CASE STUDY OF DOUBLE-BELL NOZZLE

Mageshan Kp[1]

ABSTRACT

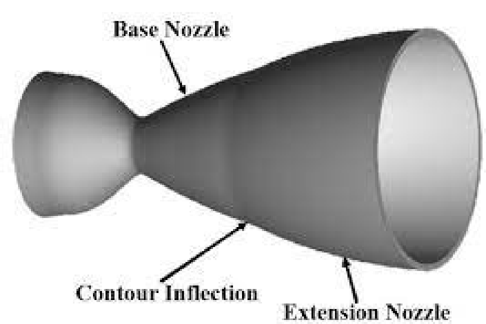
The rocket movement is largely depend on the nozzle type that are used. A rocket nozzle is basically a propulsion nozzle. The main function of this nozzle is to increase the internal pressure of the combustion chamber and the exit speed of the combustion products (gas particles) which helps in the thrust and combustion process of the rocket engine. Therefore selection of optimal design of the rocket nozzle is one of crucial process. In this case study we will see the methodology and results of design of a Double-Bell rocket nozzle. The main rocket design parameters are thrust, chamber pressure, mixture ratio, nozzle area ratio and **nozzle geometry**.

Designing of the Dual-bell nozzle is done in the SOLIDWORKS software and for consistency, a numerical simulation of flows in the obtained dual bell nozzles is achieved using a computer code ANSYS.

INTRODUCTION

The dual-bell nozzle is the most promising concept for altitude compensating nozzle. It operates under both sea-level and high altitude conditions. Its characteristic wall inflection, dividing the nozzle into base and extension, offers a one-step altitude adaptation, without any moving parts. In sea-level mode, the wall inflection forces the flow to separates controlled and symmetrically, dangerous side loads are avoided and due to a smaller effective area ratio the sea-level thrust increases. During ascent of the launcher, at a certain altitude, the nozzle flow attaches to the wall of the nozzle extension until the exit plane. The full (higher) expansion area ratio is used resulting in a higher vacuum performance.

Fig. 1 :Typical Double-bell nozzle



Because of its simple design , compared to other advanced nozzle concepts, the absence of any mechanical part which would increases its weight is one of the outstanding features.

METHODOLOGY

After a detailed study of theoretical approach to the nozzles, a full length Bell nozzle is created using SOLIDWORKS software. The meshing and the analysis of this Bell model is done using the ANSYS software.

The shape and the length of the base nozzle will define the thrust and the flow condition under sea-level mode.

The inflection angle should be at least 5.5° as a value too low will not guarantee the fixation of the separation point at the inflection between sea-level mode and the start of the transition. If the inflection angle is too high (typically for values higher than 25°) the length of the extension will grow exponentially, increasing the mass of the structure, or the wall angle at the end of the inflection will be very high, leading to increased diverging loses.

DESIGN OF A DUAL BELL NOZZLE CONTOUR

During our present work, the calculations are made with a perfect gas. The profile chosen for the implementation of the method of characteristics is the one describing the divergent section of an axisymmetric bell nozzle.

The design of the dual bell nozzle is carried out in two parts:

FIRST CONTOUR (Base nozzle) design :

The throat radius of the nozzle is kept around 0.01m and total exit radius of 33.61mm. A design Mach number (Md) equal to 2.4 is maintained.

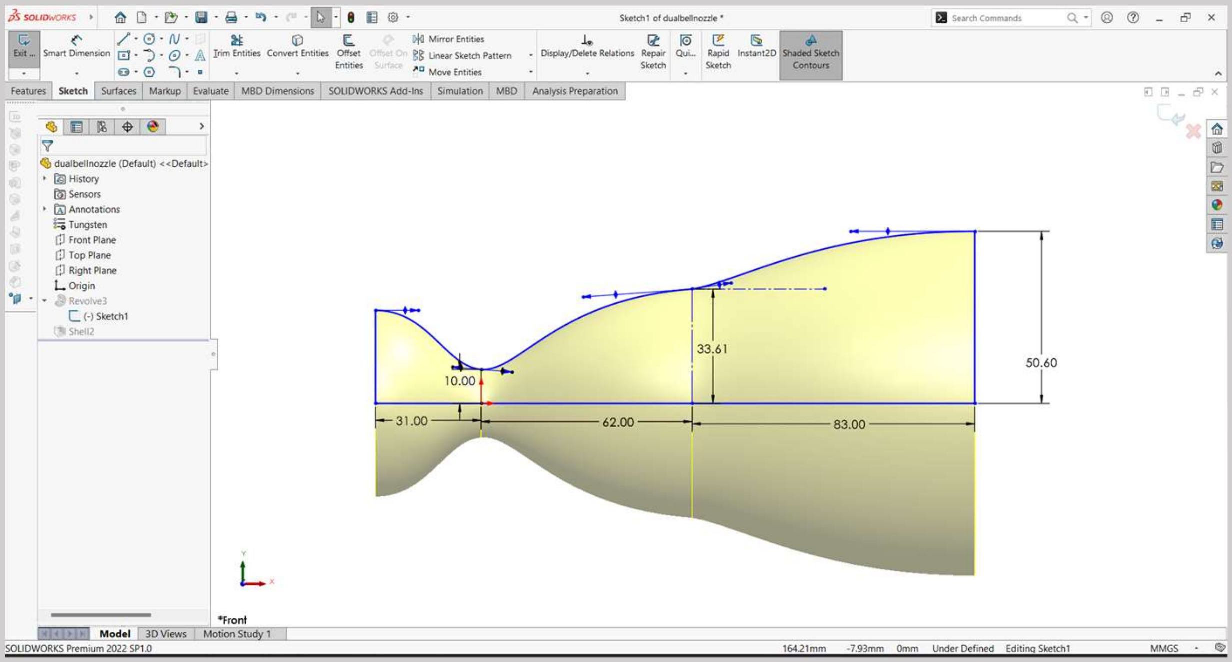


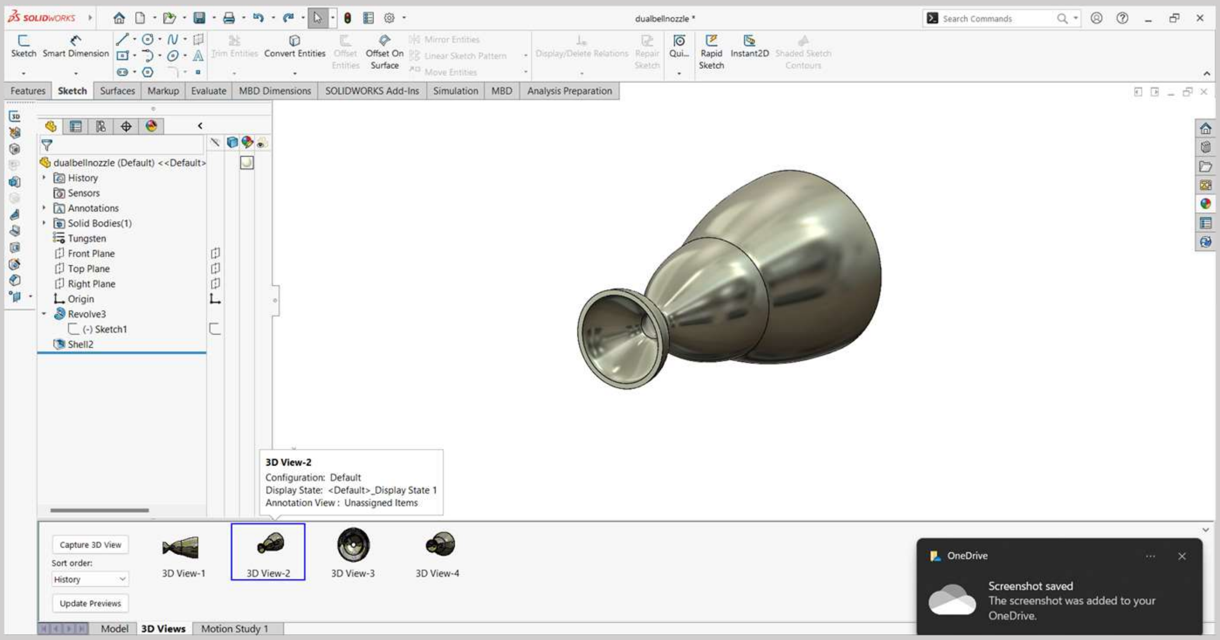
Fig. 2 : dimensioning of the DUAL-BELL NOZZLE:

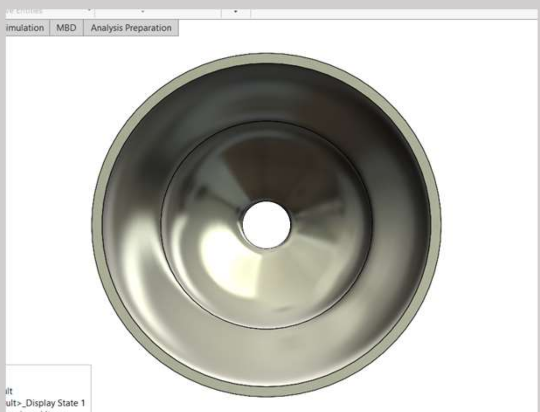
SECOND BELL(nozzle extension) design :

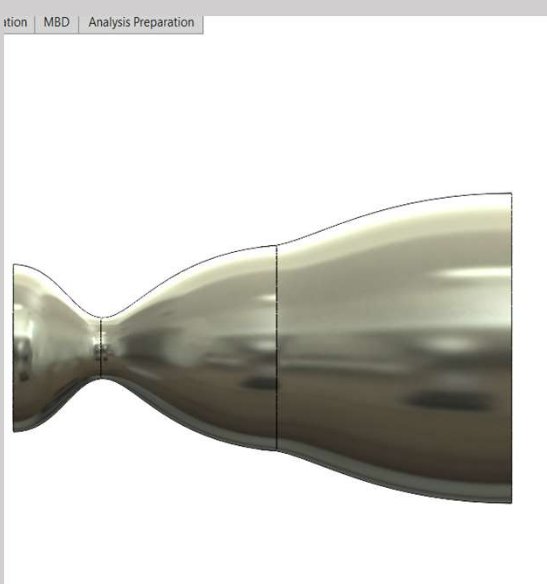
The exit radius of the nozzle extension is 50.60mm and the inflection angle is kept around 9.2 ^{o}

PARAMETERS	SYMBOL	LENGTH
Throat radius	R_{th}	10
Area ratio base nozzle	A_b/A_{th}	11.3
Area ratio extension	A_c/A_{th}	25.6
Base nozzle length	L_b/R_{th}	62
Extension length	L_c/R_{th}	83
Total nozzle length	L_t/R_{th}	145
Inflection angle	α_i	9.2

3D view of the DUAL_BELL nozzle :

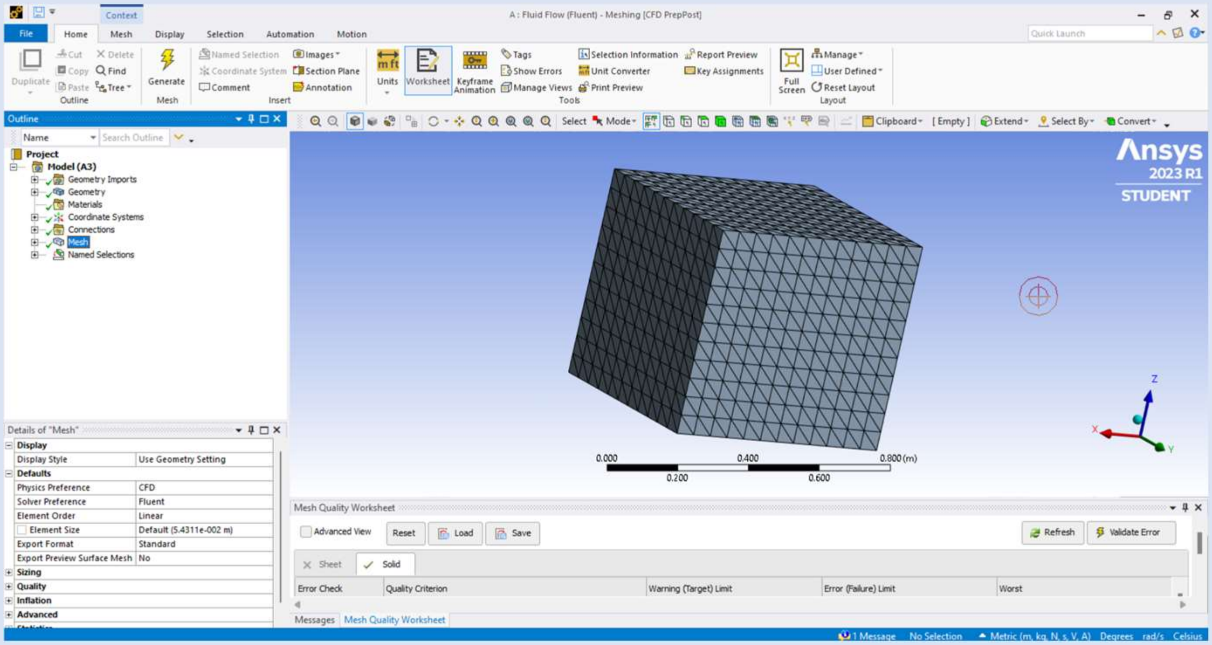




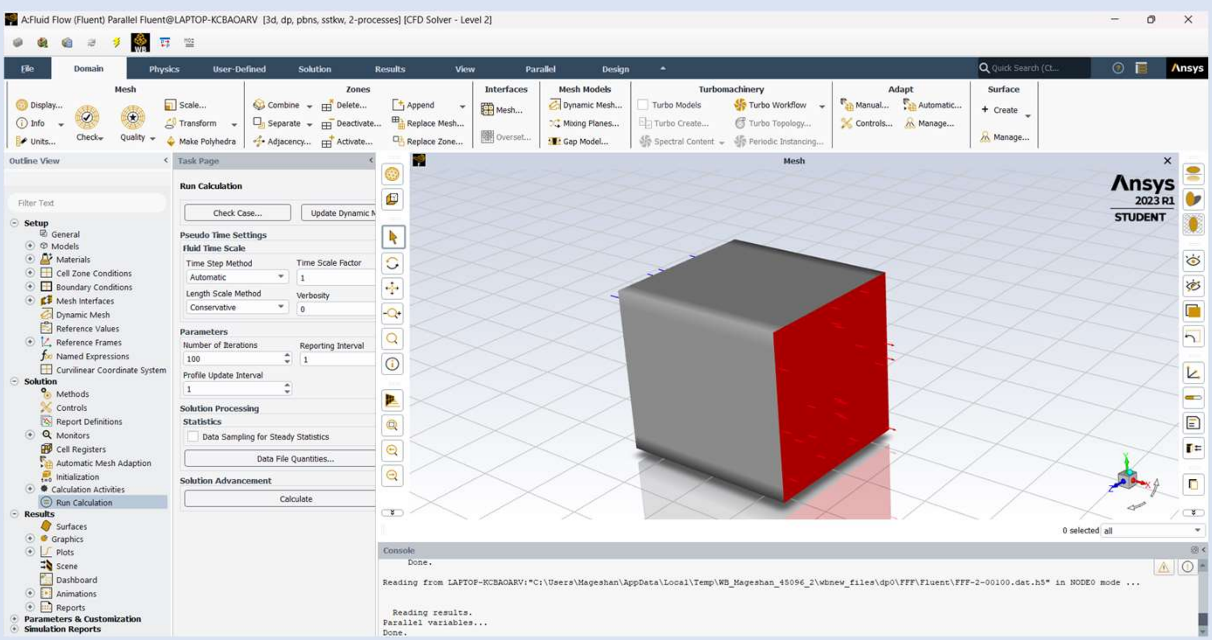


ANSYS FLUENT

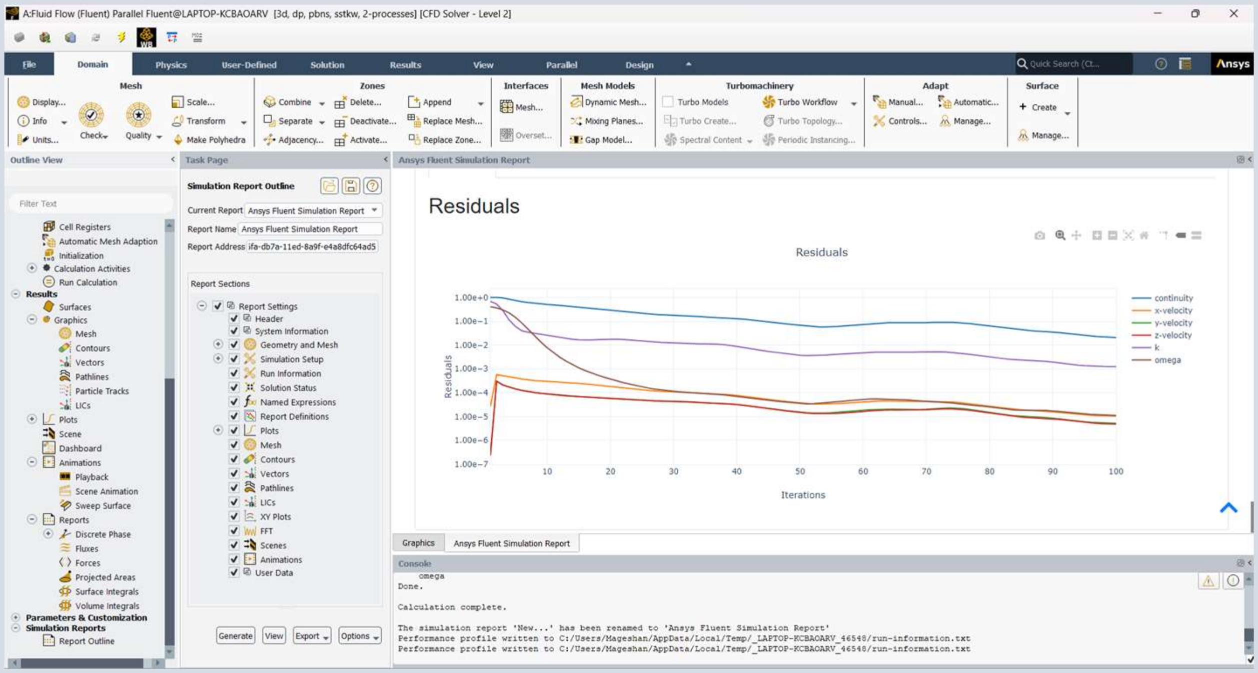
The Solidworks model is then imported in the Ansys fluent for further analysis. Enclosure is created with symmetric dimension of 0.25m and the mesh is created further as you can see in the figure below.



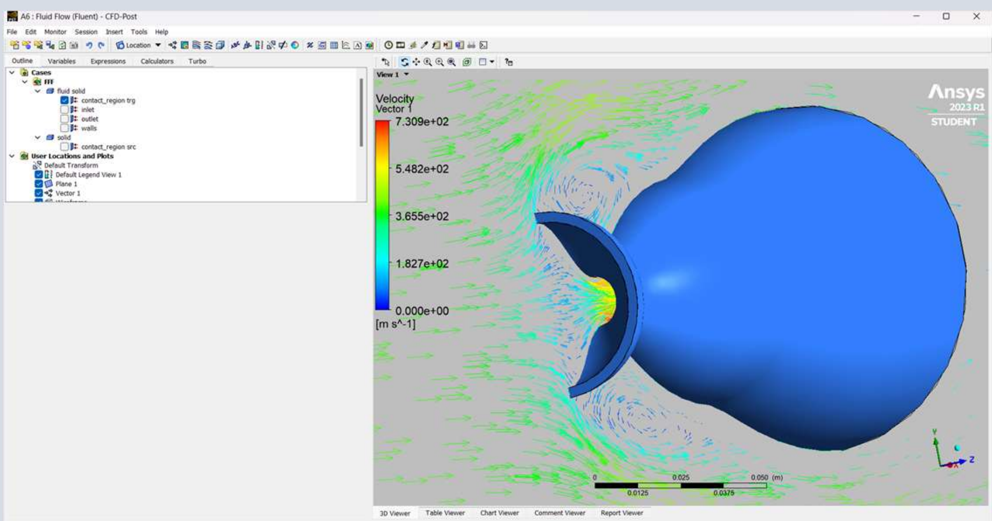
An inlet velocity of 400m/s in given as an input through the direction of inlet face of the cube and initialised the iterations.

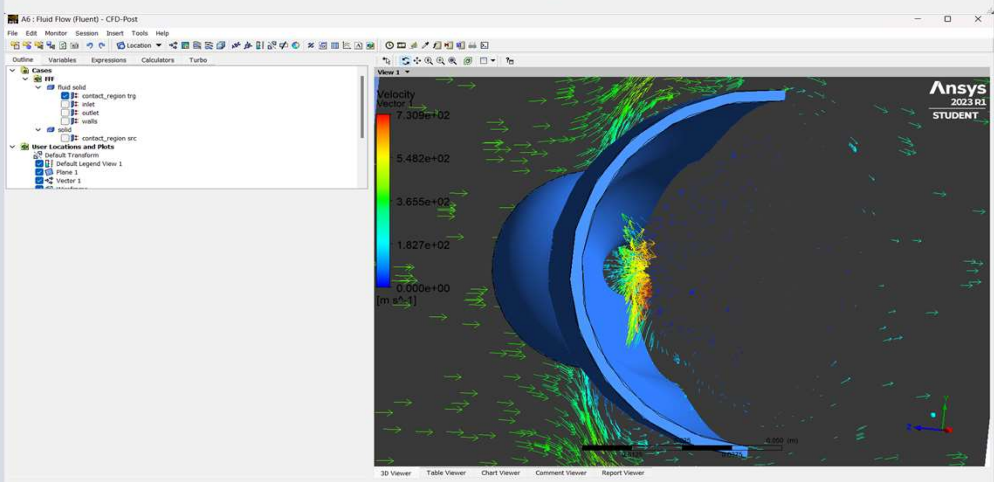


Result of velocity in each direction through 100 iterations.



The trajectory of Velocity vector of the air is clearly shown along the perpendicular plane. We are able to produce speed of 2.1 Mach.





CONCLUSION

Atmospheric pressure restricts the expansion of the exhaust gas at low altitudes so the efficiency is much higher at low altitudes . At low altitudes, a vehicle can saves 25-30% more fuel by using a dual bell nozzle. It is also able to expand the engine exhaust to a larger effective nozzle area ratio, at high altitudes. The dual bell design is suitable for Single Stage to Orbit (SSTO) flight. Better use of the base area, and has higher thrust efficiency and thus a higher average specific impulse are the other advantages of dual bell nozzle .

The numerical method yields a good simulation from the flow behavior and helps to improve the understanding of the physics of the flow.
the bell nozzle with 1.5 Mach number at the exit gives less shock and better flow separation behavior .

This design of double-bell nozzle has better use of the base area, and has higher thrust efficiency and thus a higher average specific impulse.

REFERENCES

- https://www.researchgate.net/publication/282765962_Numerical_Investigation_of_Flow_Transition_Behavior_in_Cold_Flow_Dual_Bell_Rocket_Nozzles
- https://www.ripublication.com/irph/ijert19/ijertv12n7_17.pdf
- https://core.ac.uk/download/pdf/154060575.pdf
- https://link.springer.com/chapter/10.1007/978-3-030-53847-7_25