1. Title –Supersonic flow analysis over nosecone using Ansys Fluent.

2. Objective –

1. To extract the profile of nose cone from standard mathematical formulas and design it using Spaceclaim software.
2. To mesh with proper element size and inflation layer using Ansys Mechanical.
3. To set up the case by choosing the suitable solver model, turbulence model and boundary conditions.
4. To generate some useful contours and plots and analyse them.

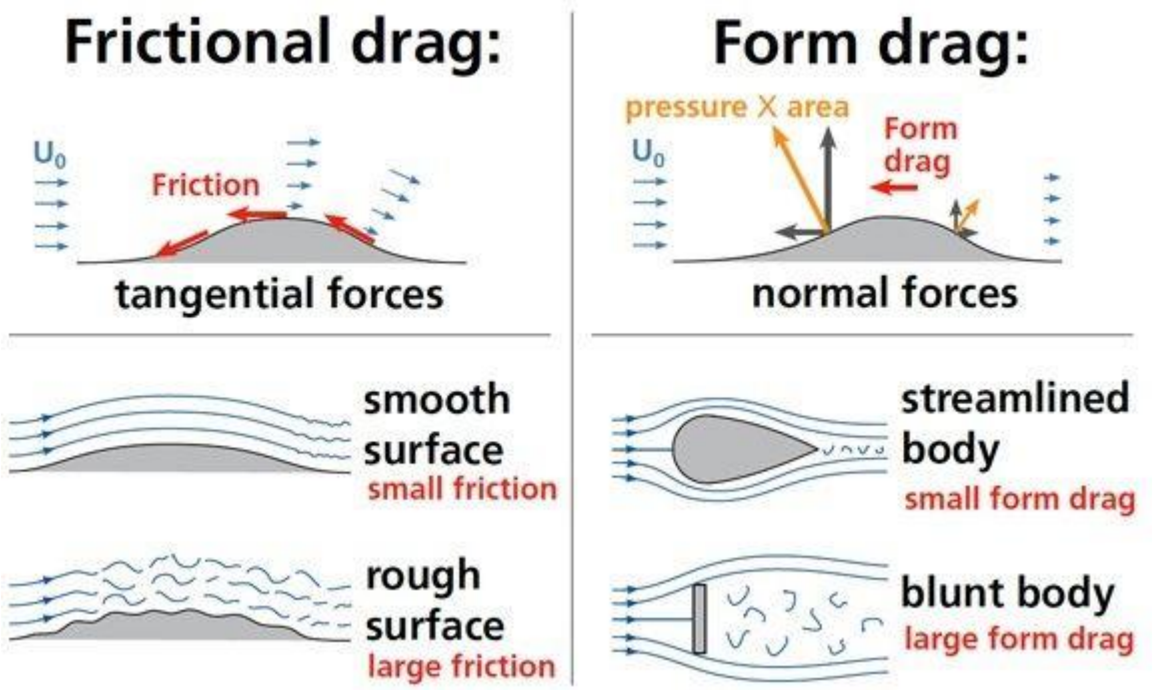
3. Introduction –

At the present era rising fuel prices is a concern for vehicle industry whether it is road, water or air vehicle. For these industries maintaining stability of vehicle and fuel economy is a major challenge. These concerns are forcing them to design more efficient vehicle. one can omit these concerns for road vehicle but for space and aircraft industry these challenges cannot be overlooked, because it attaches lots of money with it. Drag force is one the most important factor which impacts the fuel economy and stability directly. Little changes in design can lead into reduction of drag in grater manner. In this study we will look at the drag coefficient possessed by various nosecones at subsonic and supersonic designs.

4. Theory –

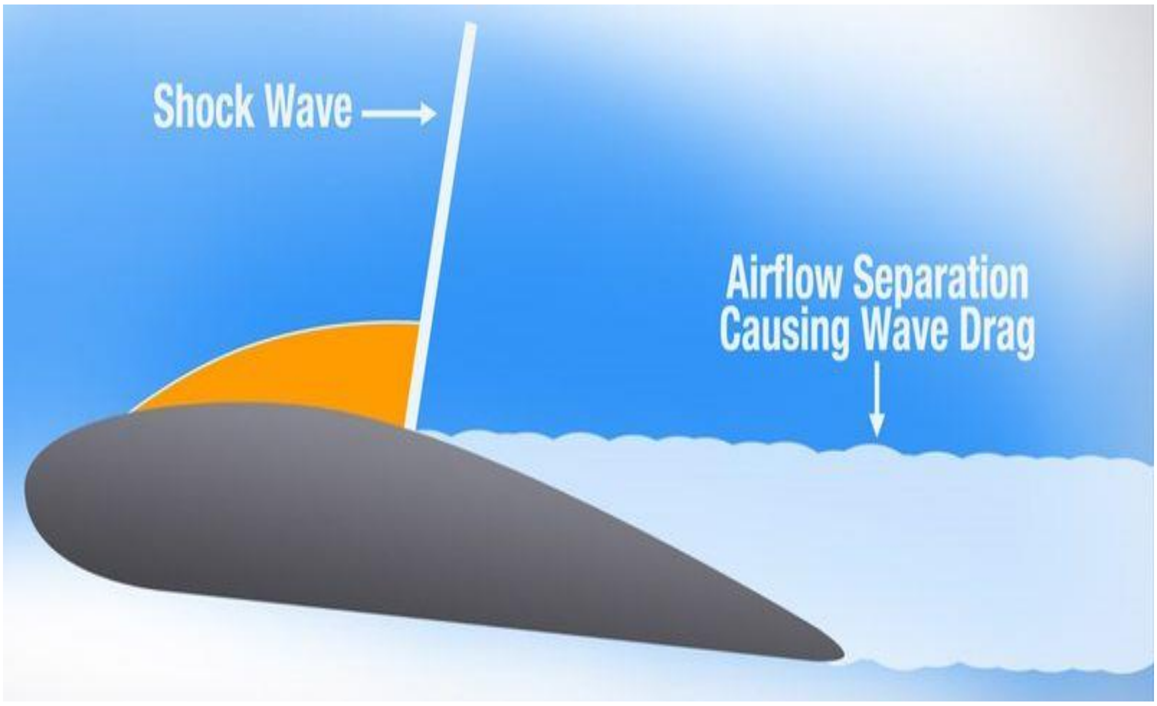
Drag is a force acting opposite to the relative motion of the object. Drag force depends on the velocity unlike dry friction. In laminar flow drag force is proportional to velocity and squared velocity for a turbulent flow. Drag is generally caused by following phenomenon –

Skin friction - In general, when a fluid flows over a stationary surface like the flat plate, the bed of a river, or the wall of a pipe, the fluid touching the surface is brought to rest by the shear stress to at the wall. This shear force is called as drag due to skin friction. This type of drag force, depends especially on the geometry, the roughness of the solid surface and on the type of fluid flow.



Form drag or pressure drag – it arises because of the shape and size of the object. According to Bernoulli's principle, faster moving air exerts less pressure. This causes, that there can be a pressure difference between surfaces of the object. The general size and shape of the body are the most important factors in form drag.

Wave drag – it is associated with shock wave. When aircraft approaches the speed of sound, shock wave are generated which produces static pressure, ultimately results into a force called wave drag.



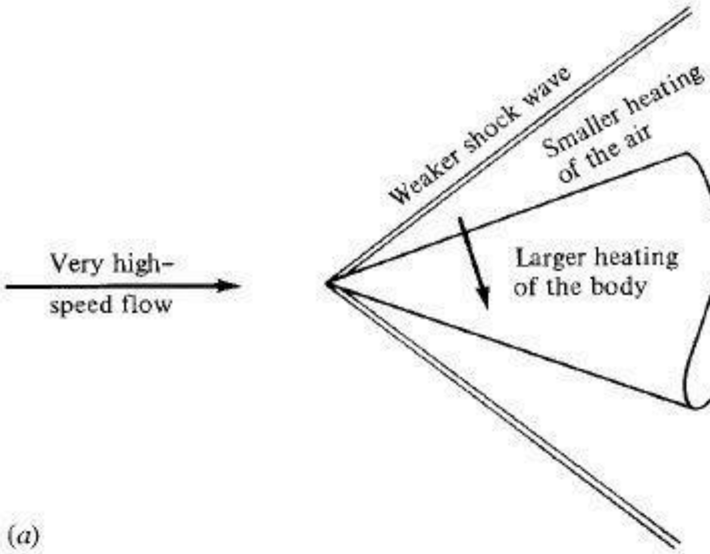
In this study we will consider the effect of all the drag forces and find the total drag force, although at high Reynolds number the impact of pressure drag is significant.

Fineness ratio - The ratio of the length of a nosecone compared to its the base diameter is known as the Fineness Ratio, for example a nosecone that is 1.5m long and 0.5 in diameter would have a fineness ratio of 3:1. In this study we will best choose best design and vary its fineness ratio in order to see the impact of bluntness on coefficient of drag.

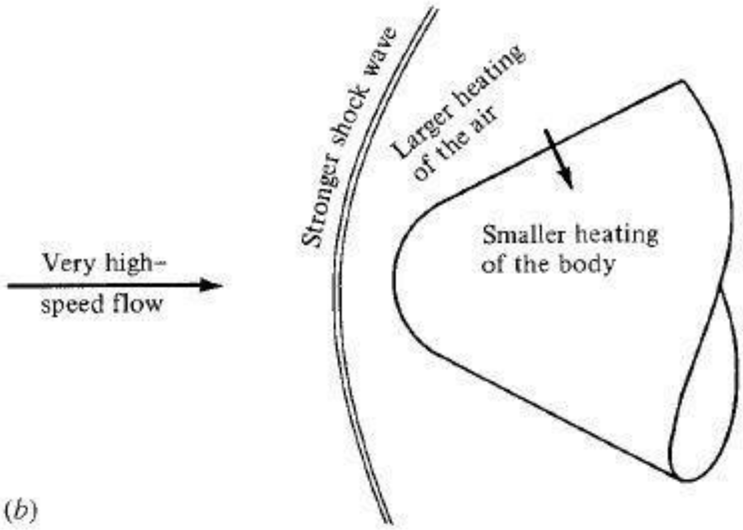
The allied problems of predicting the drag of body bodies of revolution and minimizing the drag by proper shaping of the body have been the objects of numerous theoretical investigations. The drag of bodies has now assumed greater importance because drag rise of an airplane can be the same as its equivalent body. Obviously, the airplane designer would like his airplane to have a low-drag equivalent body. Drag reductions can be obtained in two ways, first, through increasing the body fineness ratio, and second, through better shaping of the body profile at a given fineness ratio. In this study we will look at the both factors, how fineness ratio and body shape can affect coefficient of drag.

There are two types of nosecone, used in aviation industry – sharp nosecone and blunt nosecone. Pin pointed or sharp nose is majorly used in military aircrafts and ultrasonic jet and these are designed for maximum efficiency while blunt nose is designed for maximum stability and these are used in commercial aircraft. Fighter plane and rockets are designed with sharp nose because its design facilitates high speed in low time. But commercial aircraft is using blunt nosecone, why?. Well!, The disadvantage of the blunt nose is its property of offering resistance, hence due to this engine has to produce high power to overcome resistance. but, this resistance is somehow advantageous in the case of landing of the vehicle because due to resistance, the aircraft's speed is reduced. Hence during take-off, high amount of fuel is required. Also, landing of aircraft should be safe because the first and foremost priority for the aviation industry is the safety of the passenger and its crew members.

Sharp nosecone vs Blunt nosecone



(a)

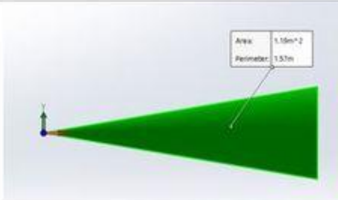
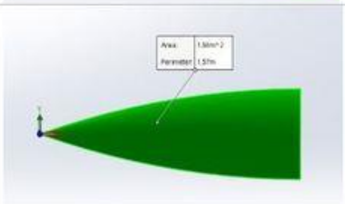
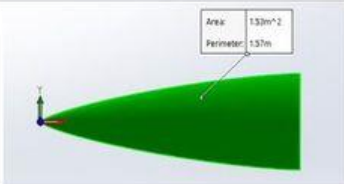
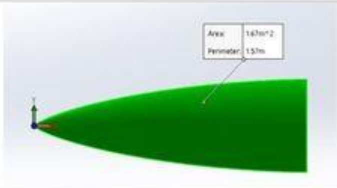
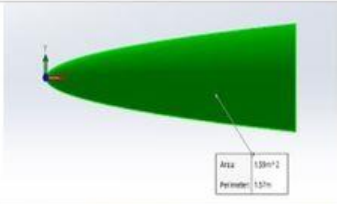
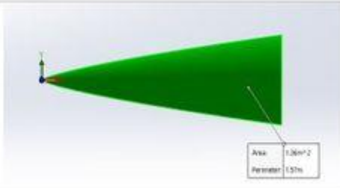


(b)

In the aircraft that has to return on the earth, the air is struck directly on the wings of the vehicle and due to this, the whole pressure is diverted from the sharp cone design to the wings of aircraft. So huge amount of friction occurs on the wings which causes its temperature rise up to 3000 degree. This causes melting of wings and the failure of the aircrafts. So, to avoid the superheating of wings, different shape of the nosecones are developed that is the blunt nose which deflects the shock waves due to its blunt contact and the shock waves remains away from the wings. And the heating caused due to the drag resistance remains low. Now it is understandable, nosecone design preference is varying according to the application.

Till now we talked about the nosecone categories in broad manner, respective to their design and pros and cons, but within those categories we can divide nosecones further on the basis of their design like power series designs, ogive series designs, Haack series design etc. we will use different standard mathematical formulas to create the profiles of the nosecone. All the equation defines the 2D profile of the nose and full body can be produced by just rotating it around central line.

Table containing nose profiles and formulas

Nose Profile Name	Nose Profile Equation	Nose Profile	3D model
Sharp Cone	$y = \frac{xR}{L}$		
Tangent Ogive	$\rho = \frac{R^2 + L^2}{2R}$ $y = \sqrt{\rho^2 - (L-x)^2} + R - \rho$		
LD-Haack (Von Karman)	$\theta = \arccos\left(1 - \frac{2x}{L}\right)$ $y = \frac{R}{\sqrt{\pi}}\sqrt{\theta - \frac{\sin(2\theta)}{2} + C\sin^3(\theta)}$ **C = 0**		
LV-Haack	$\theta = \arccos\left(1 - \frac{2x}{L}\right)$ $y = \frac{R}{\sqrt{\pi}}\sqrt{\theta - \frac{\sin(2\theta)}{2} + C\sin^3(\theta)}$ **C = 1/3**		
Power series - Parabola	$y = R\left(\frac{x}{L}\right)^n$ **n = 1/2**		
Power series - 3/4	$y = R\left(\frac{x}{L}\right)^n$ **n = 3/4**		

Note - images of 3D model is looking different due to size of image but fineness ratio was maintained 3.

5. Procedure –

Three are three main steps to approach any CFD problem – Pre-processing, Solving and Post-processing. Pre-processing includes CAD model creation or its repair, if it is imported, meshing. In solving step, we solve flow equations with suitable model of turbulence, species, multiphase, discreate particle, radiation etc., If any. Post-processing includes plot, contour, streamline creation for analysis purpose.

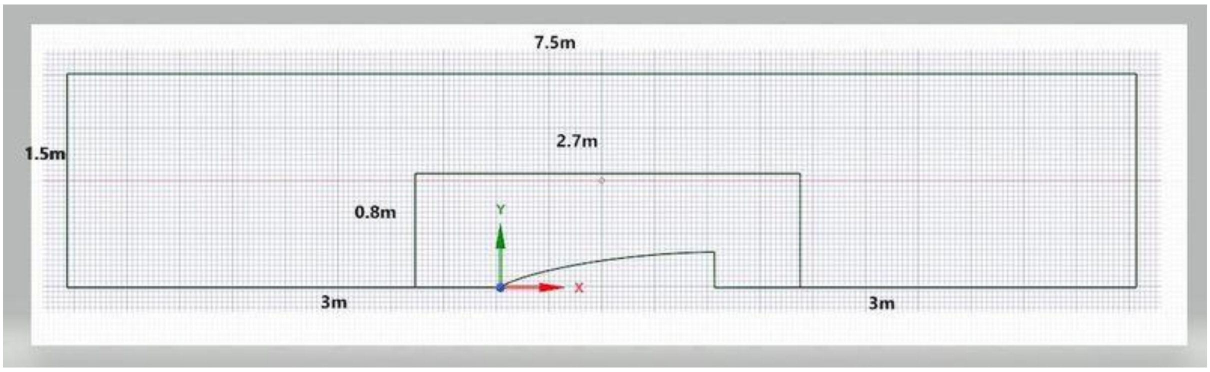
Geometry – Spaceclaim was used to create nosecone. First coordinates points of nose curve were imported and then a domain was created around it to make a fluid zone. For the coordinate points, we wrote a MATLAB program which had standard mathematical formulas respective to nose cone series. There were 7 different geometries created on a fixed fineness ratio of 3 and additionally four other geometries were created which were varying on fineness ratio.

Length of Nose = 1.5m

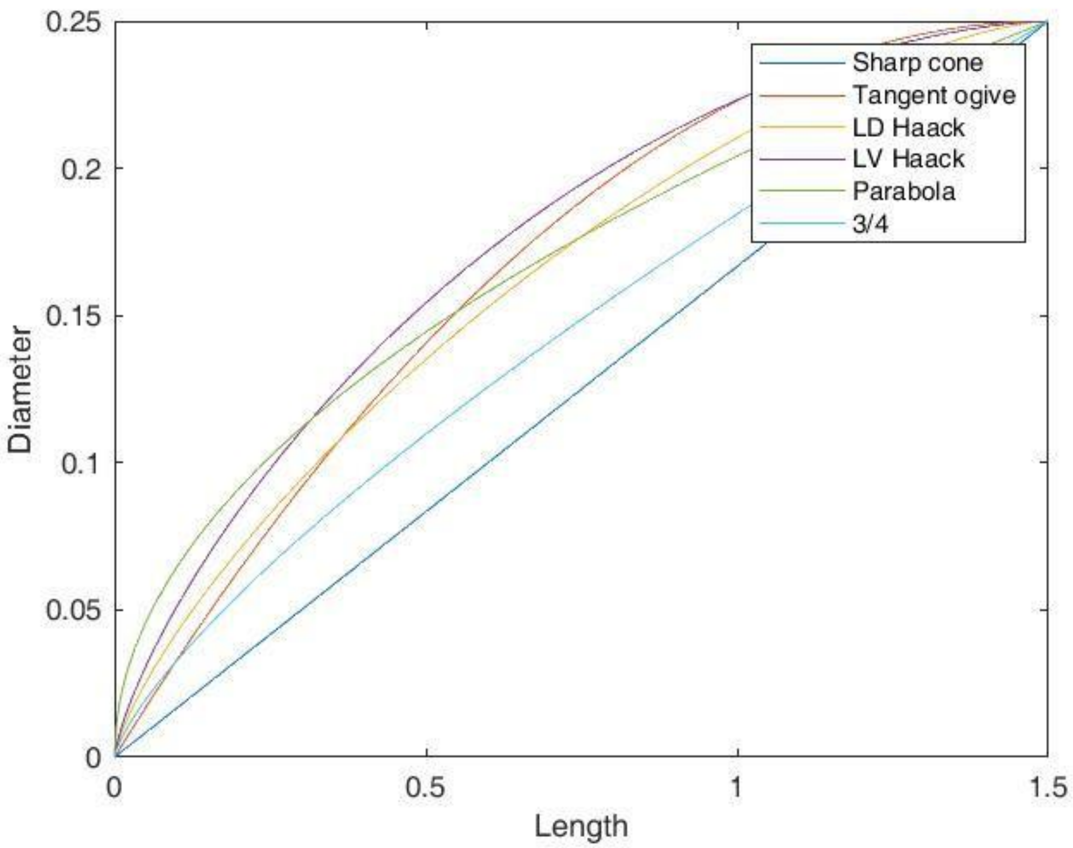
Diameter = 0.5m

Fineness ratio = 1.5/0.5 = 3

LV haack geometry and dimensions

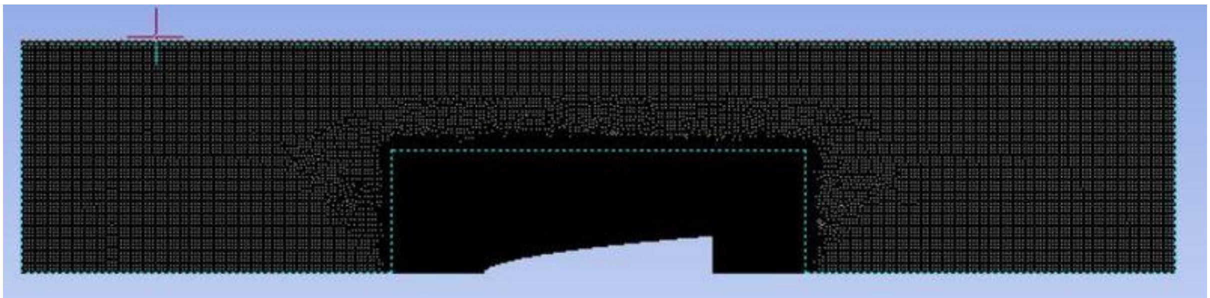


Curve comparison

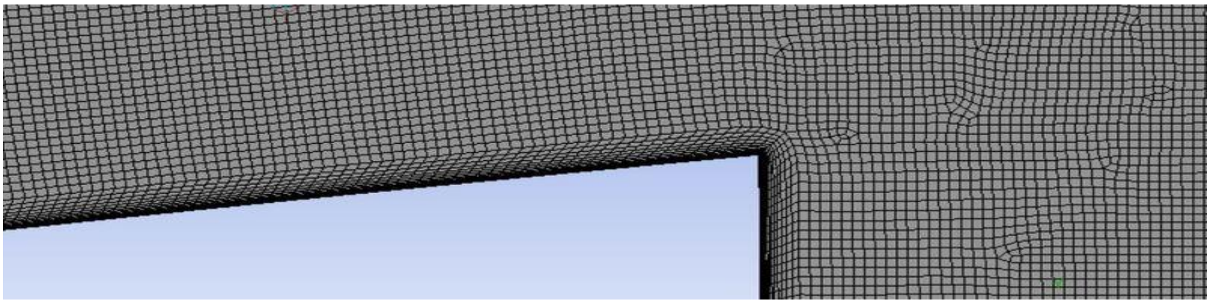


Meshing – Nosecone body is an axisymmetric body, so it is necessary to select 2D-axisymmetric in geometry setting. since the model is simple and 2D, we went with structured mesh, because it fits well in the domain and provides a smooth flow. A sub domain was created where finer mesh was deployed than main domain, because we wanted to capture physics properly around the nosecone body. Now one think, what if we put finest possible mesh over whole domain, well it will increase computational time, which is undesirable and it will not affect the results if global mesh size is selected wisely. 15 inflation layers were also added, because drag depends on the wall and to capture it properly, it is necessary to have a proper size and mesh flow around the body. We did not consider y plus values for height because at supersonic speeds, boundary layer will be very thin, so it will not affect the results. The parameters and settings can be seen in following images.

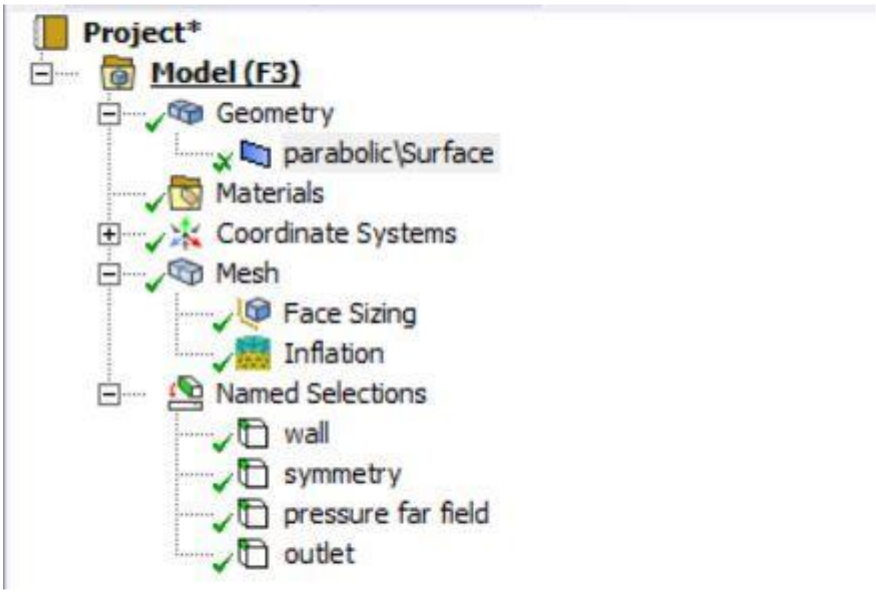
Mesh



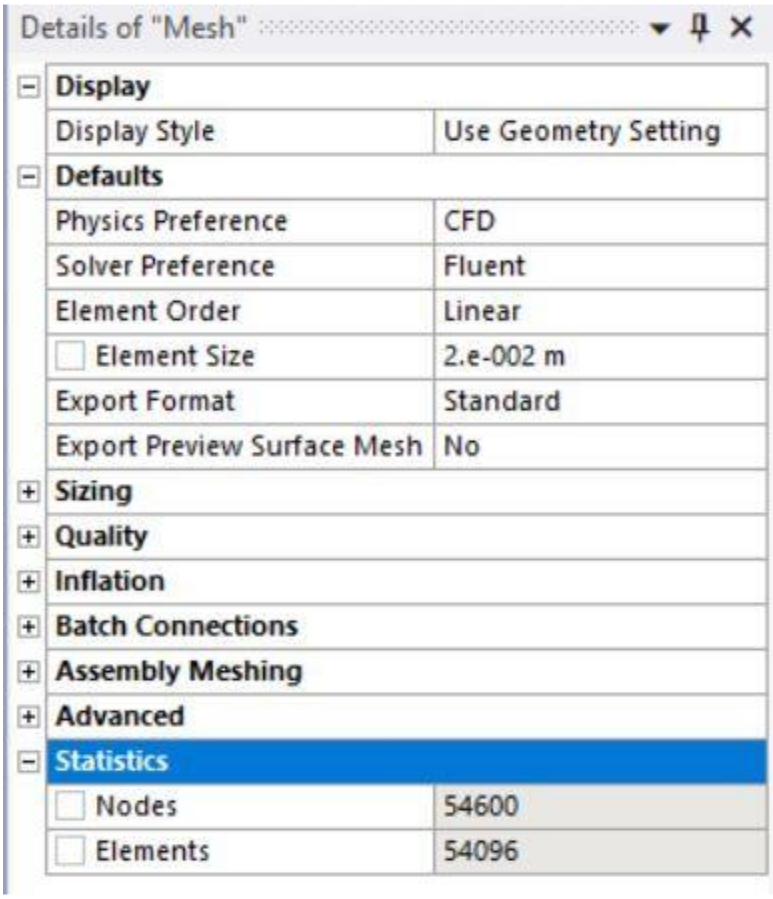
Inflation layers

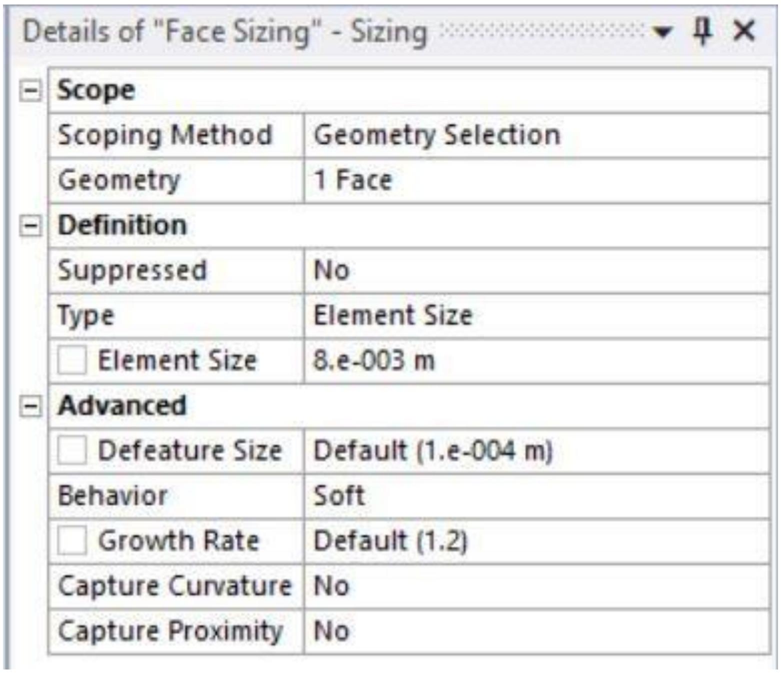


Mesh Tree



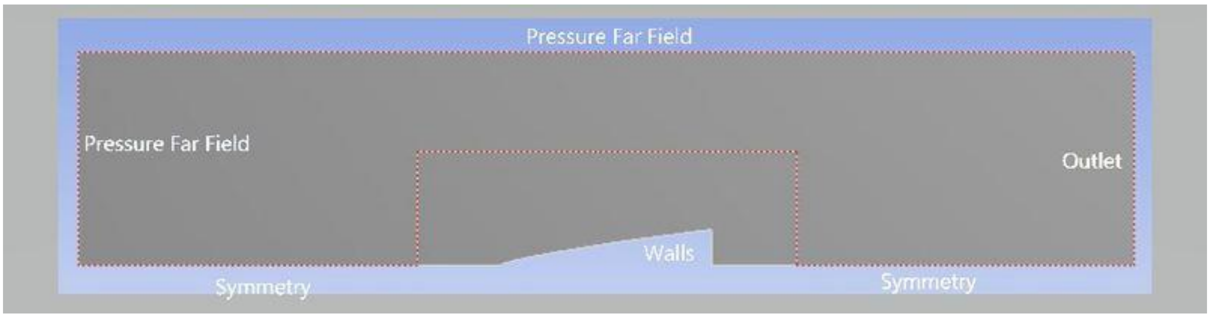
Mesh Parameters


Details of "Mesh"
Display
Display Style
Use Geometry Setting
Defaults
Physics Preference
CFD
Solver Preference
Fluent
Element Order
Linear
Element Size
2.e-002 m
Export Format
Standard
Export Preview Surface Mesh
No
Sizing
Quality
Inflation
Batch Connections
Assembly Meshing
Advanced
Statistics
Nodes
54600
Elements
54096


Details of "Face Sizing" - Sizing
Scope
Scoping Method
Geometry Selection
Geometry
1 Face
Definition
Suppressed
No
Type
Element Size
Element Size
8.e-003 m
Advanced
Defeature Size
Default (1.e-004 m)
Behavior
Soft
Growth Rate
Default (1.2)
Capture Curvature
No
Capture Proximity
No

Details of "Inflation" - Inflation	
Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	2 Edges
Inflation Option	First Layer Thickness
First Layer Height	4.e-004 m
Maximum Layers	2
Growth Rate	1.2
Inflation Algorithm	Pre

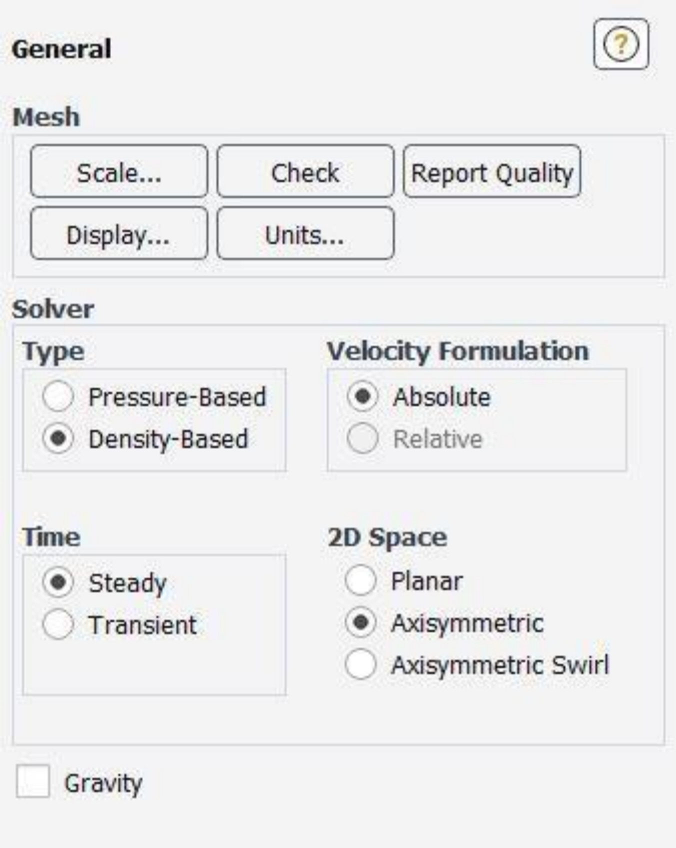
Named Boundaries



Solver setup –

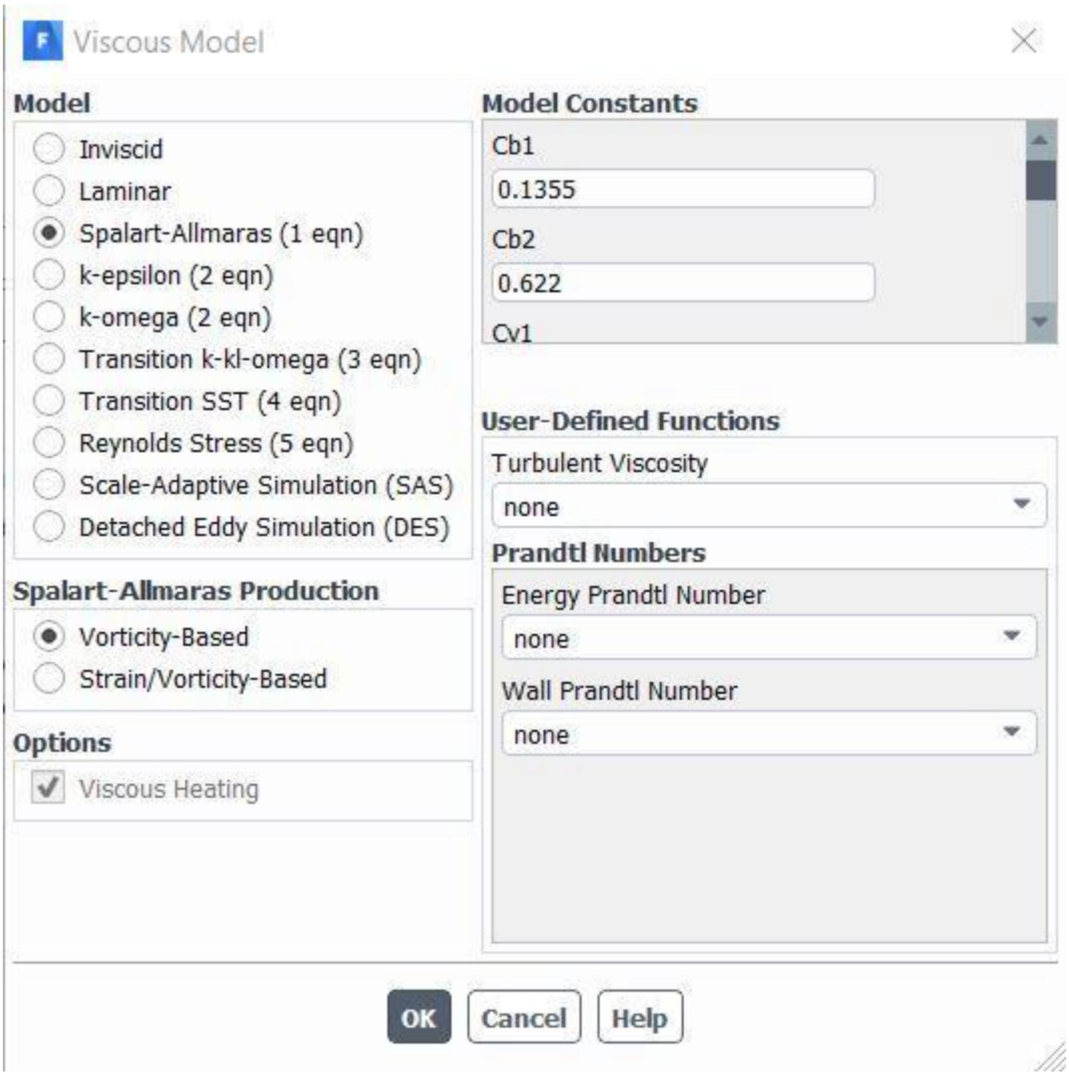
1. We started with general setting. We chose density-based solver. In incompressible flow density is assumed to constant, so flow equations are computed based on pressure equation, but in compressible flow density veries and it is coupled with energy. both the solver can be used in any application but for compressible flow density-based solver giver better accuracy.

General setting



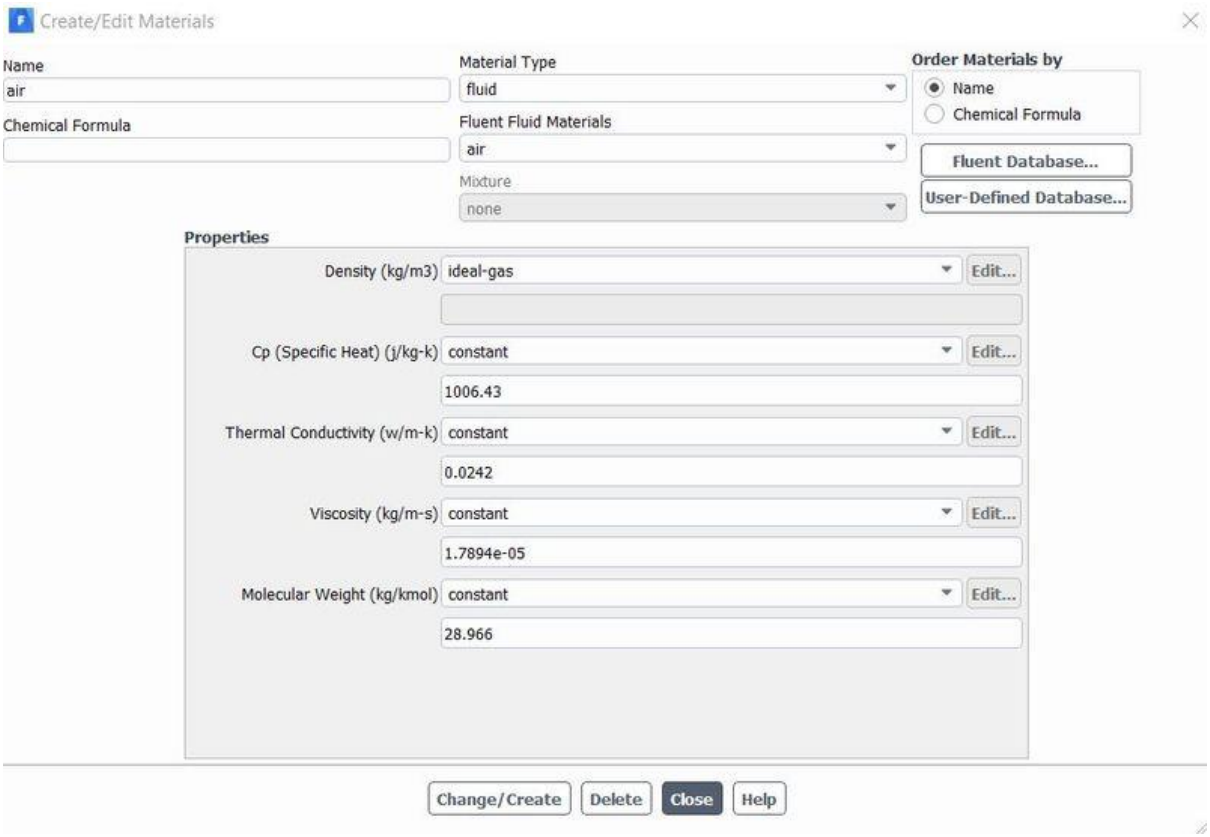
2. To capture turbulent, we used Spalart-Allmaras viscous model, which was created specially for aerodynamic simulations. It solves one equation, so computation time can be reduced. Another model we could consider K-omega SST model, which is a combination of K-epsilon and Standard K-omega model. K-epsilon model is considered for free stream and K-omega is best for near wall phenomenon. A factor was used in K-omega SST, which switches model between K-epsilon, when there is a free stream and K-omega, when we want to capture physics accurately near wall. Our simulation requires to capture both free stream and near wall phenomenon, so K-Omega model can be used. In this study we simulated using both models, the change in result was very minute, so we went with Spalart-Allmaras model, because it was consuming less time. Model constants and other parameters were used as following image.

Turbulent model selection



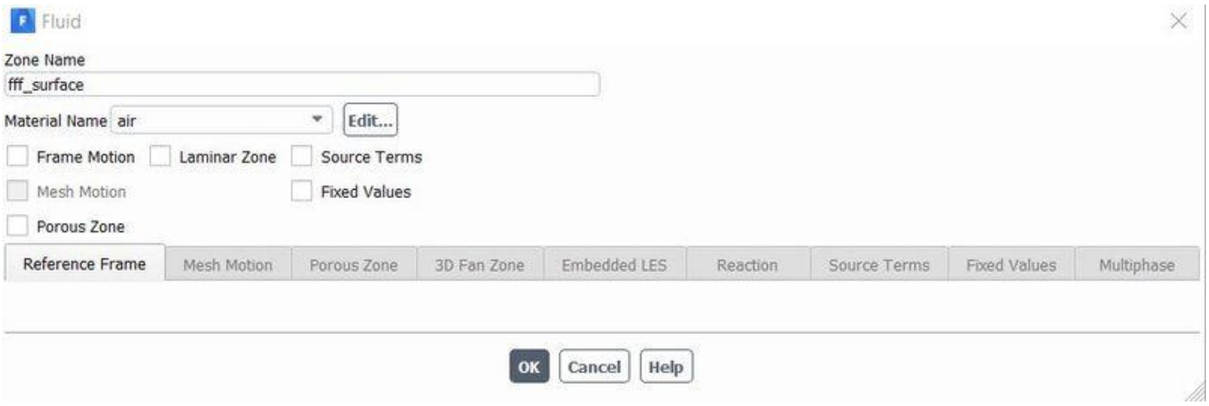
3. Air was considered as flowing fluid. For subsonic flow density and viscosity were kept constant and in supersonic flow, air was following the properties of ideal gas, because we need to consider compressibility effect. Energy equation will be enabled automatically because it is coupled with ideal gas and density solver combination.

Material panel

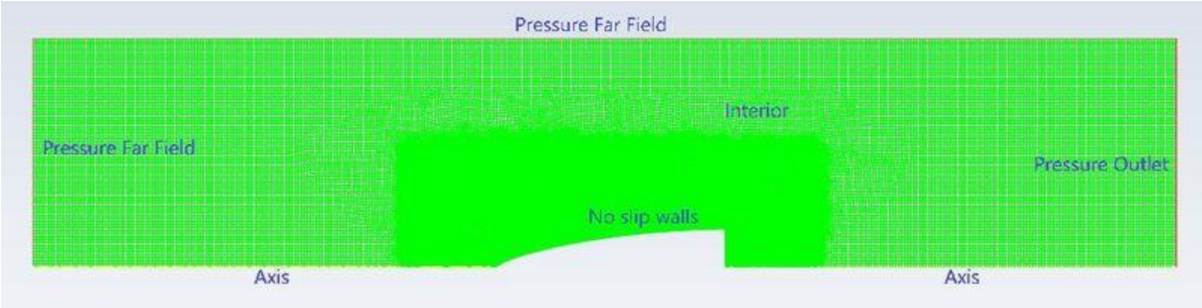


4. Fluent can automatically detects the zone type, in our case it was fluid. There were four boundaries we need to define. We cannot give motion to nosecone, so consider it stationery and air was moving with desired speed. For inlet pressure far field was chosen as boundary type, because it creates free stream, a broad space like situation. Gauge pressure and temperature were kept as defaults and we were changing Mach number only.

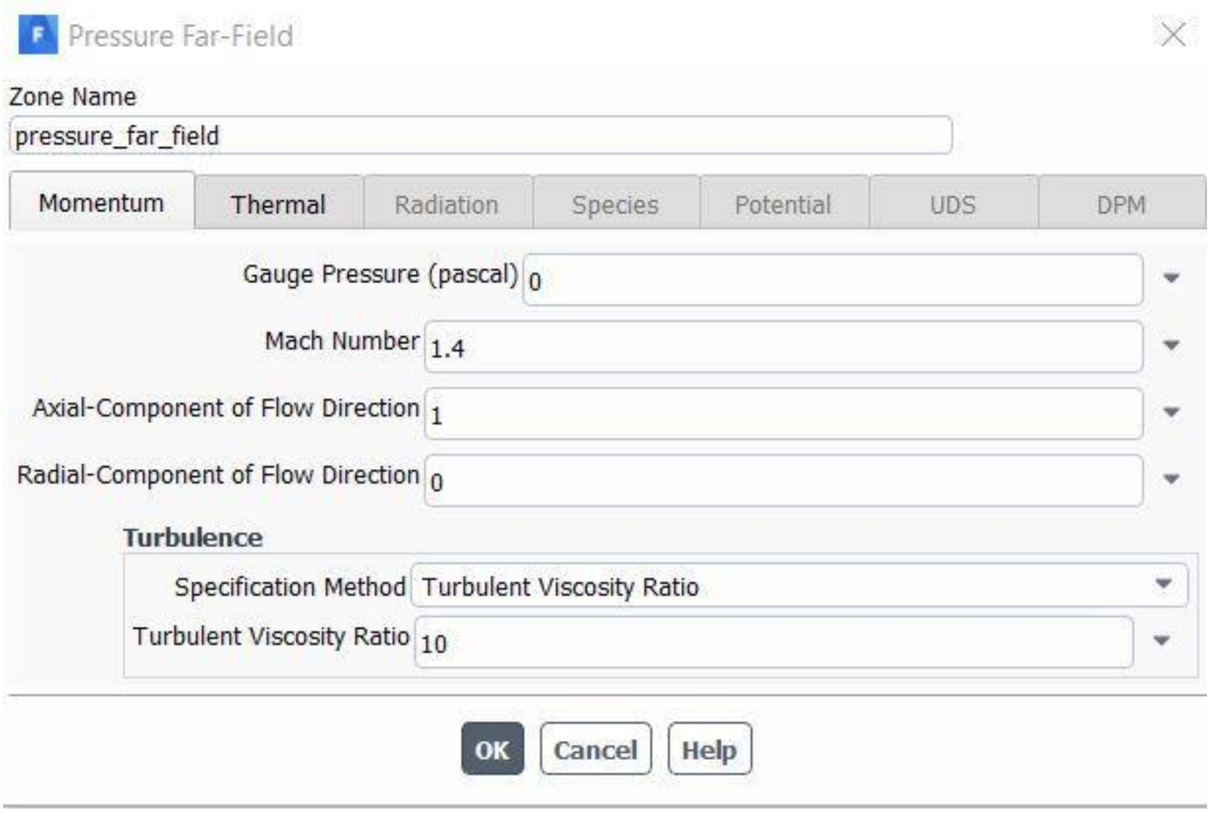
Cell zone condition -



Boundary conditions

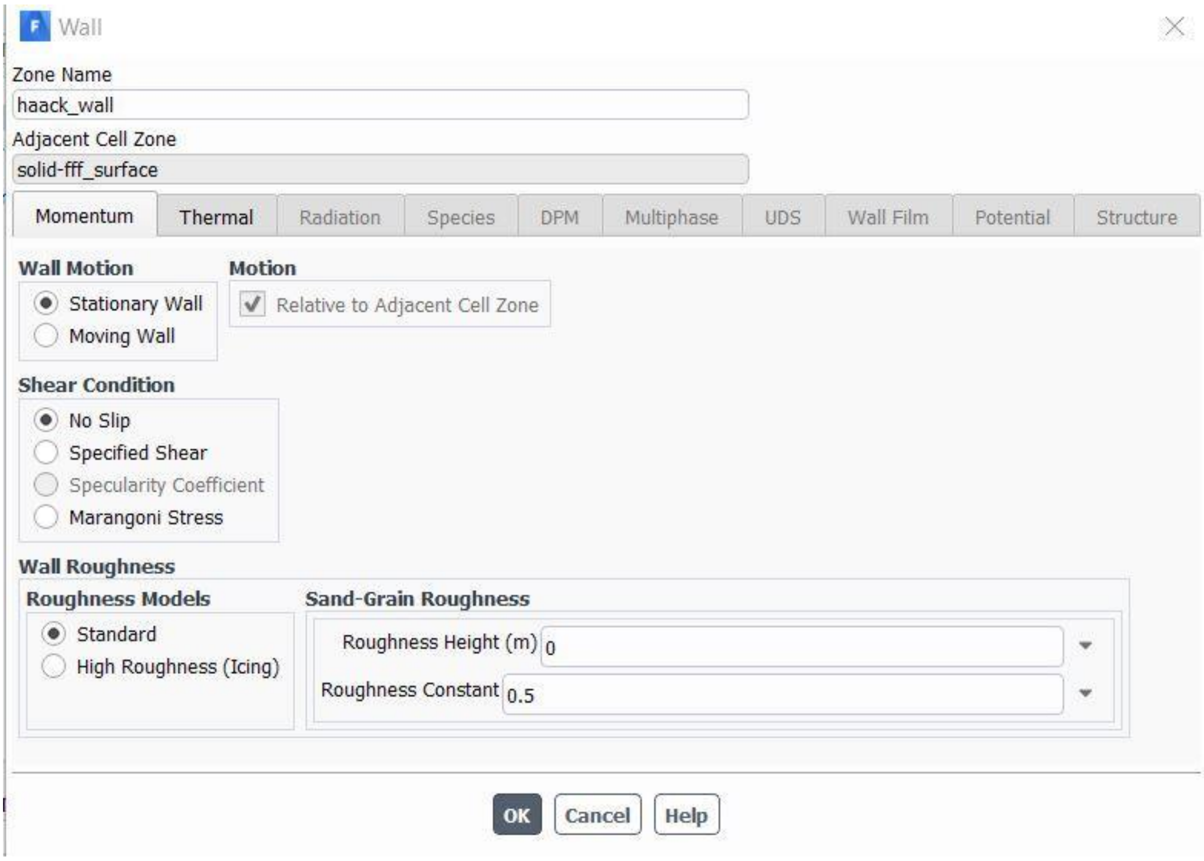


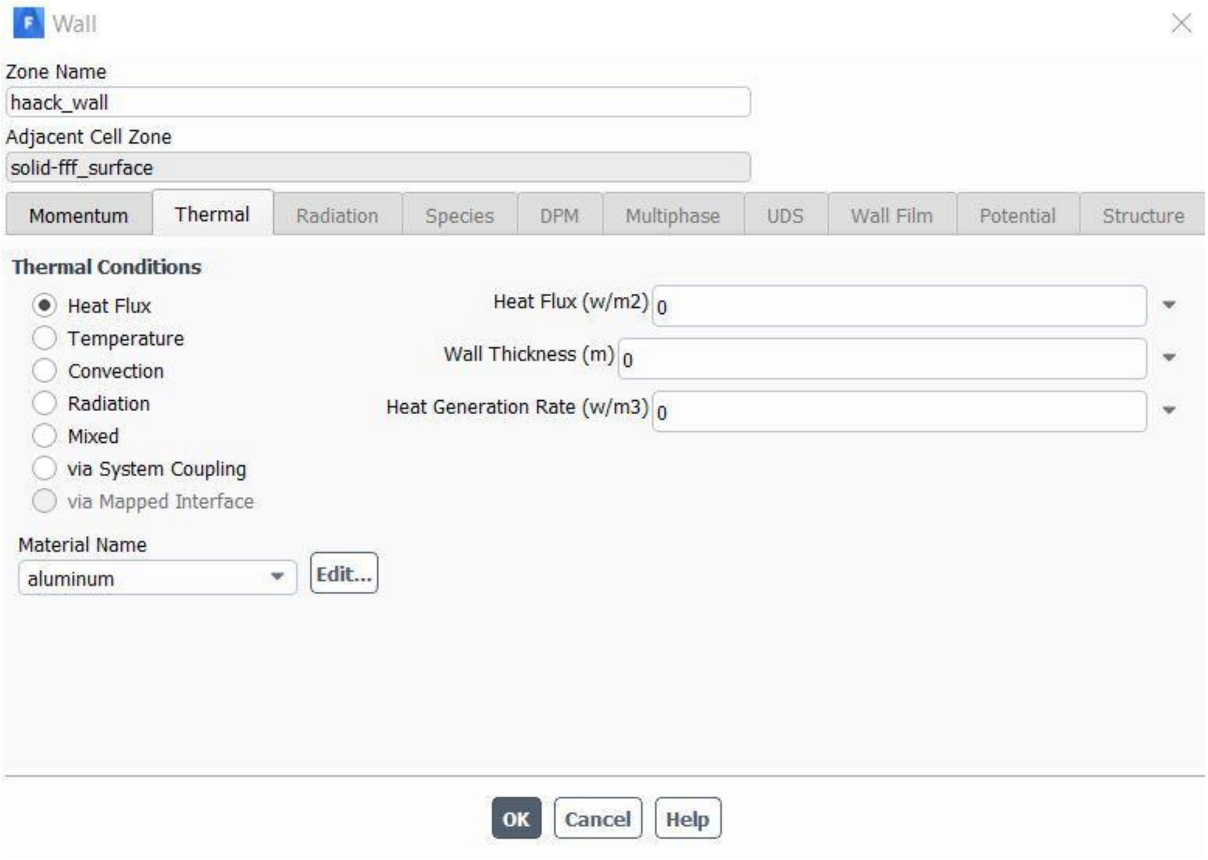
Pressure far field




Pressure Far-Field
Zone Name
pressure_far_field
Momentum
Thermal
Radiation
Species
Potential
UDS
DPM
Temperature (k)
300
OK
Cancel
Help

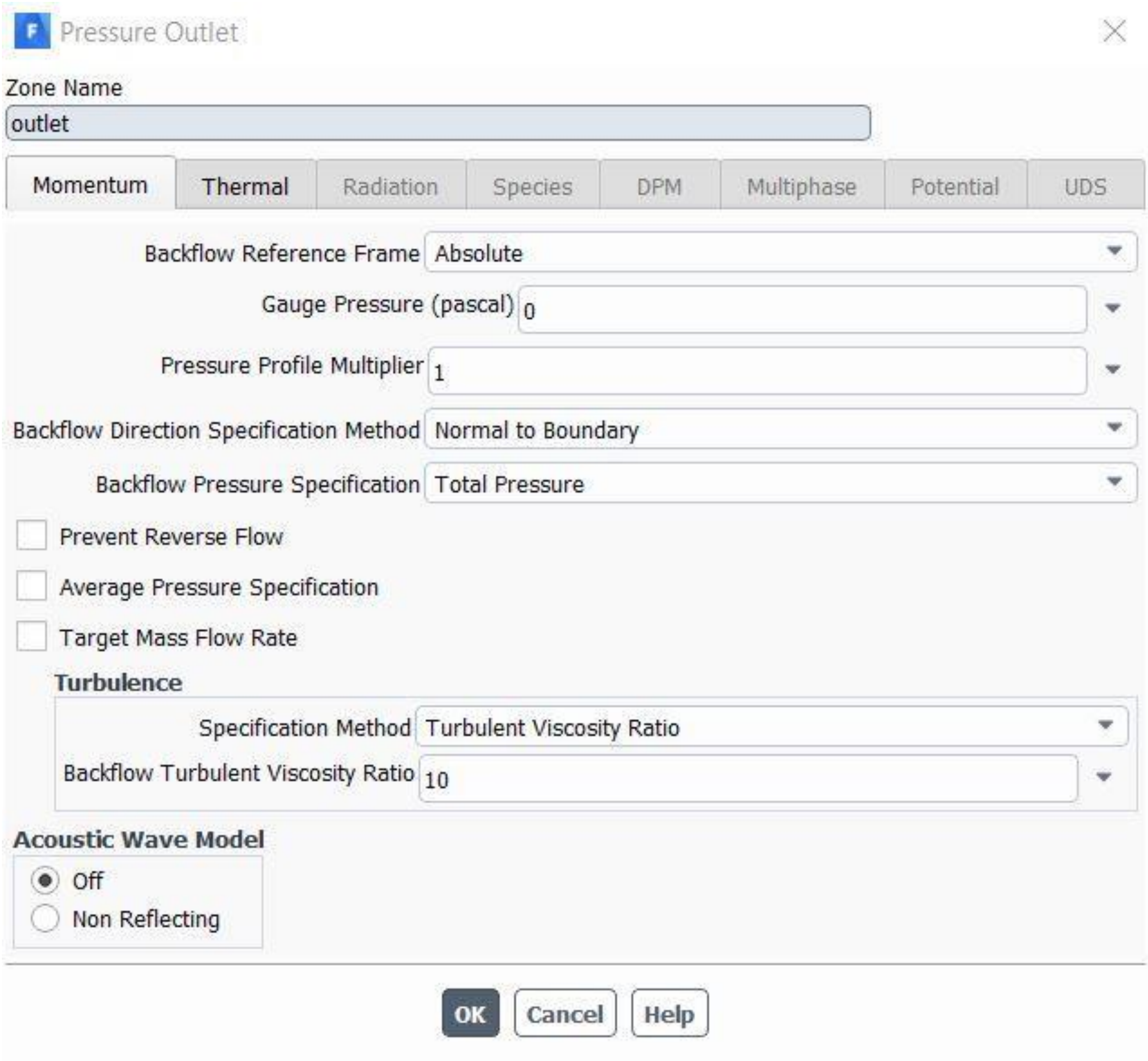
Wall

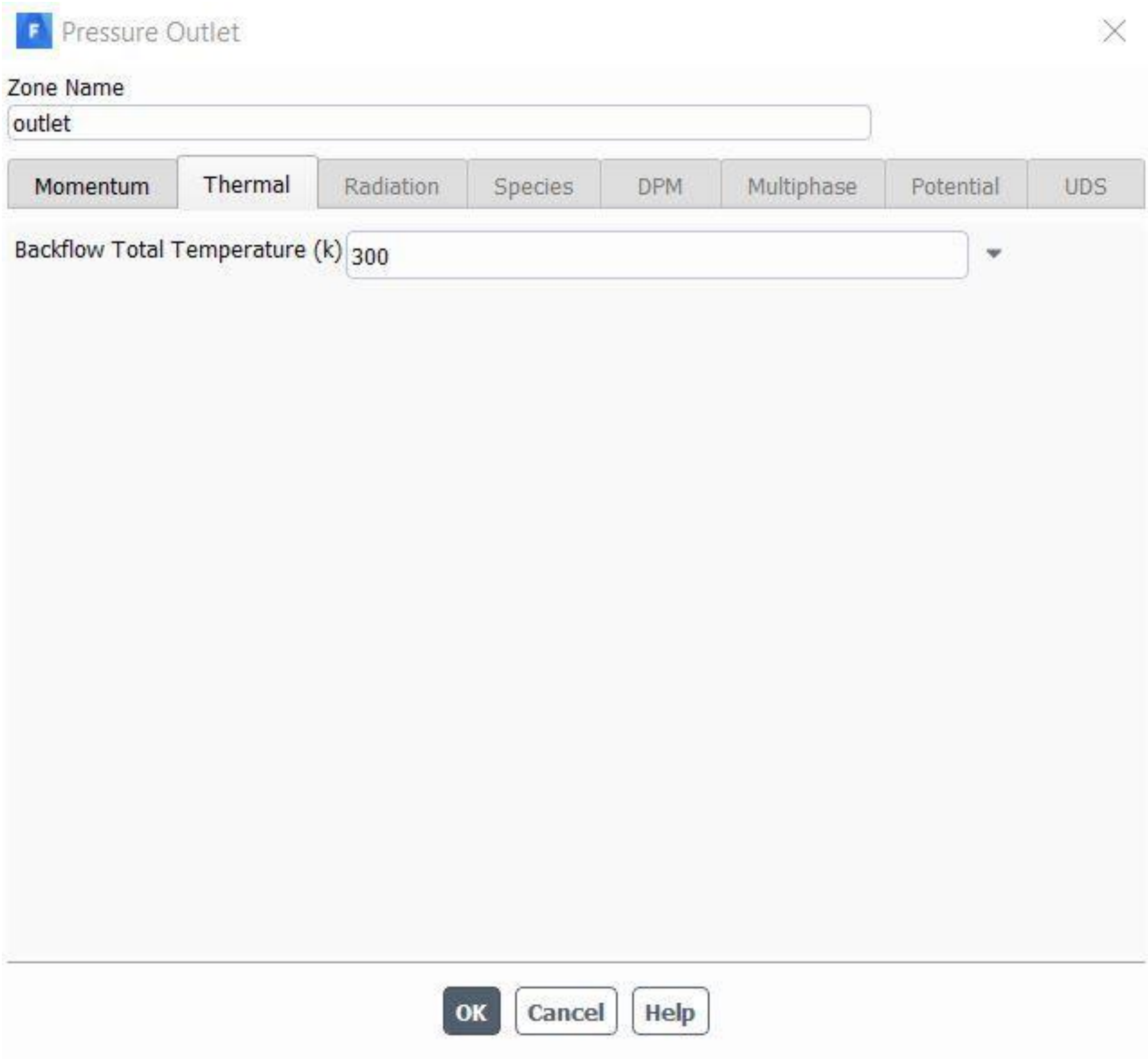

Wall
Zone Name
haack_wall
Adjacent Cell Zone
solid-fff_surface
Momentum
Thermal
Radiation
Species
DPM
Multiphase
UDS
Wall Film
Potential
Structure
Wall Motion
Stationary Wall
Moving Wall
Motion
Relative to Adjacent Cell Zone
Shear Condition
No Slip
Specified Shear
Specularity Coefficient
Marangoni Stress
Wall Roughness
Roughness Models
Standard
High Roughness (Icing)
Sand-Grain Roughness
Roughness Height (m)
0
Roughness Constant
0.5
OK
Cancel
Help



For outlet boundary pressure outlet was chosen as boundary type, we went with default pressure value. Nosecone body was treated as stationary wall and no slip was given as constraint. No slip means some fluid layers are stagnant adjacent to wall. Since it was an axisymmetric simulation, symmetric wall selected as axis boundary type. Sub domain, where mesh was finer, was considered as interior.

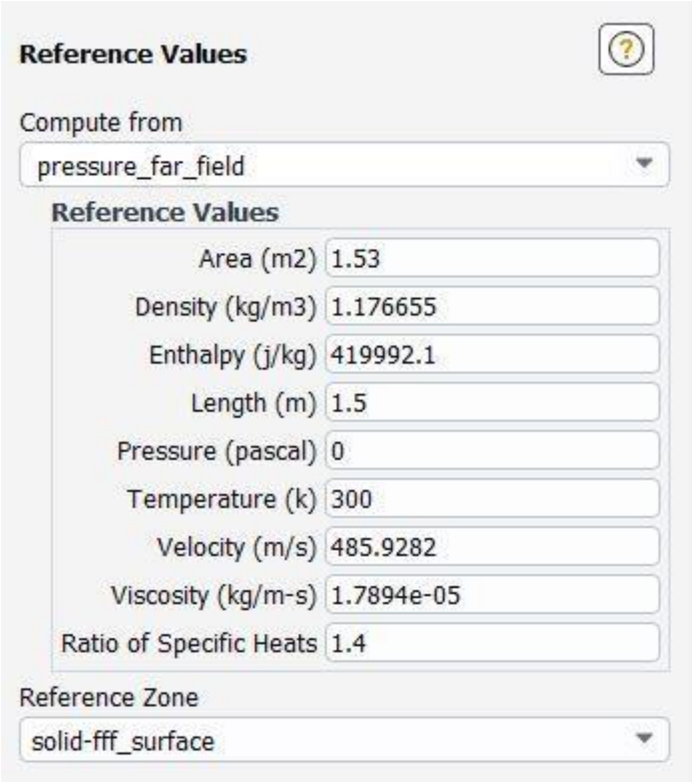
Outflow


Pressure Outlet
Zone Name
outlet
Momentum
Thermal
Radiation
Species
DPM
Multiphase
Potential
UDS
Backflow Reference Frame
Absolute
Gauge Pressure (pascal)
0
Pressure Profile Multiplier
1
Backflow Direction Specification Method
Normal to Boundary
Backflow Pressure Specification
Total Pressure
Prevent Reverse Flow
Average Pressure Specification
Target Mass Flow Rate
Turbulence
Specification Method
Turbulent Viscosity Ratio
Backflow Turbulent Viscosity Ratio
10
Acoustic Wave Model
Off
Non Reflecting
OK
Cancel
Help



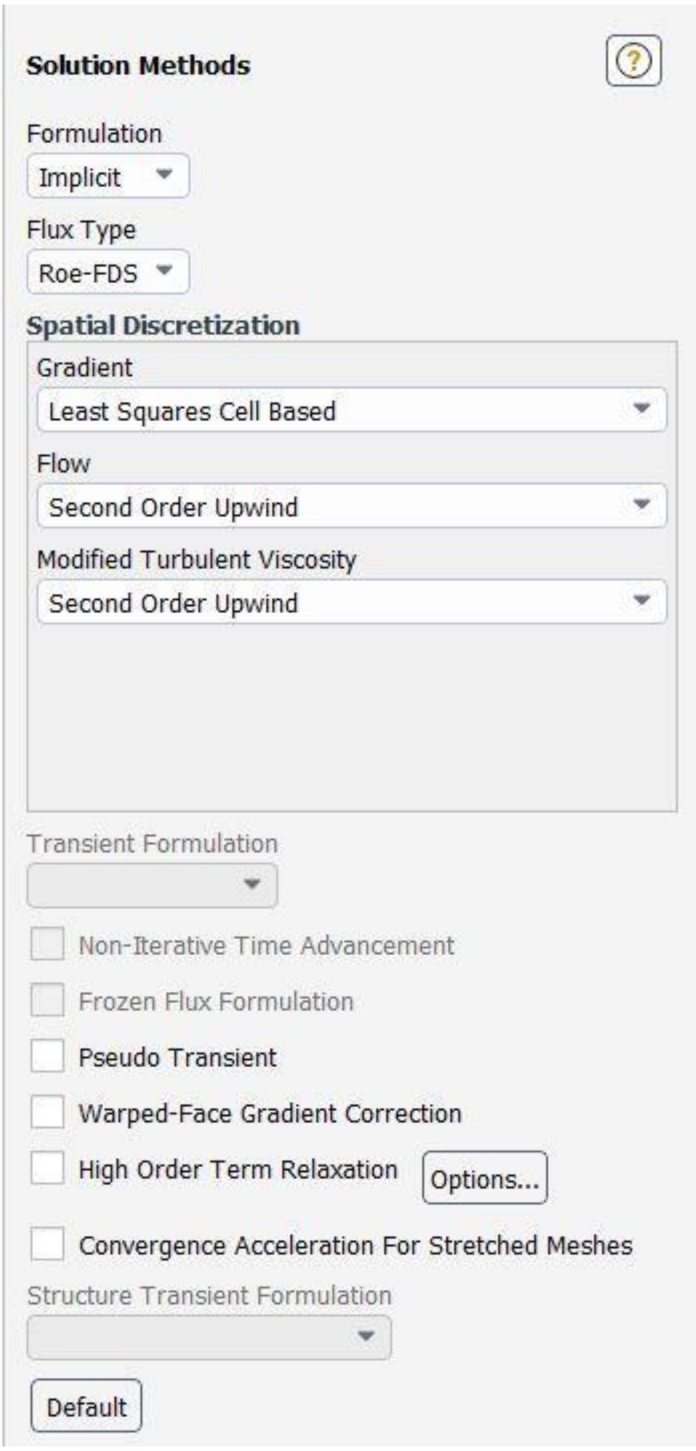
5. Reference values don't affect the results, these are used to non-dimensionalize quantities like drag force which results into drag coefficient, skin friction coefficients, heat transfer coefficient etc. these values are used only in postprocessing. For this study we were interested in drag coefficient. In reference values we changed area and length, other quantities were captured from pressure far field. We used surface area without base. One can use frontal area also, since drag force will have same value, drag coefficient can be justified by ratio of areas.

Reference values

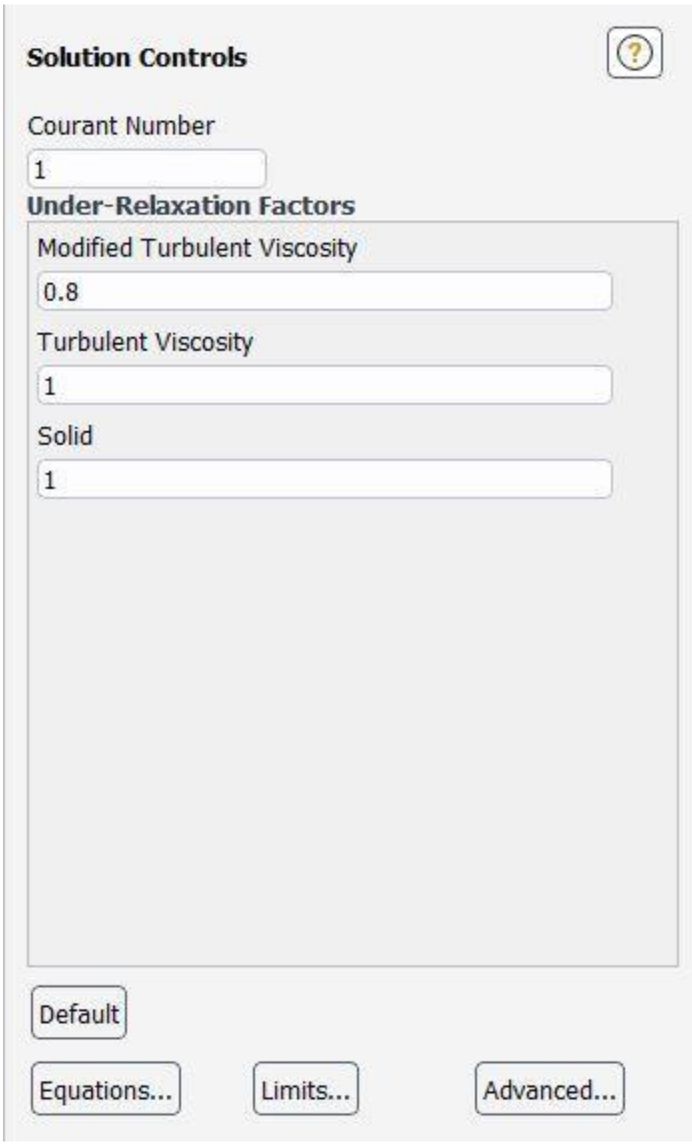


6. Accuracy of solution depends on the solution methods. Implicit scheme was used because it is unconditionally stable and order of other parameters was changed from first to second. In solution control menu courant number and relaxation factors were kept unchanged. At default courant number solution is stable in our case. Increasing courant number would reduce computational time but formulation can become chaotic. That's why we kept courant number 5.

Solution method

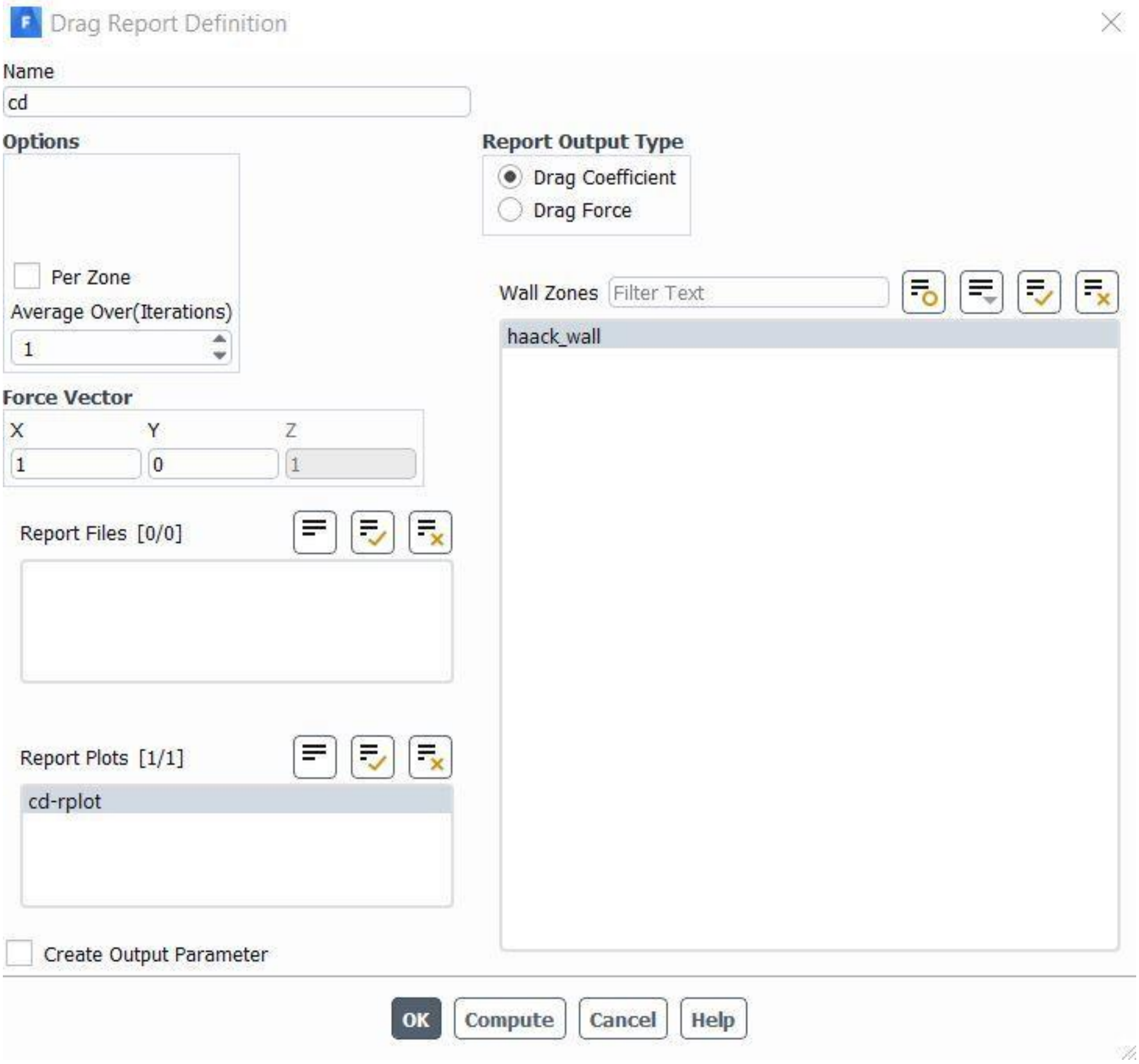


Solution control



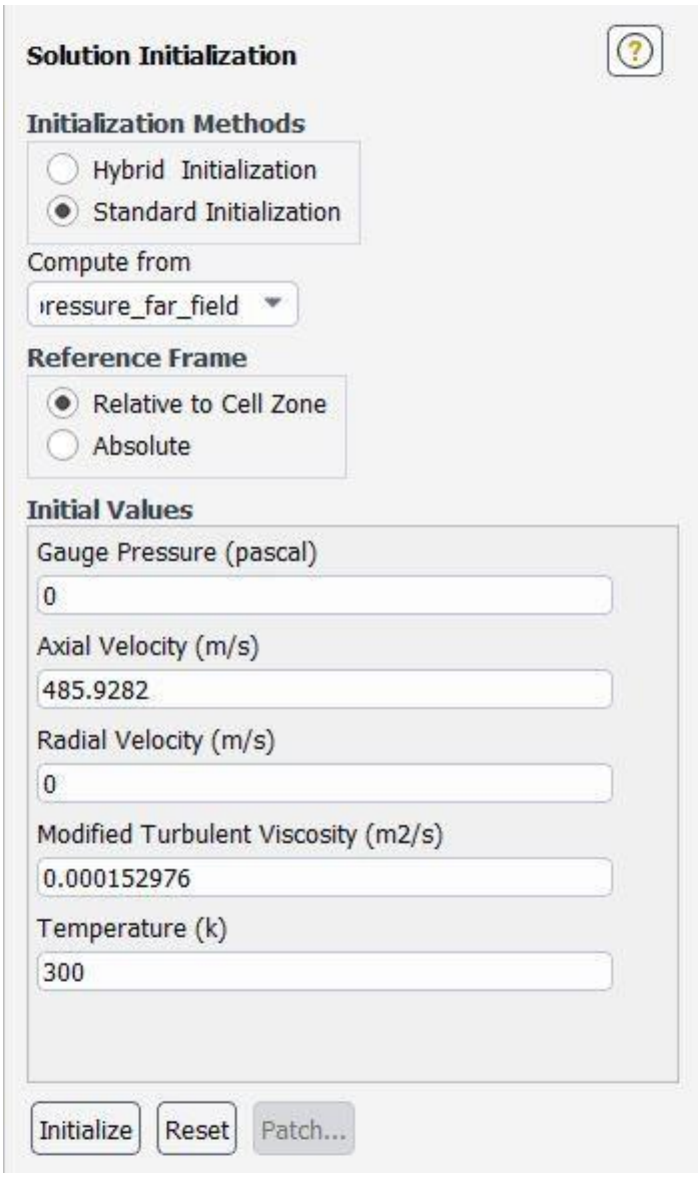
7. We created drag report to have an eye on drag coefficient plot and to get exact value of drag coefficient for analysis.

Report definition



8. We chose standard initialization, which guesses initial values from given boundary conditions. We chose pressure far field to compute initial values. We had to give number of iterations only, because it was a steady state simulation. No changes were made in convergence criteria.

Initialization



6. Results –

Drag coefficient at various mach number for different designs

Nose cone model/Mach number	**1.4**	**1.8**	**2.4**
Cone	0.084	0.060	0.043
Ogive	0.060	0.045	0.033
LD Haack	0.057	0.043	0.031
LV Haack	0.053	0.046	0.033
Power series (Parabola)	0.058	0.043	0.031
Power series (3/4)	0.070	0.050	0.035

The present results indicate that the Von Karman (LD-Haack), LV-Haack and power series parabola noses have the lowest drag over most of the Mach number range. Power series (3/4) had high drag at initial Mach number but drag coefficient is decreasing is decreasing drastically with increase in Mach number, so it can be preferred at high Mach number. Cone nose is worst among them.

Impact of fineness ratio

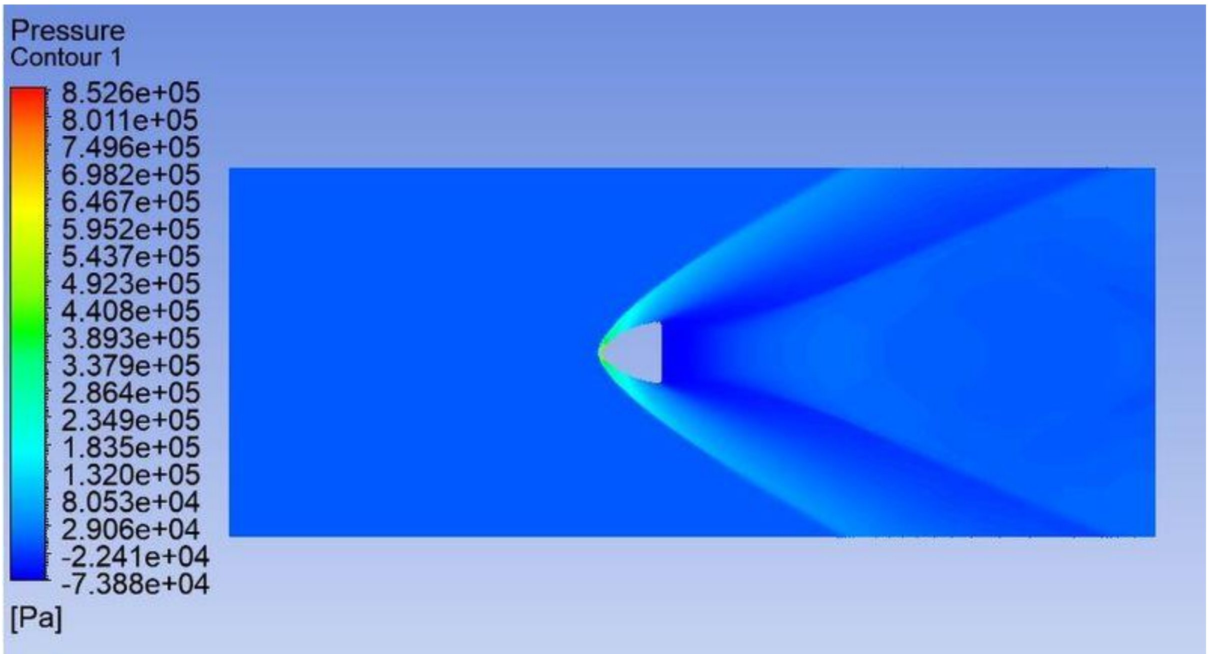
Fineness ratio/ Mach number	1.4	1.8	2.4
1	0.2820	0.2462	0.2086
2	0.0832	0.0625	0.0452
3	0.0530	0.0462	0.0331
4	0.0384	0.0278	0.0200

In phase2, we proceeded with LV-Haack nose cone with varying fineness ratio. At supersonic speeds, the fineness ratio has a very significant affect on nose cone wave drag, particularly at low ratios, but there is very little additional gain for ratios increasing beyond 5:1. as the fineness ratio increases, the wetted area, and thus the skin friction component of drag, is also going to increase. Table data proves it.

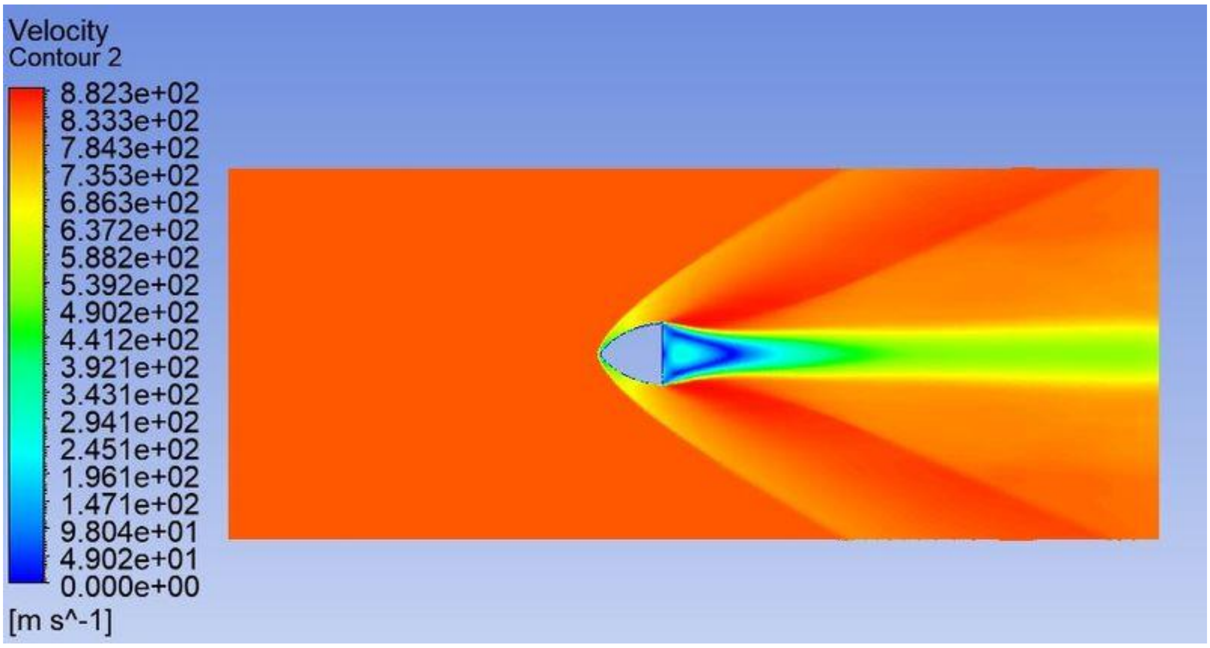
Pressure and velocity contours are also pasted to see the shock wave and minimum pressure areas. In phase one fineness ratio is 3 so pressure and velocity contours will be more or less same, so we presented pressure and velocity contours of phase2 cases only.

Fineness ratio - 1

Pressure contour

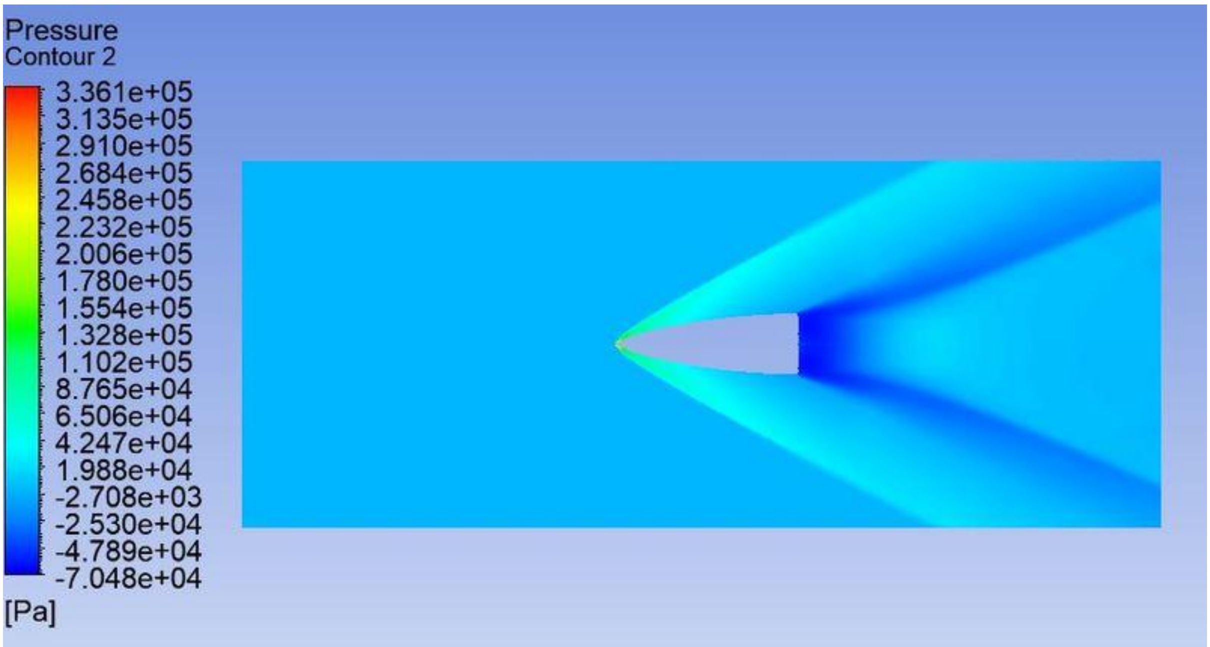


Velocity contour

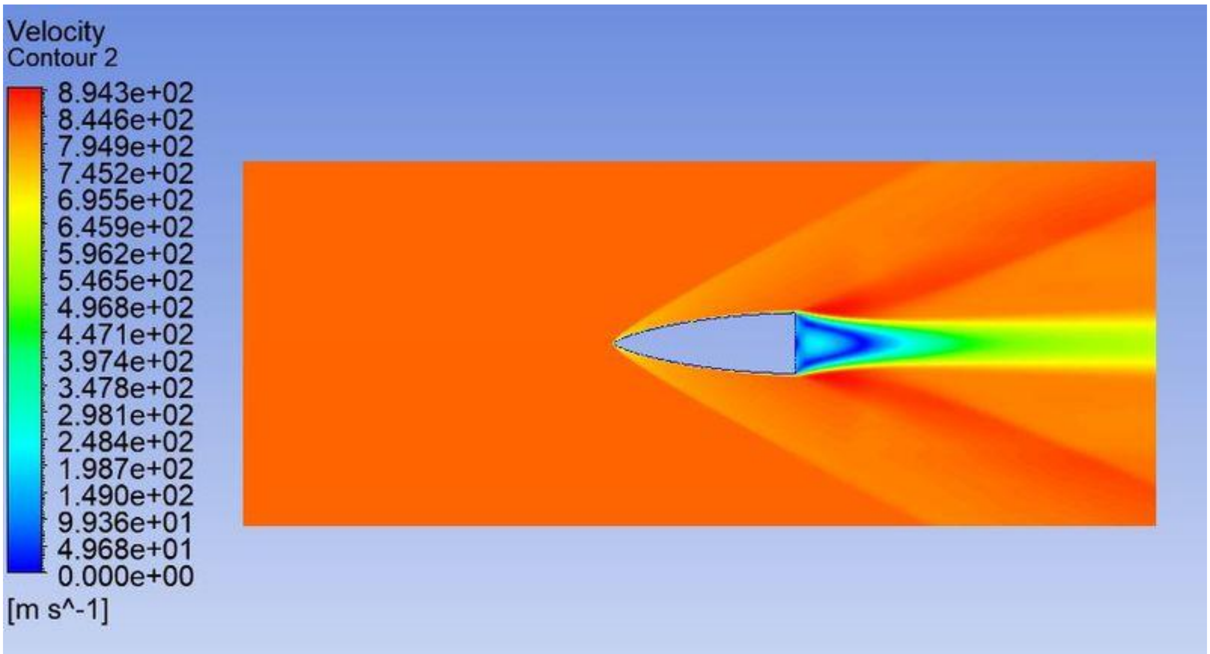


Fineness ratio - 2

Pressure contour

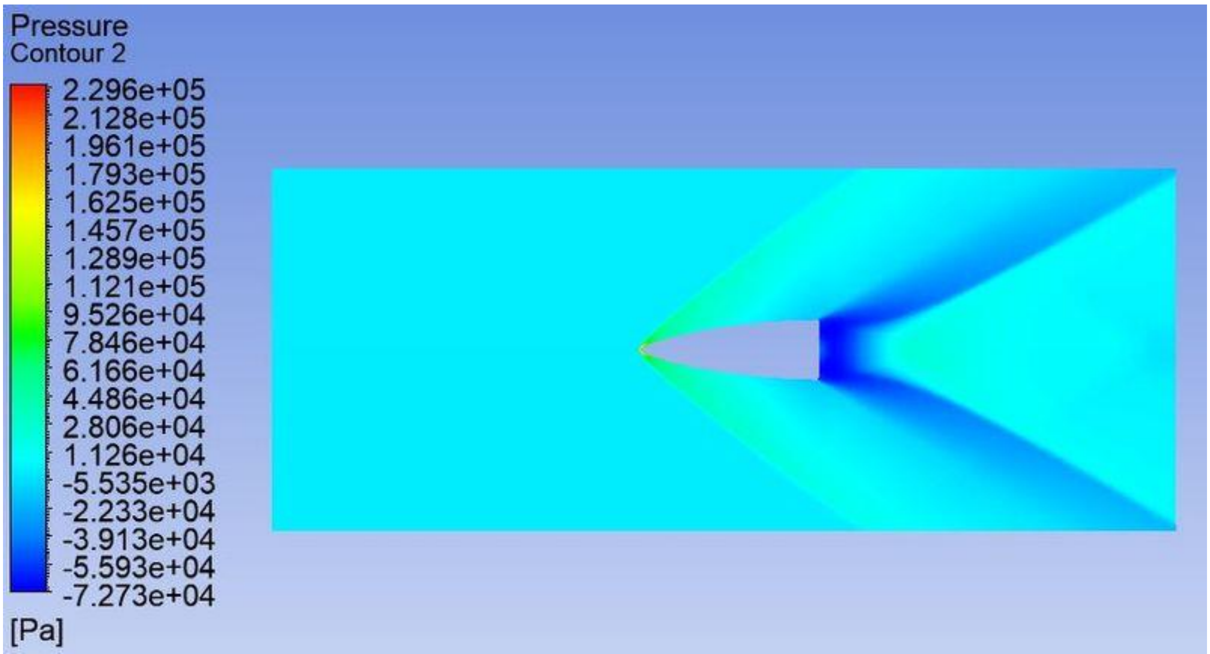


Velocity contour

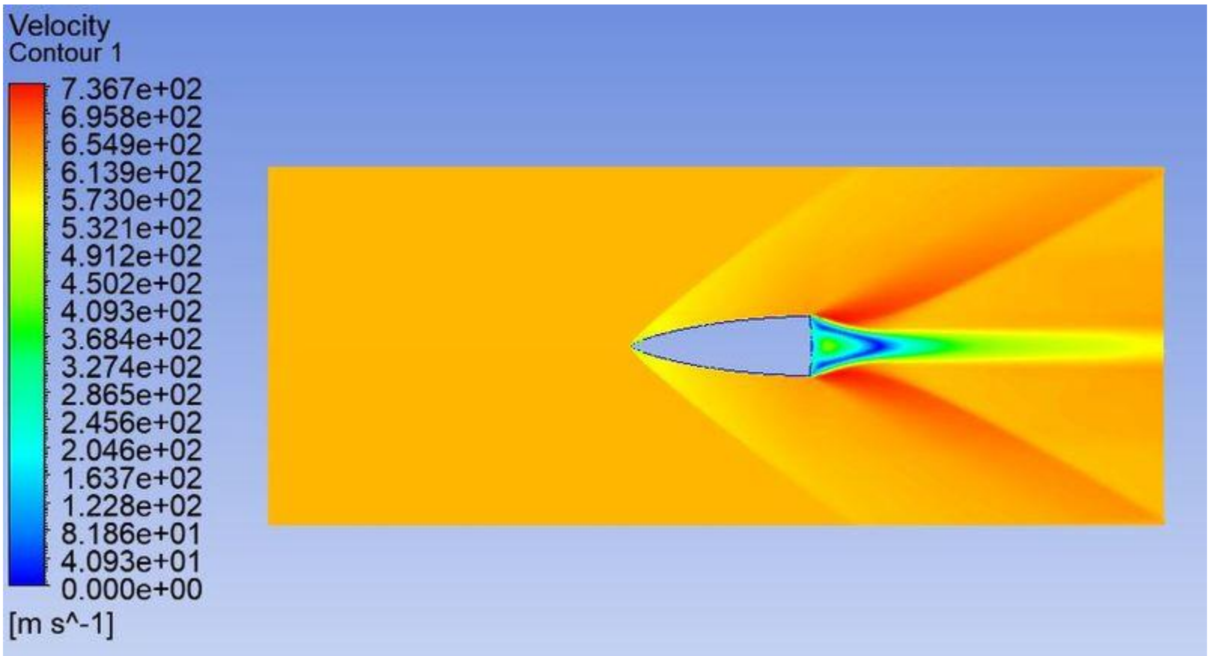


Fineness ratio - 3

Pressure contour

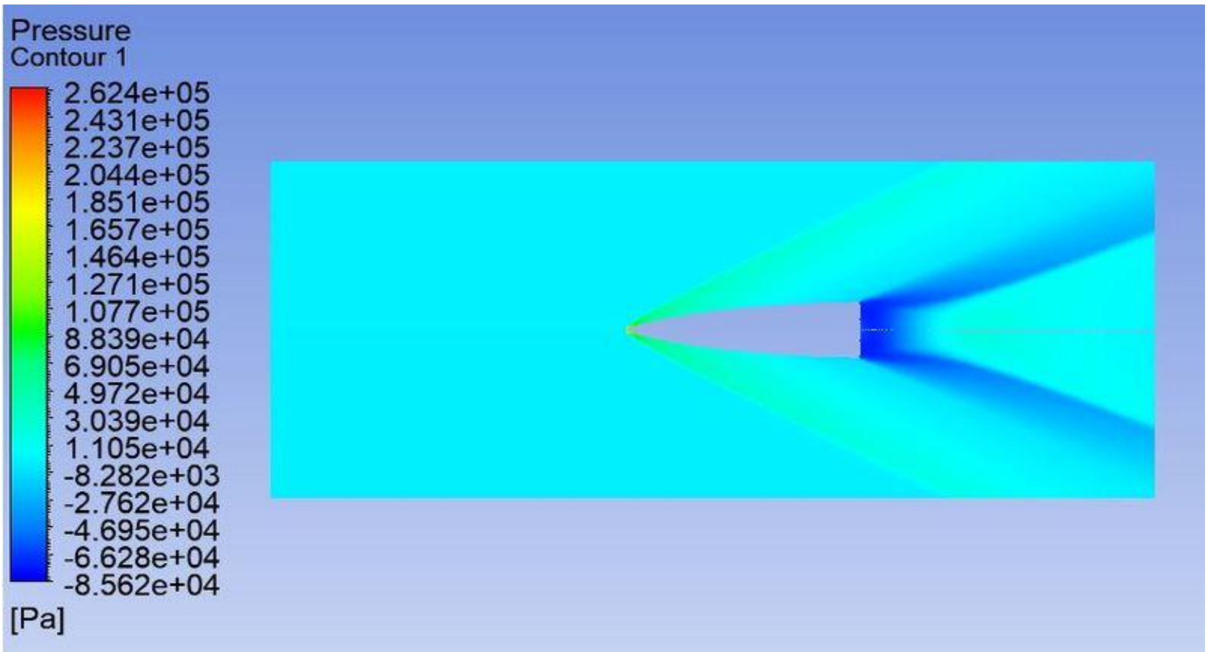


Velocity contour

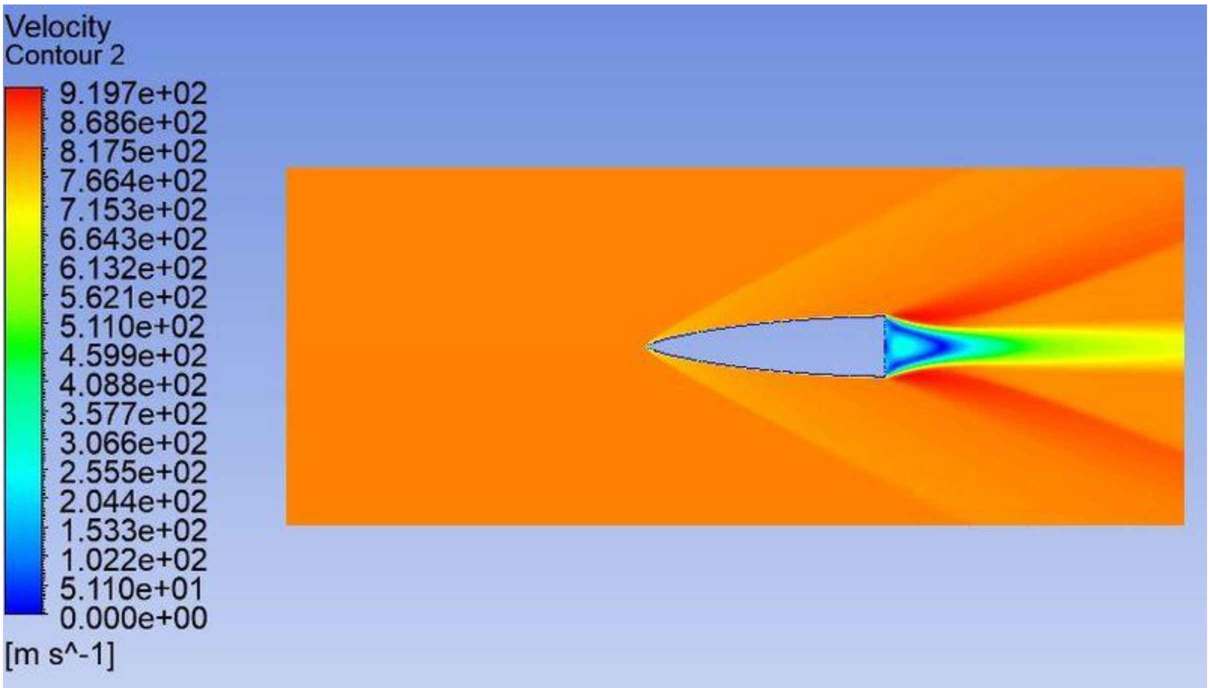


Fineness ratio - 4

Pressure contour



Velocity contour

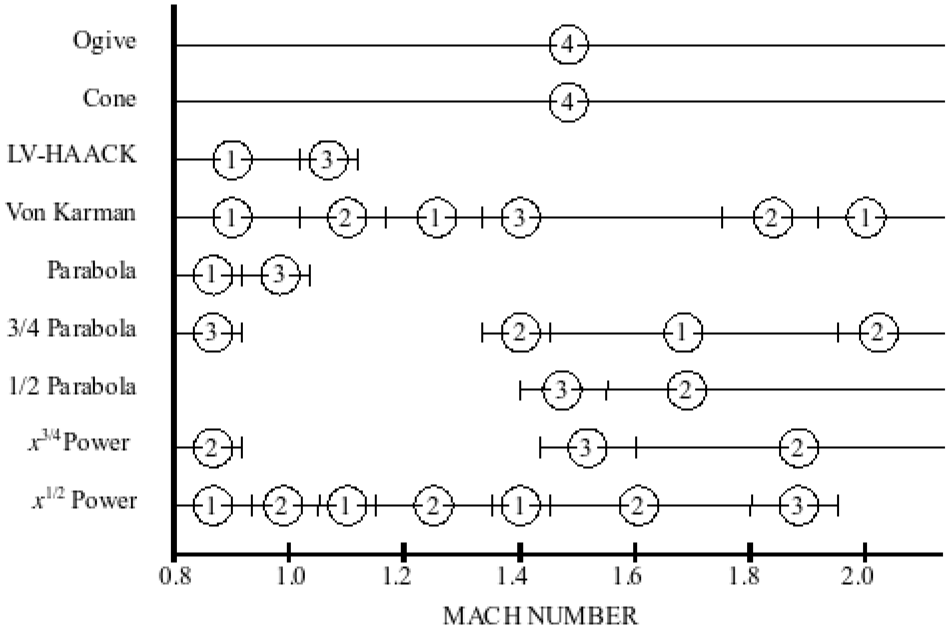


8. Validation -

Errors and uncertainties can arise whilst performing a numerical simulation. These can be due to the improper modelling of physics such as a misunderstanding of the phenomena leading to falsifying assumptions or the incorrect computational design such as making wrong approximations and simplifications about the parameters that govern the fluid dynamics. For a CFD solution to be credible, it requires a detailed analysis to be performed, to quantify the modelling and numerical uncertainties in the simulation. Verification and validation procedures are the means by which a CFD solution can be properly assessed through quantitatively estimating the inherent errors and uncertainties.

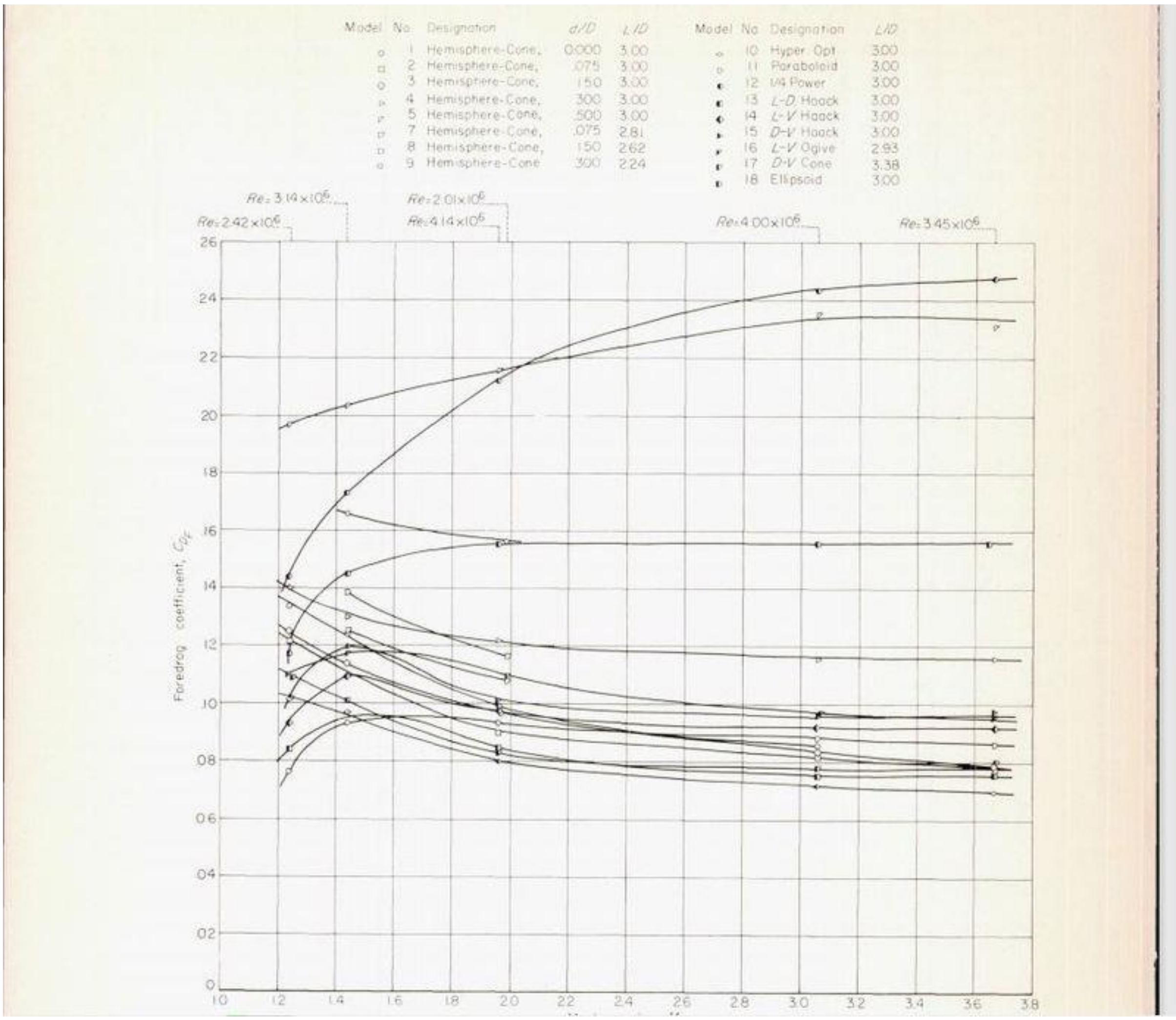
Validation can be performed in two ways analytically and experimentally. Analytically means find parameters using mathematical equations which governs the phenomena and experimental validation can be done by setup the physical model. Respective to this study, validation is not possible, because analytical way is too complex and experimental way is very costly, we need to setup a wind tunnel for it. There is no way to check the exact values of drag coefficient, but we can compare the type of design and their position in table of superiority. From following image, we can easily validate our results which tells power series specially parabola and haack series-LV haack are best among available options.

Nosecone drag comparison



A graph is also pasted, by which we can get little idea about the drag coefficient value, like where, the values should fall or the range. Although, graph contains the value of fore drag, but we compare these with our results, because fore drag dominates in these types of simulations.

Fore drag coefficient comparison



7. Conclusion –

Various cases simulated with varying Mach number and fineness ratio, but we still unable to pick best one, because some like Haack series was showing minimum drag at initial Mach number and power series seems to possess low drag at high Mach number. In general, we can recommend Haack series for low Mach number and power series for high Mach number. By increasing fineness ratio we definitely got reduced drag, but there is limitation we cannot send needle into space which would be fineness ratio 10:1 or more and more importantly after fineness ratio 5:1 there is negligible affect of it at high Mach number. best pair can be choosed to design nosecone.