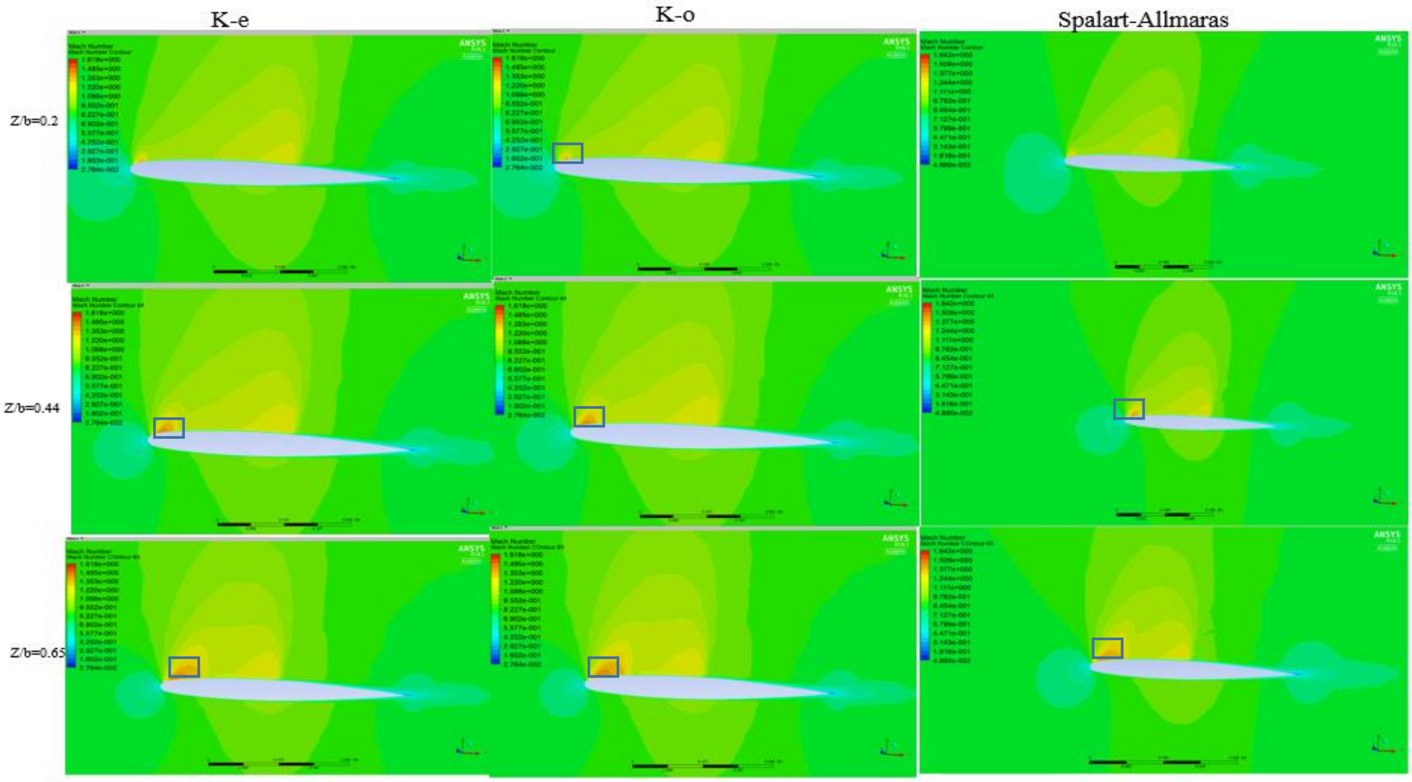


Figure 19- Mach Contour for different turbulence models at different z/b locations.

The square box indicates the part where the velocity of fluid is larger than speed of sound causing a shock wave. As you move away from the wing the Mach number increases on top of the wing surface wall causing shock wave.

Contour plot of Static pressure on different models

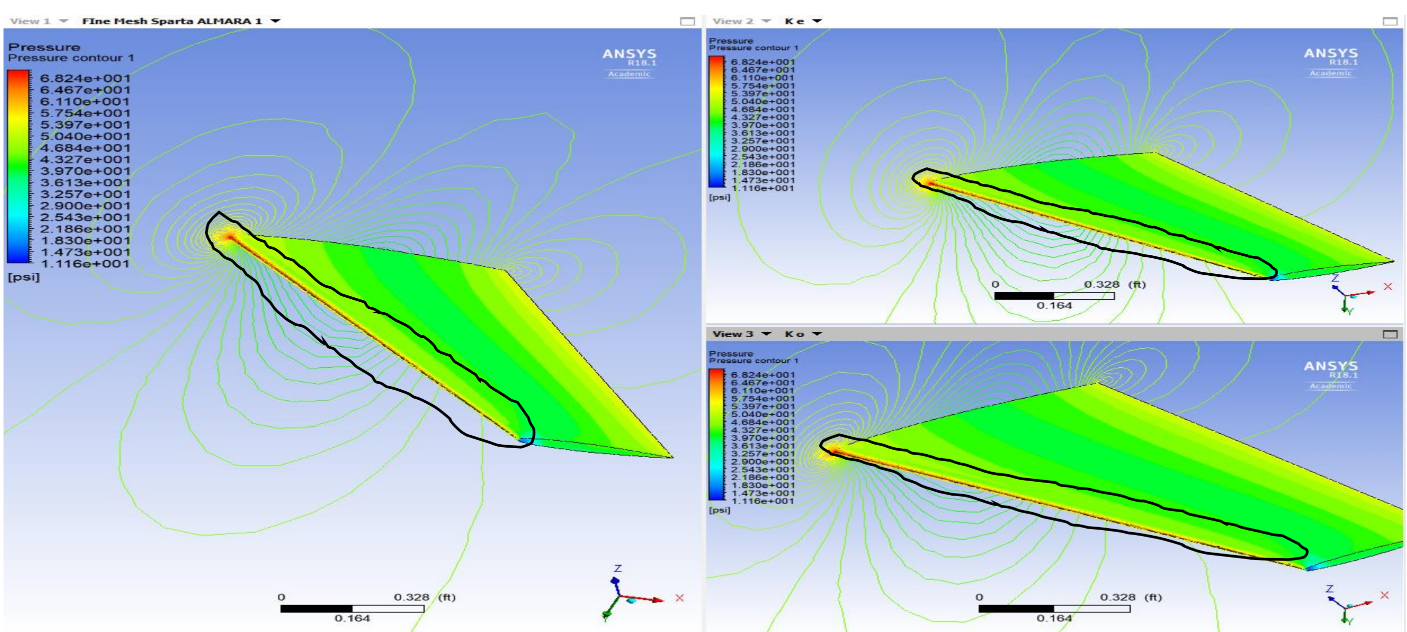


Figure 20- Static Pressure on wall_wing_surfaces for different models

From above figure we can see there is shock wave generated but it is hard to say which models gives us the best visualization for the shock wave.

Questions –

- Show that the Reynolds number based on the mean aerodynamic chord is indeed 11.72e+6?
- Reynold's Number is $\mathrm{Re} = \frac{\rho u L}{\mu}$

Therefore, for our problem the values are as follows –

a. Density = 0.2689053 lbm/ft3
b. Velocity (u) = 882.8473
c. Mean Aerodynamic Chord (L) = 0.54
d. Dynamic Viscosity = 1.09329e-5
e. Therefore, $Re = \frac{0.2689053 \times 882.8473 \times 0.54}{1.09329\mathrm{e}{-5}}$

   = 11720000

- How do you define lift, drag and pressure coefficients. Why is it important to have correct reference values, and how the reference values affect the calculation of the coefficients?
  A. The **lift coefficient** is a number that is used to model all of the complex dependencies of shape, inclination and some flow conditions on lift. This equation is simply a rearrangement of the lift equation where we solve for the lift coefficient in terms of the other variables.
  B. The **drag coefficient** is a number that is used to model all of the complex dependencies of shape, inclination and flow conditions on aircraft drag. This equation is simply a rearrangement of the drag equation where we solve for the drag coefficient in terms of the other variables.
  C. The **pressure coefficient** is the ratio of pressure forces to inertial forces and dimensionless number which describes the relative pressures throughout a flow field in fluid dynamics.
  D. The reference values are used to non-dimensionalize the forces and moments acting on the airfoil. The dimensionless forces and moments are the lift, drag, and moment coefficients