Analysis of The Fluid Dynamics of A Reentry Capsule Using Computational Methods

Gopalakrishnan Dharmaraj Ms
Dept of Aerospace Engineering
University Of Hertfordshire, Uk.

Abstract- *The field of space science plays an important part in the expanding technology of the contemporary period, thanks to the enormous contribution it has made to the growth of humanity. The advancement of science and technology, including rocketry, has made space travel more effective. However, in today's day and age, nations are keener than ever to send humans into space to carry out research under the guise of the word "Astronaut." The journey into space on a rocket vehicle is very vital, but just as crucial is the astronauts' safe return to earth in reentry capsules after completing their mission.*

This project involves the design of a conceptual design for a REENTRY vehicle as well as the performance of a hypersonic simulation to determine the velocities, pressure, wall stress, vorticity, and temperature in order to analyses the performance of the REENTRY vehicle along with the changes in angle of attack.

The conceptual design of the REENTRY vehicle is evaluated with various densities, angles of attack, drag with enhanced value in case of Landing in General, and temperatures dependent on altitude.

I. INTRODUCTION

Utilizing various mathematical formulations, I have conceptually created REENTRY VEHICLES for the purpose of my scientific study, and these vehicles have a wide range of diverse configurations. After that, I investigated the REENTRY VEHICLE's behavior at hypersonic speeds by using the computational fluid dynamics method. When considering the amount of drag that is produced by each individual REENTRY VEHICLE form, comparisons are conducted between the various REENTRY VEHICLE shapes. The objective is to determine which profile produces the HIGHEST amount of drag as a result of the REENTRY vehicle's journey into the earth's atmosphere. In this portion of the simulation, the various positive angles of attack with varying velocities are used to estimate how well it will perform upon reentry into the atmosphere of earth.

II. OBJECTIVES

The goal of this article is to create a REENTRY CAPSULE with a handful of measurement modifications, then perform a numerically based simulation to calculate its pressure, drag, and wall stress along with a variety of angle of attacks and velocities. Finally, compare each of these parameters against each other in order to discover which values are the highest achievable. After that, the specific design may be simulated with different temperatures to examine how well it performs during REENTRY into the atmosphere of the earth, and it can also be simulated with vorticity to evaluate how the flow behaves while travelling at hypersonic speeds.

III. METHODOLOGY

My project entails the creation of a hypothetical REENTRY CAPSULE using the 3D modelling platform Solid works, with a handful of modified Measurements.

The 3D cad model saved in IGS format is first loaded into the CFD simulation programme known as starccm+.

After that, the domain for the reentry simulation is constructed.

The generation of polyhedral grids on both the domain and the REENTRY CAPSULE comes next after the meshing procedure that was just completed. Before beginning to generate the mesh, it is necessary to allocate the subtracted domain and the capsule to the region.

After the meshing process is complete, the physics continua for the model need to be carefully selected according on the kind of flow and the way it acts.

Then, under the area heading, the borders need to be given in the appropriate manner.

After that, scenes have to be crafted depending on the necessary characteristics in order to see the contours.

Following that, the report that will depict the drag force for the simulation has to be developed.

The simulation should then be conducted for the number of iterations that were requested, which is the very final step.

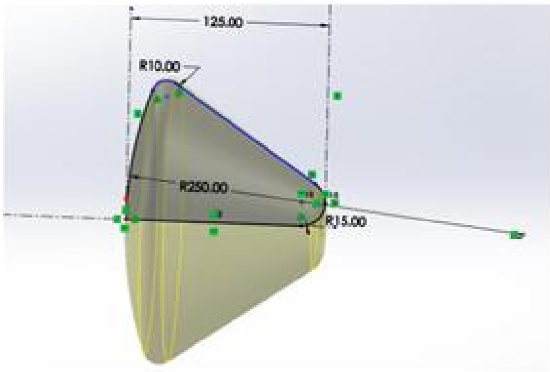


FIG 1: REENTRY CAPSULE DESIGN 1

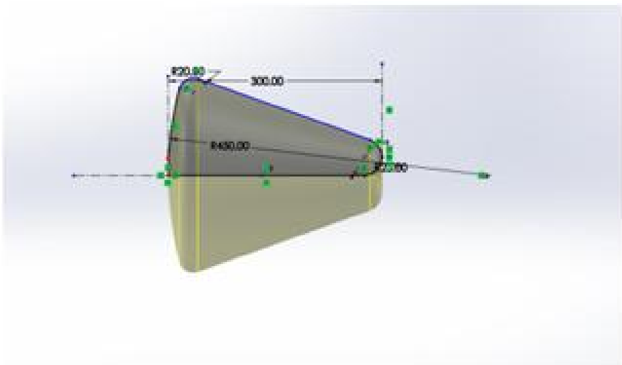


FIG 2: REENTRY CAPSULE DESIGN 2

The two designs that were developed in solid works and saved in igs format are shown in the figures that have been presented above. After that, the two models are brought in one at a time for the simulation approach that was described before in the section under "Methodology." After that, the drag of each design is measured to determine which model provides the maximum drag so that it may be used in further simulations.

TABLE 1: SIMULATION RESULTS OF TWO DESIGNS.

MODEL	DENSITY (kg/m^3)	AOA' (deg)	VELOCITY (m/s)	DRAG (N)
FIG 1	0.0000338	0	8240	26.47468487227219
FIG 2	0.0000338	0	8240	54.07997798231245

Because the vehicle's thrust will be somewhat less than the drag in this scenario, the reentry capsules that carry people back to earth after the mission is over are clearly the ones with the maximum drag, as I said before. Therefore, the design that produces the greatest amount of drag force is selected for further simulation.

POST PROCESSING

The pre-processing step and the post-processing stage are both started with various parameter settings after the outcome of two distinct simulations with redesigned geometries that used 0 AOA. Then, in each simulation, the process is repeated, and once again, the drag value of each simulation is obtained and compared with the values of the other simulations. Before moving on to the post-processing stage, the preprocessing stage must first be completed. This stage is known as "Meshing," and it is the process of generating grids based on measurements. These grids are then generated on the cad model, which is based in the domain, in order to visualize the contours and see the flow patterns.

TABLE 2: PARAMETERS USED FOR REENTRY CAPSULE TO CHECK THE PERFORMANCE.

ALTITUDE (km)	VELOCITY (m/s)	MACH	DENSITY (kg/m^3)	TEMPERATURE (k)	
76.8	8.24	28.6	3.38e-5	205	→ TYPE 1
66	7.07	23.2	1.53e-4	231	→ TYPE 2
58.2	5.62	17.6	4.14e-4	252	→ TYPE 3

VELOCITY CONTOUR

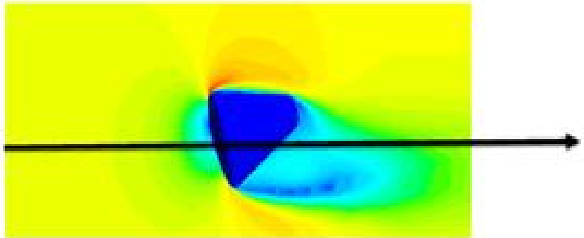


FIG 3: VELOCITY CONTOUR OF 18 AOA'

When the flow passes over the model that was positioned in the domain via the inlet, it will behave as free stream velocity, but once it encounters the model, it will behave as stagnation velocity.

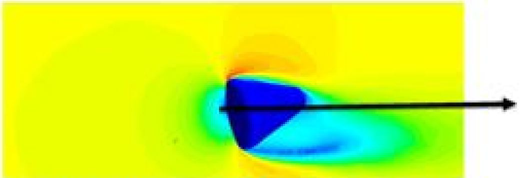


FIG 4: VELOCITY CONTOUR OF 15 AOA'

STAGNATION POINT: When the fluid's velocity is zero, a point of stagnation occurs. The stagnation pressure of an incompressible flow is the sum of the static and dynamic pressures of the free stream. Since they both have the same numerical value, pitot pressure and stagnation pressure are frequently used synonymously.

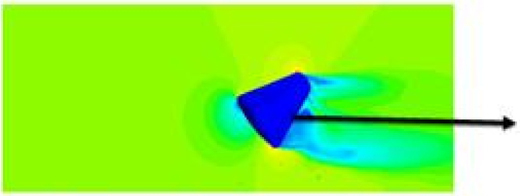


FIG 5: VELOCITY CONTOUR OF 40 AOA'

It is obvious that the angle of attack, which was simulated using two distinct boundary layer separations at 18 and 40 degrees, is important when a vehicle is trying to reenter the atmosphere. When taking drag into account, which is crucial for a gradual, impact-free landing for both the vehicle and the astronauts, this gap is significant. However, my study effort included approximation simulation with the appropriate parameters since the parachute would deploy in response to an altitude control sensor as the reentry vehicle descends, increasing drag. My research indicates that the greatest angle of attack produces the most drag. Therefore, adding a parachute will cause even more drag than the drag produced without one.

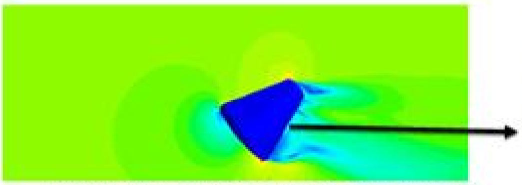


FIG 6: VELOCITY CONTOUR OF 40 AOA'

Here, it is seen that the boundary layer separation velocity at the flat surface of the capsules underneath it exhibits low value separation, which implies the boundary layer separation at that location separates the air particles with low velocity movement. This indicates that the air has not yet reached free stream velocity.

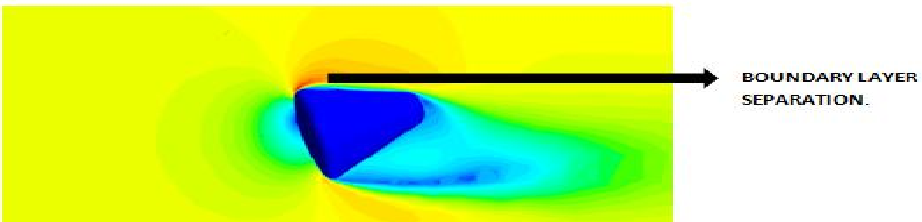


FIG 7: VELOCITY CONTOUR OF 15 AOA'

BOUNDARY LAYER THEOREM

It was mentioned earlier that the body in stationary motion has a velocity of zero, which means that the velocity of the fluid particle will automatically get reduced to zero while it is remaining on the boundary of the solid body. A streamline flow with a velocity is being sent to the stationary solid body, and the fluid particle that will cling on the surface of that solid body will maintain the same velocity as the solid body. However, according to the theory, the particle will not be in a state of rest at any point in time. Within a short amount of time, the particle of fluid flows and shows evidence of velocity gradient. Velocity gradient is the term used to describe differences in velocity in proportion to distance. After passing through the layer, the velocity will return to its free-flowing condition and continue on its journey. These alterations to the velocity will only be maintained for a brief period of time inside the layer. The term "boundary layer" refers to the phenomena that takes place when there is a shift in the velocity of particles that are contained inside a thin layer. Within this boundary layer, shear stress is generated for a small amount of time; however, as soon as the velocity reaches the freestream condition, the shear stress will no longer be present and will be equal to zero.

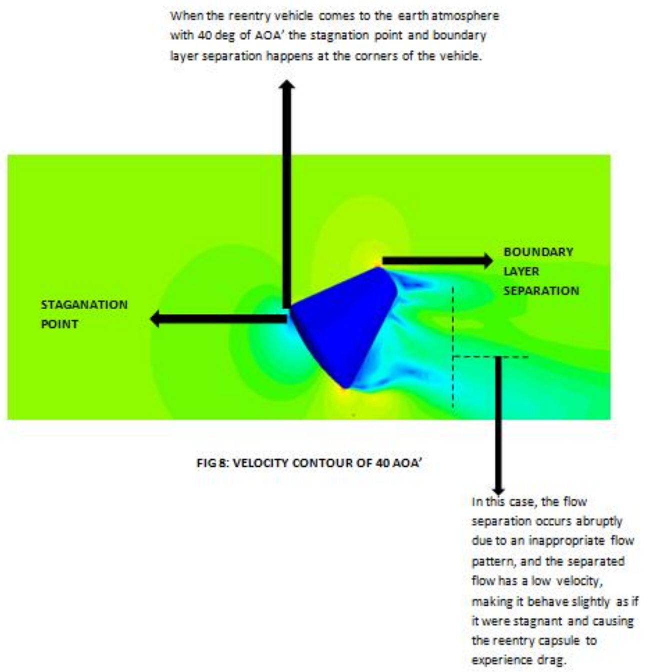


FIG 8: VELOCITY CONTOUR OF 40 AOA'

Following the completion of the vehicle's velocity examination, parallel checks of the pressure and a great number of other parameters have been carried out in order to evaluate the functioning of the capsule during its reentry.

PRESSURE CONTOUR

The ratio of the force that is exerted to an item over its whole area is the definition of pressure, and the unit of pressure is Pascal.

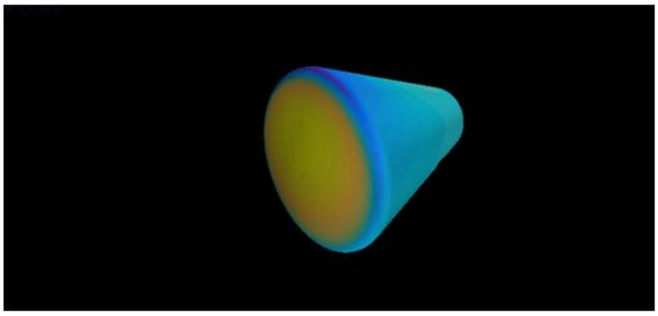


FIG 9: PRESSURE CONTOUR OF 15 AOA'

The blunt body of the capsule is designed with a special ablative materialled heat shield that will burn while the capsule is returning to earth to save the command module. This is the reason why the blunt section of the capsule shows the highest pressure on the color chart. The obvious reason for this is that while the capsule is descending at a high Mach number, the vehicle is travelling through compressible air, and the blunt body shows the highest pressure. However, the heat that the capsules experience in front of it will remain ahead of it because to the blunt form, which causes bow shock and provides a considerable deal of resistance to the heat that is created by high-speed flight in compressible air.

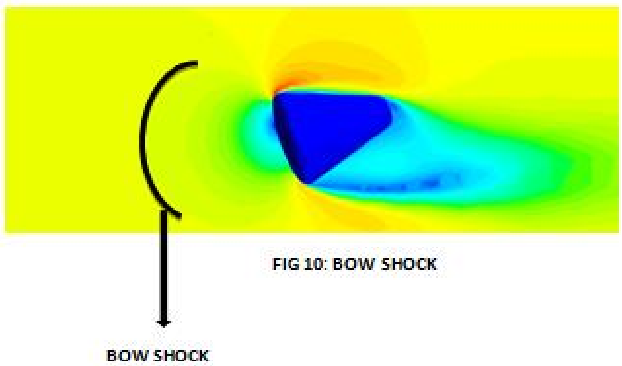


FIG 10: BOW SHOCK

A nonlinear phenomenon known as the shock wave generating a continuous pattern around the body happens when an oblique shock is likely to emerge at an angle that cannot remain on the surface. In this scenario, the shock cannot stay on the surface. This pattern is the result of the shock wave being unable to stay on the surface for an extended period of time. **Bow shocks** are the name given to this particular kind of shock.

WALL SHEAR STRESS

The shear stress that exists in the layer of fluid that is immediately next to an object's wall is referred to as "wall shear stress."

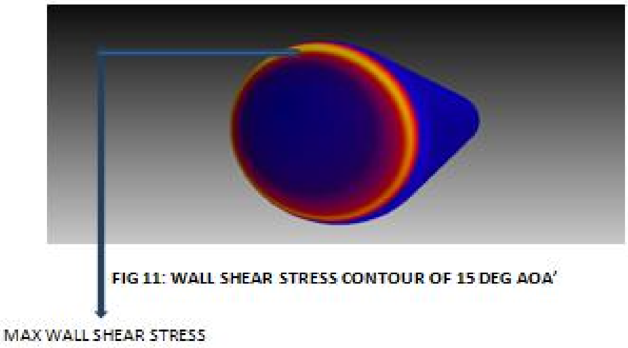


FIG 11: WALL SHEAR STRESS CONTOUR OF 15 DEG AOA'

Wall shear stress contour is seen in Figure 11. As illustrated in Fig. 11, the wall shear stress reaches a maximum value of 307 Pa in the nose-cone area when a vehicle approaches at 15 AoA' with 24 times the speed of sound. The flow transition over the body's downstream causes the most stress to be seen in the nose cone area. The wall shear stress can be calculated by a formula,

$$\tau = F/A$$

SCIENTIFIC NOTATION:

τ = SHEAR STRESS
F = APPLIED FORCE
A = CROSS SECTIONAL AREA.

ENTHALPY

A thermodynamic system's energy is quantified using a quantity called the enthalpy. The entire amount of heat that is contained in a system is equal to the quantity of enthalpy, which is equal to the sum of the system's internal energy as well as the product of the system's volume and pressure. The formula used to calculate the enthalpy is,

$$H = E + PV$$

SCIENTIFIC NOTATION

E = Enthalpy
H =HEAT
P = PRESSURE
V = VOLUME.

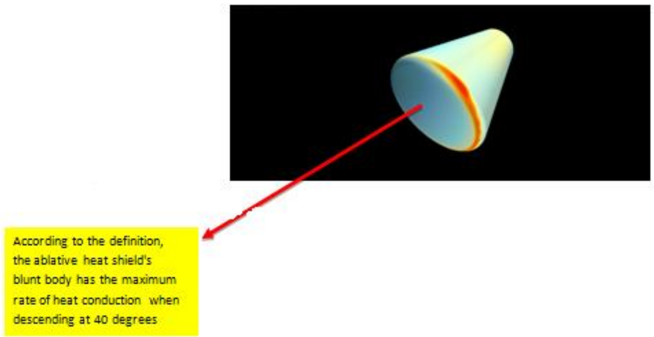


FIG 12: ENTHALPY CONTOUR OF 40 DEG AOA'

CALCULATION

PRESSURE

As I mentioned before the pressure is equal to the ratio of force per the unit area and the formula to calculate pressure is,

$$P = F/A$$

SCIENTIFIC NOTATION

P = PRESSURE
F = FORCE
A = AREA

The chart of atmospheric properties indicates that the pressure is **3.55998E+4** at an altitude of 8 km/hr.

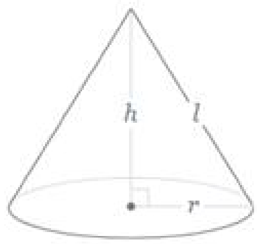


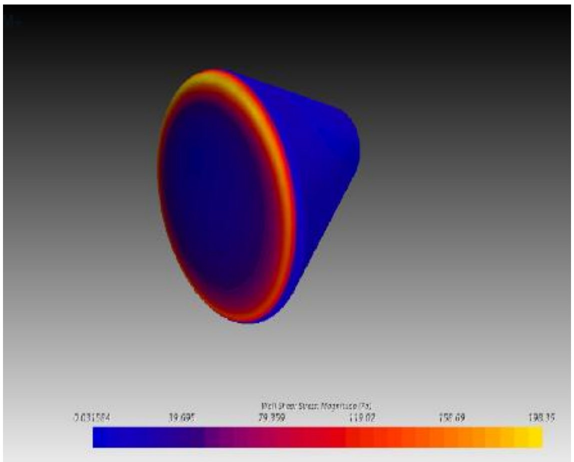
FIG 13: RIGHT CIRCULAR CONE (courtesy: byjus)

AREA OF RIGHT CIRCULAR CONE,

$$A = \pi r\,(r + \text{sq.rt}(h + r))$$

HEIGHT OF CAPSULE = 132.5 mm.
RADIUS OF THE CONE = 450 mm.
Area of the capsule is 1.3 $*10^6$ mm^2
3.55998E+4 = F / 1.3 $*10^6$
F = 4.98 $*10^{10}$ N.

WALL SHEAR STRESS,



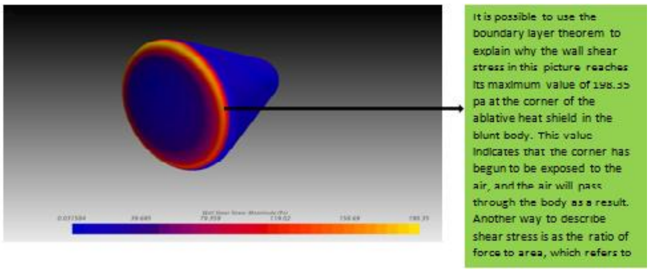
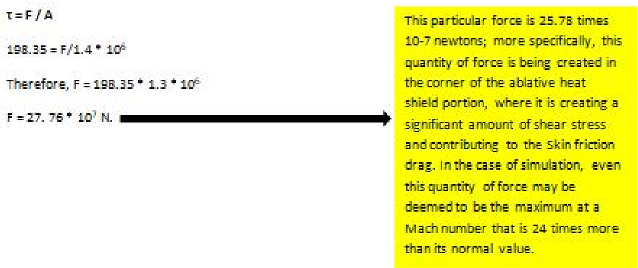


FIG 14: WALL SHEAR STRESS CONTOUR OF 18 DEG AOA'



The **skin friction drag** can be calculated by,

$C_f = \tau w / 0.5 * \rho * v^2$
C_f= SKIN FRICTION COEFFICIENT.
τw = SKIN SHEAR STRESS ON THE SURFACE OF THE BODY.
ρ = DENSITY OF AIR (kg/m^3).
V = FREE STREAM VELOCITY (m/s).
C_f = 192.35 / 0.5* 0.525168* 24 Mach ^ 2
C_f = 1.18027 * 10^-5.

ENTHALPY

$$H = E + PV$$

SCIENTIFIC NOTATION

E = Enthalpy
H = HEAT
P = PRESSURE
V = VOLUME.

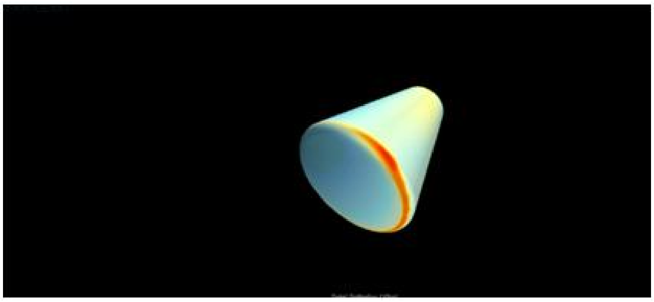


FIG 15: ENTHALPY CONTOUR OF 40 DEG AOA'

2517800 = E + 35599.8 * 2.81*10^7
E = -9.99 * 10^11.

From the standpoint of thermodynamics, it makes no difference to us whether the internal energy is positive or negative; in fact, we don't even know. The only thing that really matters to us is the shift in internal energy, shown by dU. If there is a rise in entropy S, and the temperature T of the system remains the same, then the value of dU will goes up (it will become more positive). So, dU is greater than TdS. If the system increases in volume V, then dU will decrease, which means it will become more negative. Hence dU ~ - PdV.

DRAG

A force will be applied to the surface of the object by the fluid that is flowing around it. A component of this force that operates in a parallel direction to the object is called lift. The drag can be measured in Newton and it's called by a formula,

$$\mathbf{D = 0.5\ \rho\ V^2 S C_D}$$

SCIENTIFIC NOTATION:

1. D= DRAG.
2. ρ = DENSITY OF FLUIDS.
3. V = VELOCITY OF FLUIDS.
4. S = THE SURFACE AREA OF THE OBJECT.
5. C_D = COEFFICIENT OF DRAG.

$$D = 0.5 * 0.525168 * 8000^2 * 1.3 * 10^6 * 1.6 * 10^{-11}$$

D = 350 N.

APPENDIX

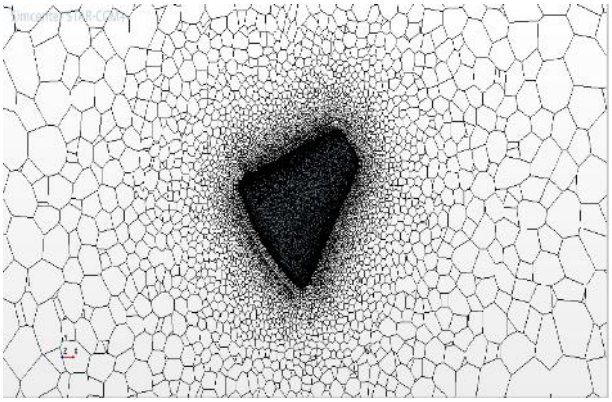


FIG 16: MESHING OF CAPSULE AT 18AOA'

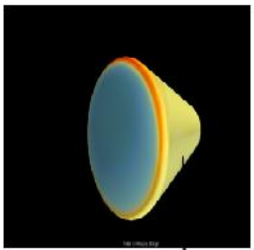


FIG 16: ENTHALPY AT 15DEG

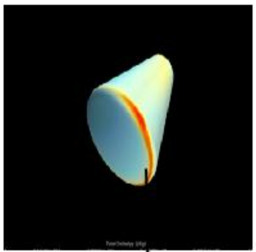
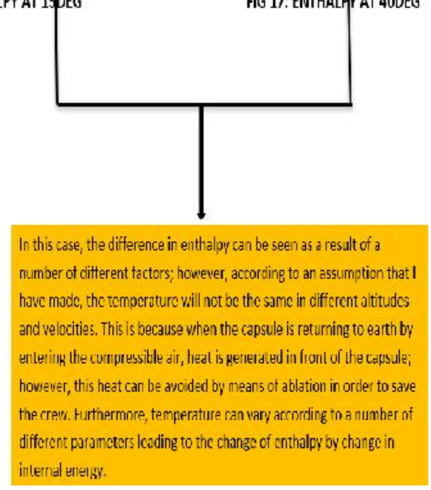


FIG 17: ENTHALPY AT 40DEG



IV. CONCLUSION

To summarize the work that I've done, the reentry vehicle has to be carefully investigated using a wide range of variables, including temperature, thermal conductivity, vorticity, and so on. In spite of this, during the course of my investigation, I came across some information and references pertaining to reentry capsules. As a result of this, I was able to design and simulate a reentry vehicle. In the future, structural analysis may also be performed in order to gain a deeper comprehension of the behavior pattern exhibited by reentry vehicles both inside and outside of the earth's atmosphere.

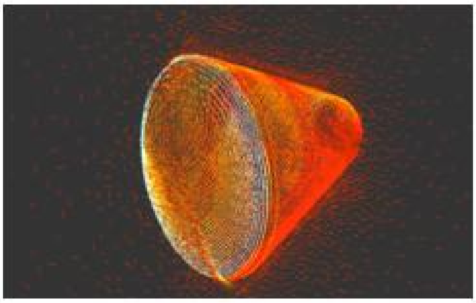


FIG 19: VORTICITY

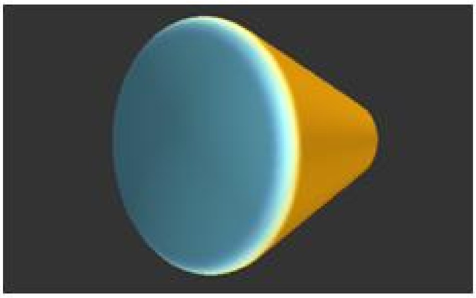


FIG 20: TEMPERATURE

REFERENCES

[1] 1.Terry, J. and Barber, T., 2007. CFD and experimental study of an inflatable re-entry vehicle model at Mach 3 conditions. *Acta Astronautica*, *61*(10), pp.854-865.

[2] Takahashi, Y., 2015. Advanced validation of CFD-FDTD combined method using highly applicable solver for reentry blackout prediction. *Journal of Physics D: Applied Physics*, *49*(1), p.015201.

[3] Viviani, A., Aprovitola, A., Iuspa, L. and Pezzella, G., 2020. Aeroshape design of reusable re-entry vehicles by multidisciplinary optimization and computational fluid dynamics. *Aerospace Science and Technology*, *105*, p.106029.

[4] 4.Longo, J.M., Orlowski, M. and Brück, S., 2000. Considerations on CFD modeling for the design of re-entry vehicles. *Aerospace science and technology*, *4*(5), pp.337-345.

[5] 5.Shafeeque, A.P., 2017. CFD analysis on an atmospheric re-entry module. *International Research Journal of Engineering and Technology*, *4*(01).