HCLTech | Supercharging Progress™

Prediction of shockwaves using CFD method based on adaptive mesh technology

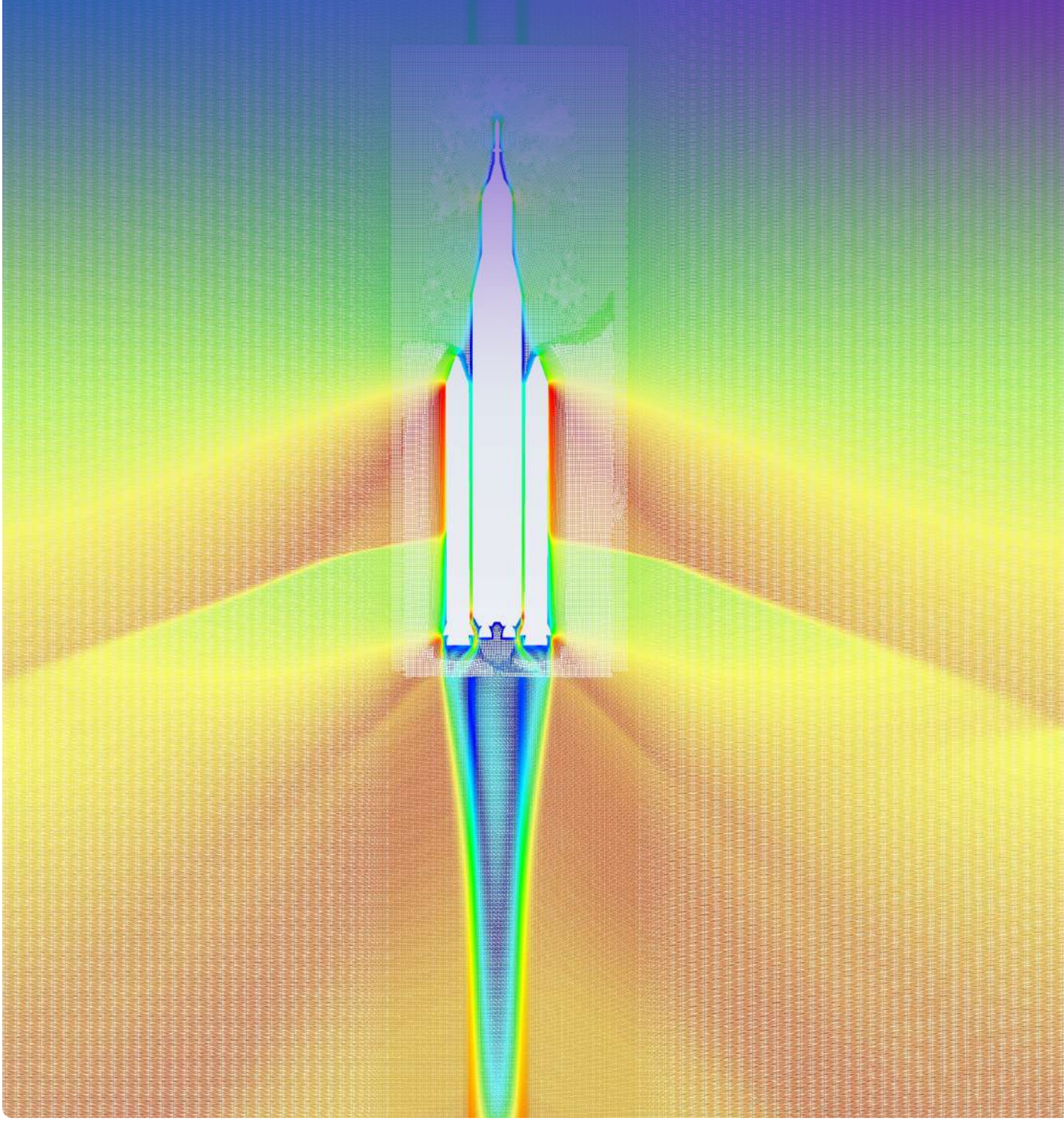


Table of contents

Abbreviations

AMR	Adaptive Mesh Refinement
CFD	Computational Fluid Dynamics
NS	Normal Shock
OS	Oblique Shock
A	Geometric angle
θ	Flow deflection angle
β	Shock wave angle
C_d	Drag coefficient
D	Drag
M	Mach number
M_1	Mach number of upstream flow
M_2	Mach number of downstream flow
P_1	Static pressure of upstream flow
P_2	Static pressure of downstream flow
T_1	Static temperature of upstream flow
T_2	Static temperature of upstream flow
V_{source}	Velocity of source/object
V_{Wave}	Velocity of sound
R	Characteristic gas constant (R_{air} =287 J/KgK)
P	Density of flow stream
Y	Ratio of specific heats (y_{air}=1.4)
RANS	Reynolds -Averaged Navier-Stokes
DES	Direct Eddy Simulation
LES	Large Eddy Simulation
SA	Spalart-Allmaras
k	Turbulence Kinetic Energy
E	Rate of dissipation of the turbulent kinetic energy
W	Specific rate of dissipation of turbulent kinetic energy
atm	Standard Atmosphere Pressure unit
K	Kelvin

Introduction

A shockwave is one of the most efficient mechanisms of energy transfer. Any sudden release of energy would result in a shockwave. They are generally present in supersonic or hypersonic flows while flowing over or through any object and there are two types of oblique shockwaves (Anderson, 1990). If the flow remains supersonic after the shock wave, it is called a weak oblique shockwave and if the flow changes to a subsonic after the shock wave, it is called a strong oblique shockwave.

The fluid flow properties, such as static pressure, static temperature, velocity and total pressure, change drastically across these shockwaves. Particularly, shockwaves travel much faster than sound and can be seen in a number of engineering applications, such as on the surfaces of transonic and supersonic flights, missiles, rockets, space shuttles, re-entry vehicles, engine intakes of supersonic aircraft, inlet ducts of scramjets, transonic gas turbine blade tip gaps or turbine blade passages where flows are at transonic speeds (Anderson, 1990).

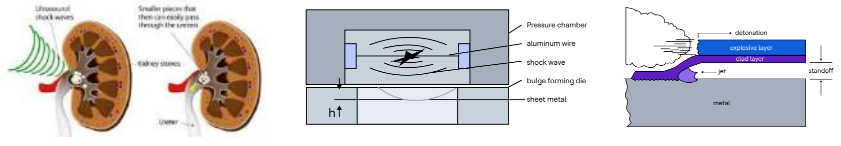
For the design of these systems, it is important to understand the properties and behaviour of the shockwaves. The wind tunnels intended for testing have limitations in generating the supersonic flows at realistic test conditions and capturing the shockwaves for physical study (Anderson, 1990). These restrictions can be overcome with the help of Computational Fluid Dynamics (CFD) analysis.

Shockwaves can be predicted precisely by using the adaptive mesh technology in the CFD analysis (ANSYS, Inc, 2009).

This paper discusses about CFD analysis done on wedge, cone, space-shuttle and double-wedge geometries, along with a case study to arrive at the best CFD methodology to capture the shockwaves.

Industrial and medical applications of shock wave

Apart from the aerospace domain, the applications of shockwaves are seen in several industrial and medical systems. Explosive welding, sandalwood oil extraction, pencil manufacturing industry and metal forming are some of the areas where shockwaves are used effectively in industrial domain (Anoop R B, 2019). Applications in the medical domain include bile duct stone treatment, cardiac shockwave therapy, extracorporeal shockwave therapy and treatment of upper urinary tract stones (Ohtani, 2005).



Kidney stone treatment

Metal forming

Explosive welding

Figure 1 – Applications of shockwaves (Lithotripsy (ESWL), 2023)

Methodology

The solver type and turbulence model are crucial in determining stability, convergence and accuracy of the CFD flow analysis. Therefore, have been chosen accurately from a case study analysis. In general, two types of solvers with different algorithms (Figure 2) are available for solving flow equations (ANSYS, Inc, 2009) and both solver methods can calculate the velocity field from the momentum equation.

In the pressure-based solver approach, the pressure field is calculated from the pressure-correction equation, which is based on the continuity and momentum equations, whereas in the density-based solver approach, the pressure field is calculated from the state of equations and the density field is calculated from the continuity equation.

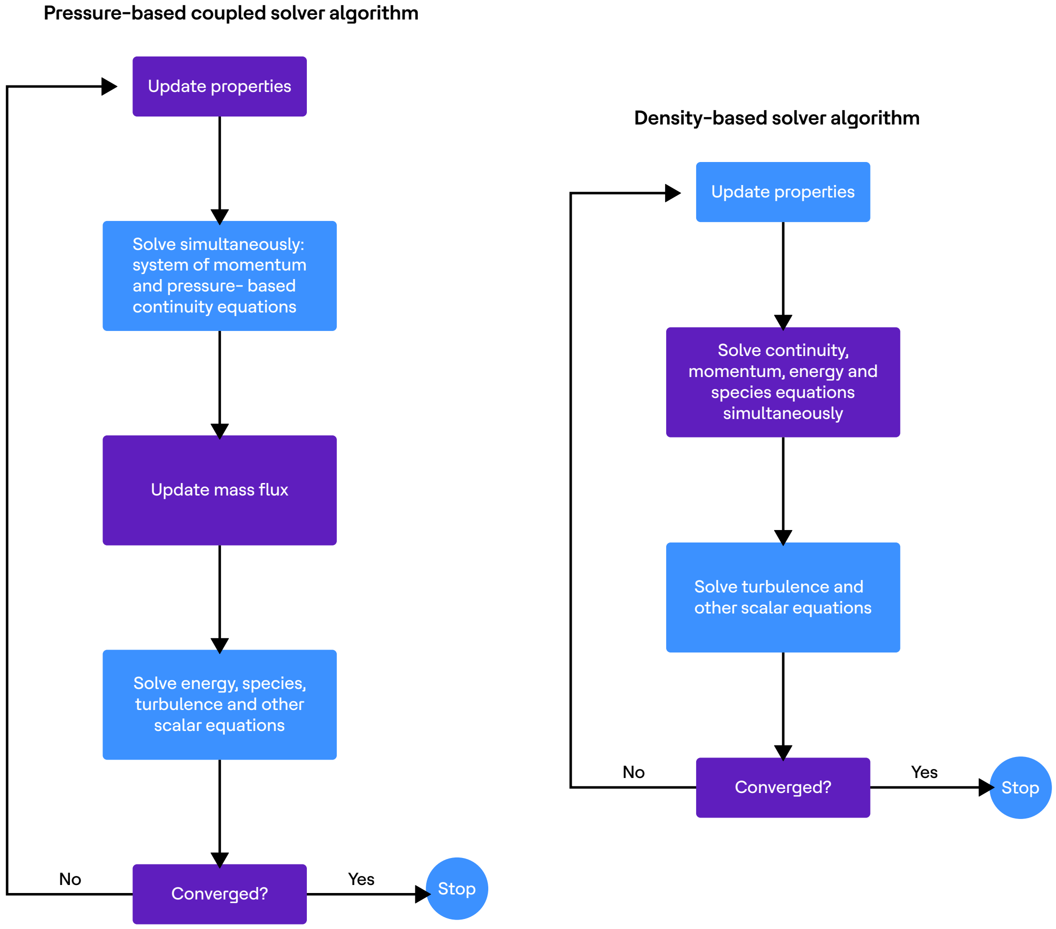


Figure 2 - Solver algorithm of pressure and density based solver

From the case study analysis, we noticed that by choosing a density-based solver for solving the flow equations, the residual convergence patterns are faster and smoother. Furthermore, the fluctuations of pressure acting on the solid surface are very minimal. For instance, consider a residual plot from one of the case analysis of flow over a wedge of angle 35° and incoming airstream at Mach number 3. It can be observed that the density-based solver is able to converge the flow residuals smoothly as compared to the pressure-based solver. Therefore, the density-based solver has been chosen as the default solver for all analysis.

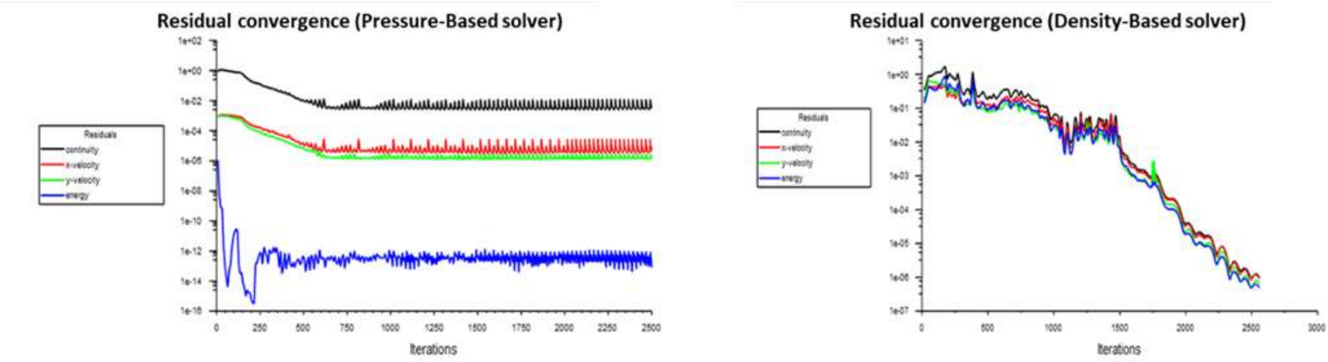


Figure 3 – Pressure based vs density based solver

Usually, during analysis where the objective is to capture the shockwaves, the inviscid flow assumption is quite sufficient. However, if studying the interaction of shockwaves with the wall boundary layers and capturing the shear stresses acting on the surface is required, it is essential to model the flow as viscous for the analysis. To capture the viscous flow and its boundary layer a turbulent model needs to be enabled. The RANS turbulence model has been chosen because to its faster convergence and less computational effort compared to DES or LES. Amongst the many RANS turbulence models available, an appropriate model was imperative for the current analysis.

From the case study of the supersonic flow of Mach number 3 over a wedge, it was discovered that Spalart-Allmaras (SA) model would be able to converge faster based on the comparison between residual plots with different turbulence models (Figure 4). Hence, the SA model was chosen for the analysis. The SA turbulence model was developed specifically for aerospace applications involving wall-bounded external flows (Spalart, 1992).

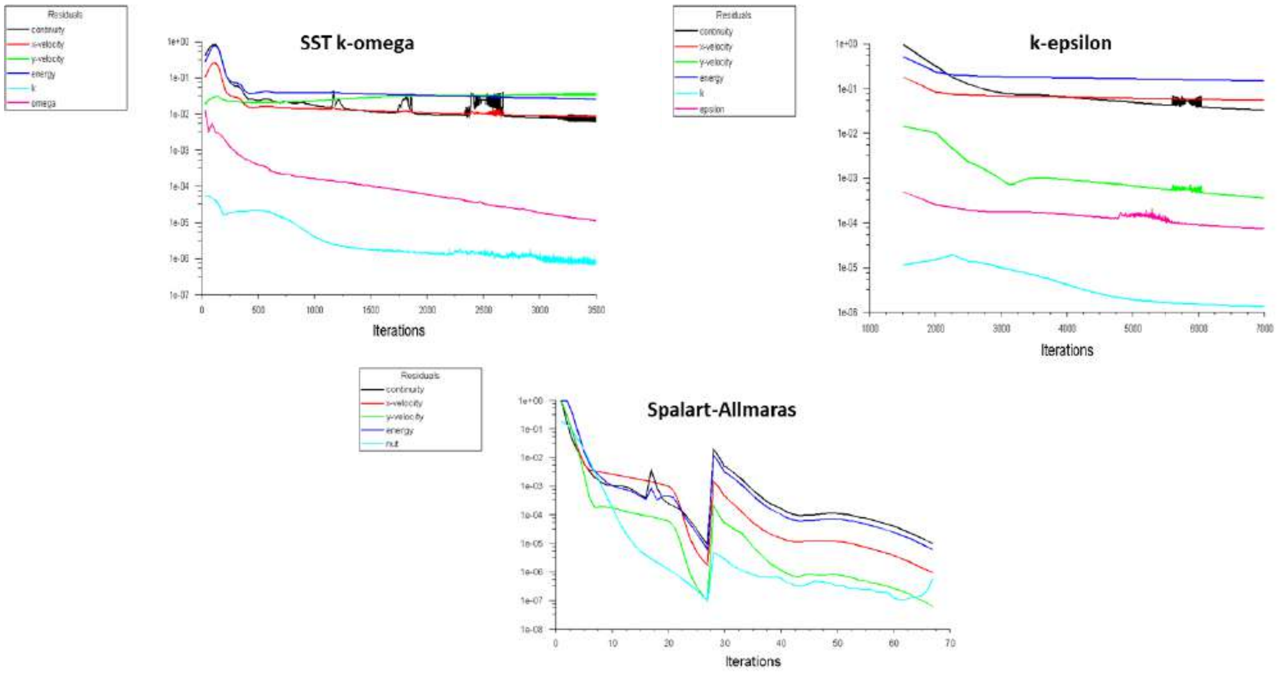


Figure 4 - RANS turbulence model comparison

Adaptive mesh technology available in the ANSYS Fluent tool was used to refine the shockwaves boundary layer more precisely. By using this adaptive mesh function in this tool, the mesh zones surrounding the shockwave could be refined to any level. On reiterating the flow solver with this modified mesh, the shockwave boundary layers would be confined to a sharp layer, as shown in Figure 5.

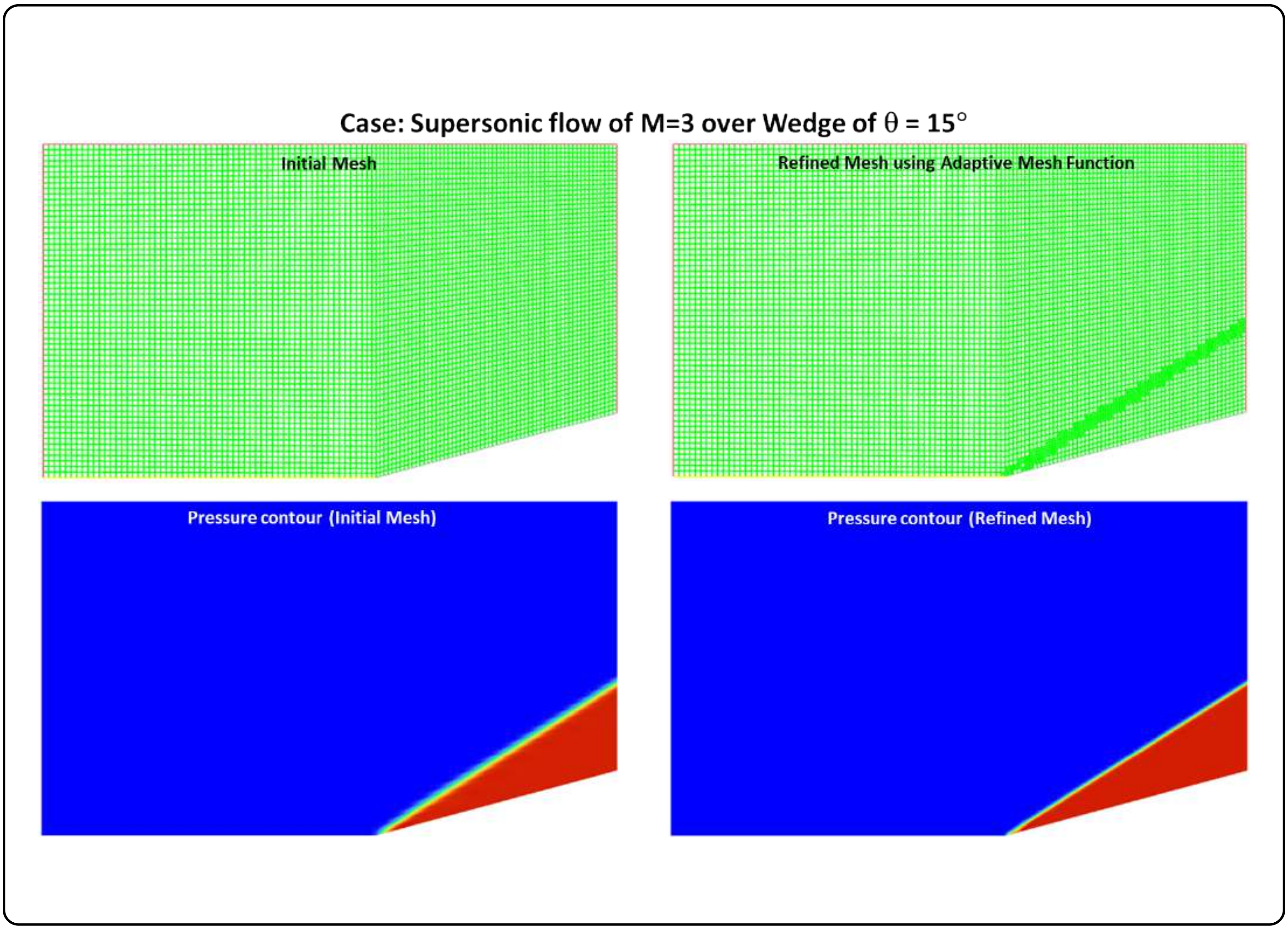


Figure 5 - Adaptive mesh technology (ANSYS Fluent)

The CFD result from one of the case studies is shown in Figure 5, which shows atmospheric air flowing at Mach no. 3 over a wedge surface angle of 15°. From the analysis, the shockwaves formed at the front tip of the wedge have been captured.

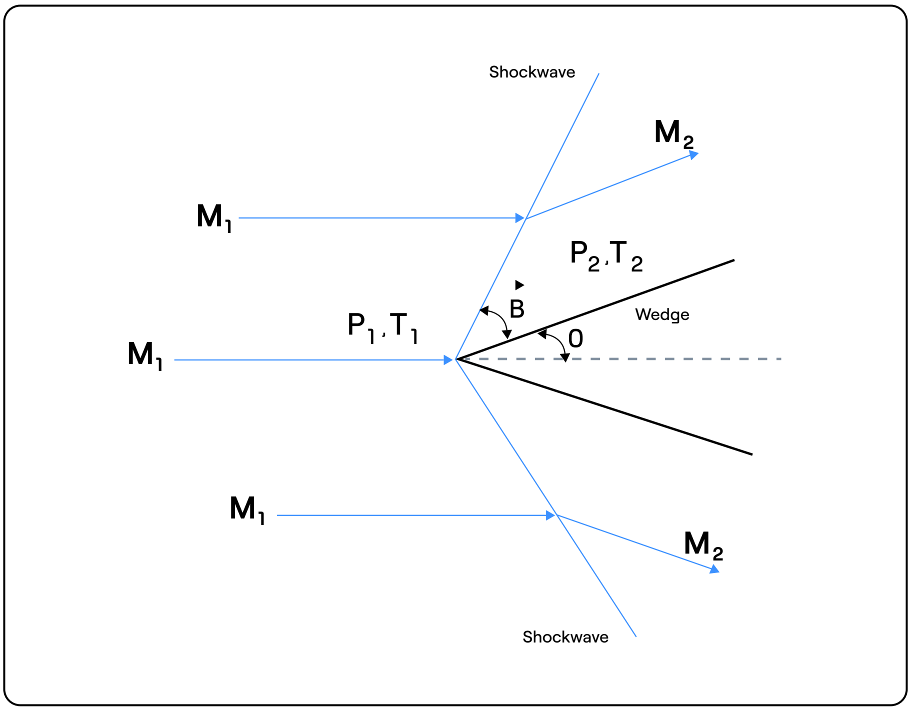


Figure 6 - Supersonic flow over a symmetric wedge

The results were compared and validated with the theoretical model. Table 1 shows how the adaptive mesh has increased the accuracy of the analysis.

Variable	Theory	Base Mesh	Error (%)	Adaptive Mesh	Error (%)
β	32.24	32.02	0.6	32.02	0.6
M_2	2.254	2.356	4.4	2.260	2.7
P_2 (atm)	2.821	2.691	4.6	2.726	3.2
T_2 (K)	416.4	415.5	0.2	417.0	0.1

Table 1 – Validation with theoretical model

Flow over wedge vs. cone

The wedge shape inlet in the supersonic aircraft's engine are used to create shockwaves and to reduce the incoming flow velocity, while the cone shape can be noticed in the front section of fighter jets, rockets and missiles flying at supersonic velocity. The cone shape inlet helps in reducing the speed of the incoming supersonic flow by creating an oblique shockwave.

The supersonic flow over a wedge surface creates a very strong shockwave, whereas the same flow over a same size cone with the same incident angle creates a weak shockwave.

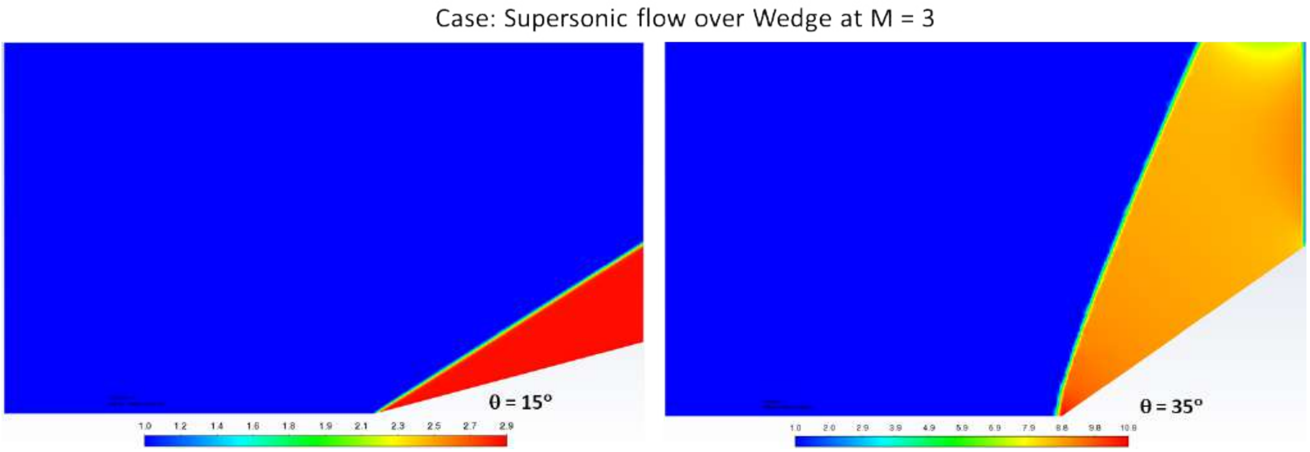


Figure 7 - Static pressure (atm) contour ofwedge

Consider a case of supersonic flow of M = 3 over wedges of angles 15° and 35°. From Figure 7, it can be noted that a strong shockwave has formed over both wedges but in the case of the wedge of angle 35°, the shockwave has detached from the front tip surface of the wedge.

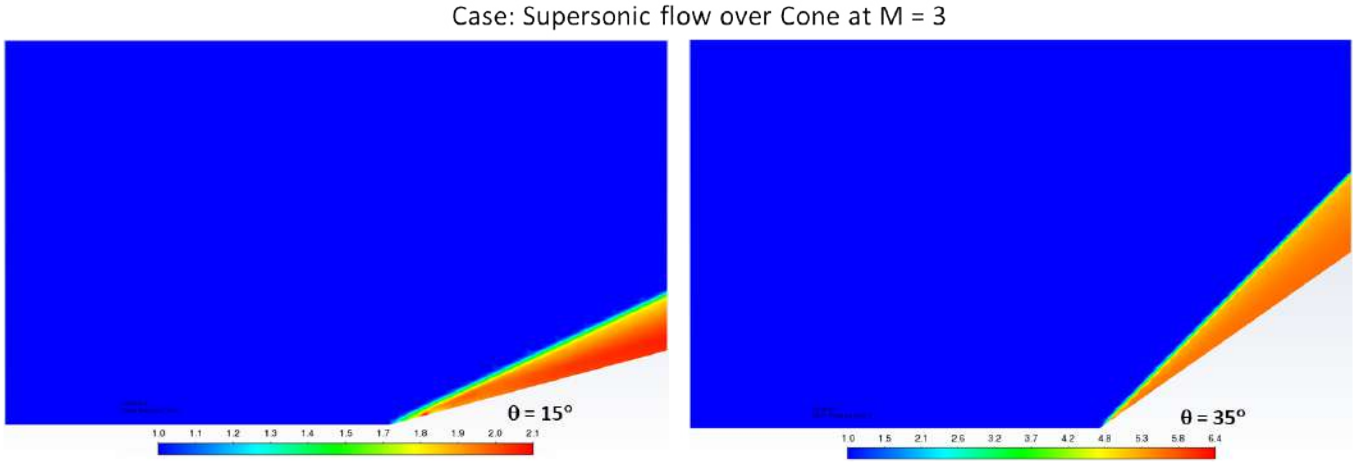


Figure 8 - Static pressure (atm) contour of cone

In the case of the same supersonic flow of M = 3 over cones of angles 15° and 35°, a weaker shockwave was observed (Figure 8). At a higher incident angle (θ=35°), the shockwave detached from the tip of the cone surface and moved towards the downstream position instead of the upstream position, like in the case of the wedge.

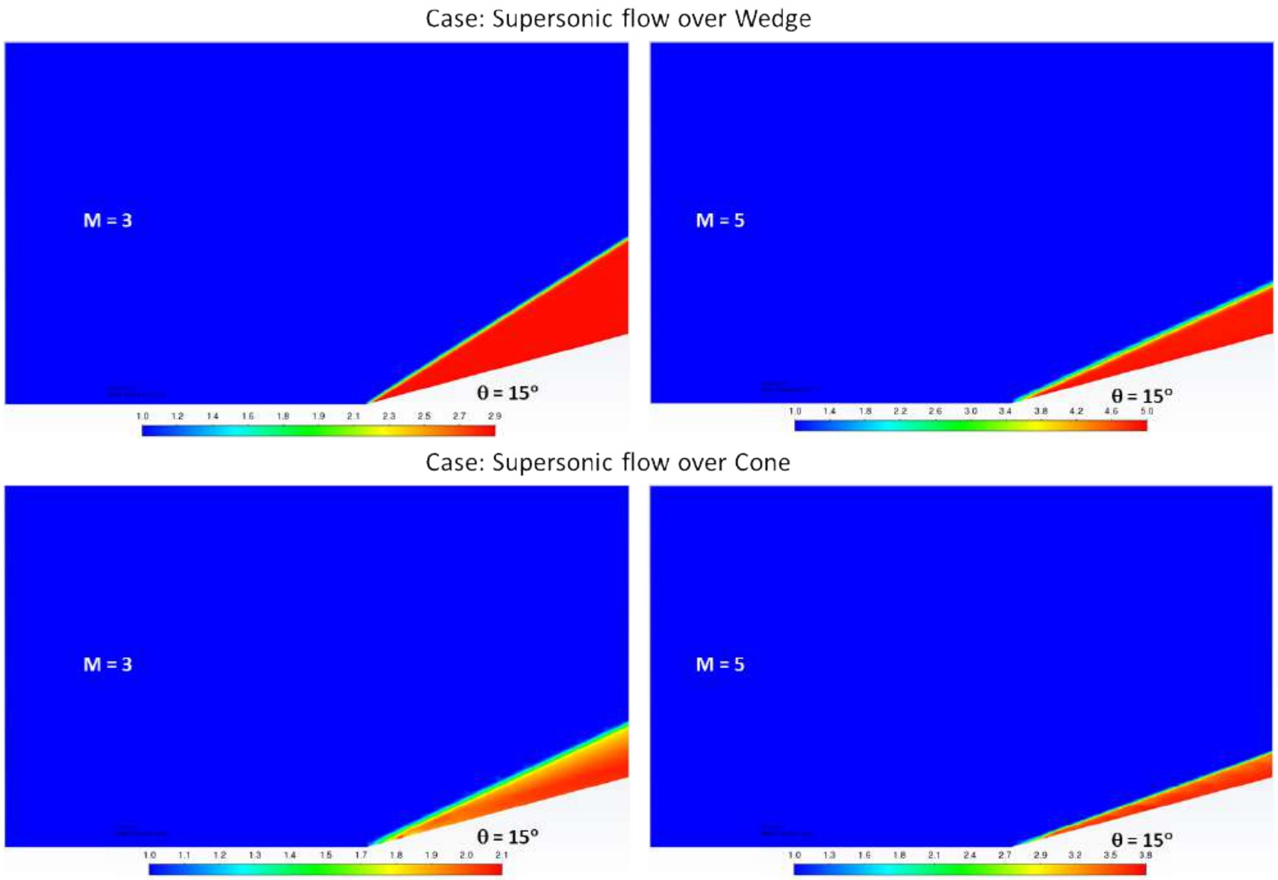


Figure 9 - Static pressure (atm) contour of wedge and cone at M = 3 and 5

Moreover, Figure 9 suggests that the angle β of the oblique shock decreases with an increase in Mach number. The oblique shockwaves are moving closer to the surface at higher Mach number which leads to a very high temperature profile close to the surface at hypersonic Mach number. To avoid this, blunt shapes are used at the front end instead of sharp pointed edges in the hypersonic missiles or rockets.

Flow over blunted edge cone

As mentioned in the previous section, to avoid higher thermal heating on the wall surface due to hypersonic flow, blunted edge cone is used to detach the shockwaves from the wall surface.

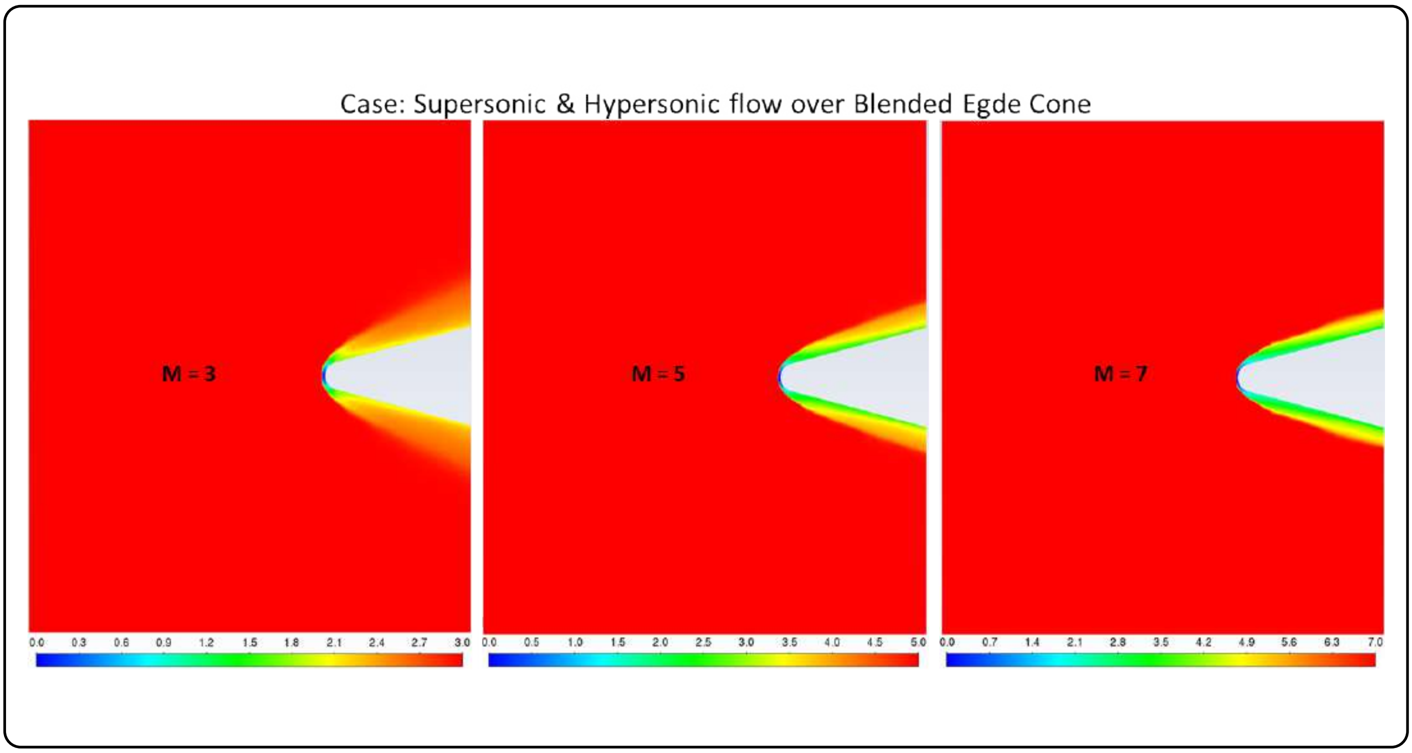


Figure 10 - Mach contour of blunted-edge cone

From Figure 10 it can be seen that by using the blunted edge cone, stronger shockwaves are created and detached from the front tip of the cone at Mach numbers 3, 5 and 7, unlike in the sharp edge cone model, which is the primary reason for using the blunted-edge front section in hypersonic vehicles like space shuttles (Figure 11).

Figure 11 - Space shuttle (The Editors of Encyclopaedia Britannica, 2022)

Flow over double wedge

In a similar way, the analysis was extended to the double-wedge model to understand the flow behavior and the shockwave pattern.

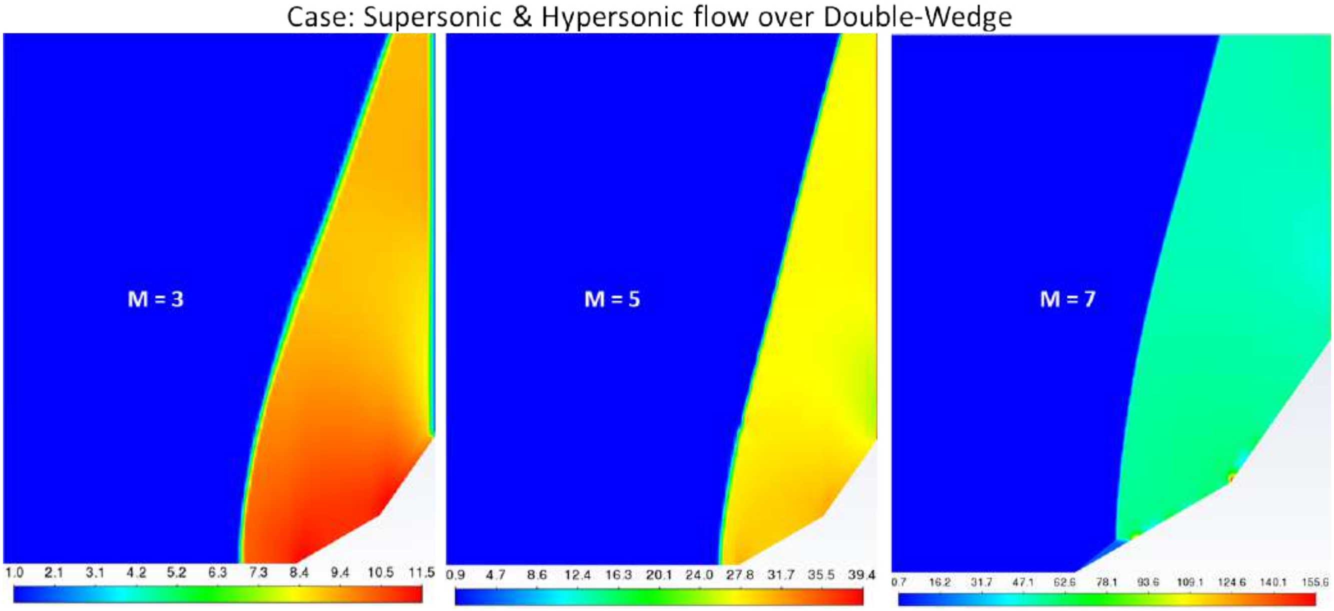


Figure 12 - Static pressure contour (atm) over double wedge

From Figure 12 it can be noted that at Mach numbers 3 and 5, a strong detached shockwave at the front end of the model can be observed. But at Mach no 7, two different shockwaves: one weaker shockwave at the front end and a stronger shockwave at the rear end were found.

Conclusion

A CFD-based methodology was developed to predict the shock waves resulting from supersonic and hypersonic flows and was successfully implemented in the case study analysis. The new CFD methodology based on adaptive mesh technology has shown good convergence with good accuracy in the flow solution. CFD engineers are encouraged to implement this methodology in any similar supersonic or hypersonic analysis and to use the adaptive mesh technology to capture the shockwaves precisely. The methodology can also be adapted to other engineering applications in industrial, manufacturing and medical domains.

References

Andersen, B. W. (2001). The Analysis and Design of Pneumatic Systems. Malabar, Florida: Krieger Publishing Company.

Anderson, J. D. (1990). Modern Compressible Flow. McGraw-Hill Publishing Company.

ANSYS, Inc. (2009, 01 23). ANSYS FLUENT 12.0 Theory Guide. Retrieved from https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/main_pre.htm

Anoop R B et al. "A Review of Applications of Shock Wave", International Research Journal of Engineering and Technology (IRJET) 6 (2019)

Denise Meuken, Erik Carton, "Explosive Welding and Cladding", July 2004, https://www.researchgate.net/publication/252721350_Explosive_Welding_and_Cladding/citations #fullTextFileContent

Langstädtler, Lasse, et al. "Electrohydraulic incremental bulk metal forming." MATEC Web of Conferences. Vol. 190. EDP Sciences, 2018.

Andersen, B. W. (2001). The Analysis and Design of Pneumatic Systems. Malabar, Florida: Krieger Publishing Company.

Anderson, J. D. (1990). Modern Compressible Flow. McGraw-Hill Publishing Company.

Anoop R B, J. k. (2019). A REVIEW ON APPLICATIONS OF SHOCK WAVE. International Research Journal of Engineering and Technology.

ANSYS, Inc. (2009, 01 23). ANSYS FLUENT 12.0 Theory Guide. Retrieved from https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/main_pre.htm

Lithotripsy (ESWL). (2023). Retrieved from Urology San Antonio: https://www.urologysanantonio.com/kidney-stones/lithotripsy

Ohtani, K. T. (2005). Applications of shock wave research to medicine. WIT Transactions on Modelling and Simulation.

Spalart, P. R. (1992). A One-Equation Turbulence Model for Aerodynamic Flows. AIAA, Paper 92-0439.

The Editors of Encyclopaedia Britannica. (2022, 11 18). STS, Space Transportation System. Retrieved from Britannica: https://www.britannica.com/biography/Ronald-McNair

Author information

Dinesh Babu

Dinesh Babu has been working at HCLTech since 2015. He has more than 12 years of industry experience in CFD and aerospace domains. He has a master's in mechanical engineering in aerospace technology from SUSSEX University, UK. He has extensive experience in numerical simulation and analysis of multi-domain industry applications and has good exposure to open source CAE domains and coding.

Kavya S

Kavya S has been working at HCLTech since 2022. She joined as a fresher and completed an internship with HCLTech. She has a bachelor's in aerospace engineering from M S Ramaiah University of Applied Sciences and has experience running several CFD simulations and strong domain knowledge in the areas of aerodynamics, fluid flow, aero-engines and propulsion. She is also familiar with open-source CFD coding and tools.

Basamma Gaddad

Basamma Gaddad has been working at HCLTech since 2022. She joined as a fresher and completed an internship at HCLTech. She has a bachelor's in aerospace engineering from M S Ramaiah University of Applied Sciences and has experience running several CFD simulations and significant subject matter expertise in the fields of propulsion, aerodynamics, fluid flow and structures. She is also knowledgeable about preprocessing and CFD tools.

Vijay Kumar Turaga

Vijay Kumar Turaga has been working at HCLTech since 2010. He has close to 26 years of industry experience in the CFD Domain. He has a master's in mechanical engineering from IIT Madras, Chennai, with specialization in internal combustion engines. He has handled a wide variety of industrial problems involving conjugate heat transfer, air fuel mixing and combustion, rotating machinery and multiphase. At HCLTech, he is a part of the MCOE CFD and Thermal team and has experience of working across all verticals such as aerospace, automotive, industrial, consumer products, Hi-Tech, semi, telecom and medical devices.

HCLTech | Supercharging Progress™

hcltech.com