1. INTRODUCTION

During the evolutionary process of aerospace vehicle design, it is very difficult to conduct wind-tunnel experiments everytime some design changes are made to the product, be it a missile/aircraft airframe, blades of turbofan-engine, a rocket-motor or even a sub-system level component of a flight vehicle, though wind tunnel experiments are very important. The time consumption, cost and risk involved in conducting experiment for every design change are quite high. Under such circumstances, computational fluid dynamics (CFD) is the preferred and reliable option of flow analysis for majority of the designers where fluid-flow dictates the real terms and conditions. Dawning of this realization can be attributed to the maturity level of numerical methods in handling complex geometries, complex algorithms, in-depth and broad spectrum understanding of the complex physical modelling, accumulated experiences of decades of code developments and availability of large high performance computing resources. Complex modeling through CFD is time consuming with intricacies of grid generation, flow modeling and establishing grid independency of solutions, but once established it can account for design changes many times in a design cycle without any serious hierarchical work-flows as in the other spheres of flow analysis. CFD no more deals just with fluid flows alone and the CFD codes are equipped to handle interactions with many other physical disciplines as well as to compliment experiments and creation of valuable aerodynamic databases. Keeping in mind the requirement of all such needs and stringent demands, development of CFD codes that can satiate the in-house designer becomes necessary and sometimes inevitable due to various external factors, one such being non-availability of commercial CFD codes due to control regimes or sanctions.

CERANS\textsuperscript{1} is one such flagship RANS code of DRDL, developed indigenously to overcome the technology denial for addressing the aerodynamic design and high-speed flow problems of DRDO missile configurations.

In this paper, the efforts towards indigenous development of Compressible Euler/Reynolds Averaged Navier-Stokes solver also called CERANS, its present capabilities, the validation aspects, recent applications to DRDO missiles and the immediate and future roadmap were discussed.

2. GENESIS AND DEVELOPMENT OF CERANS

CERANS, Compressible Euler/Reynolds Averaged Navier-Stokes solver is a general purpose CFD code developed at DRDL, Hyderabad, for solving the RANS equations in cell centered finite volume framework on sequential and parallel computers. The baseline code was developed during early years of this decade in FORTRAN-77 and later ‘C’ language was adopted for the requirement of dynamic memory allocations. Since the code is designed to be grid format independent, several structured and unstructured grid preprocessors have also been developed requiring treatment of different types of data structures for each of these grid types. The code can handle flow regimes from low subsonic to hypersonic Mach numbers and can tackle complex geometries involving both structured as well as, recently, the unstructured polyhedral meshes. The code had been extensively validated for studying its accuracy, robustness range of applicability and limitations.

3. CAPABILITIES AND LIMITATIONS

The interfacial convective fluxes are modeled using several state-of-the-art numerical flux formulae such as KFVS, van Leer’s, Rusanov’s, Roe’e family, AUSM family, Steger-Warming family and HLL family of numerical schemes. Higher order spatial accuracy is obtained using the method of reconstruction for structured meshes or the in-house R&D based Polyhedral MUSCL\textsuperscript{2} approach for the unstructured meshes. The interfacial flow gradients required for the evaluation of viscous stresses are obtained by using cell-centered weighted least squares method as in the case of a structured grid or the diamond path reconstruction approach as used for unstructured grid. An unified characteristic boundary condition is applied at all the external boundaries. The turbulence closure is effectuated by the one-equation Spalart-Allmaras\textsuperscript{3} turbulence model. An adaptive, robust and reliable, blended wall function based on Spalding’s universal law of the wall\textsuperscript{4} with Christoph and White correction accounting for compressibility and heat transfer has been used to evaluate the wall shear stress and wall heat flux. This near wall treatment enables the use of fine grids in regions where accurate results are sought and coarse grids in the region where reasonably acceptable solutions are sufficient without compromising the solution accuracy.
The solver has been parallelized using the standard MPI libraries. Domain decomposition is carried using the Open-ware METIS. The geometric data structures required for parallel version of CERANS had been explicitly provided by a parallel preprocessor developed in-house. An efficient and robust graph based scheduling algorithm had been implemented in CERANS for data transfer across various processors. The matrix-free Data Parallel Lower-Upper Relaxation (DP-LUR) and the Hybrid Lower Upper Symmetric Gauss-Seidel (Hybrid-LU-SGS) implicit algorithms along with options of local or global time stepping have been implemented in the parallel version for accelerating the convergence of flow evolution to steady state. The Weiss and Smith preconditioner implemented in CERANS allow simulation of even incompressible flows. CERANS is routinely being used for simulating viscous, high-speed, turbulent flows past complex flight vehicle configuration of interest to DRDO. Copyright had been obtained for CERANS from the Copyrights Office, New Delhi, through DRDO Headquarters.

One of the major limitations of CERANS as had been thought of by the developers is that the code in its present form is not user-friendly and direct intervention of the developers is required whenever a new model is required to be implemented or the inability of the users themselves to sort a problem out while using the code for their own applications. Also only one turbulence model, namely the Spalart-Allmaras one equation model is presently available in the application version, though it is always recommended to have at least a 1-equation and a 2-equation turbulence model in a CFD code. Barring these shortcomings, which will be addressed in due course, CERANS had been broadly accepted in the CFD community as an industrial standard CFD code.

4. VALIDATION OF CERANS

By definition, validation is the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model. Accordingly, CERANS had been validated extensively right from 2D bench mark problems, 3D axisymmetric to complex high speed flight vehicle configurations. Wide range of flow conditions from very low subsonic to hypersonic Mach numbers, laminar, tripped transition and turbulent flows, flows involving shock-wave boundary layer interactions with wall heat transfer effects, high angle of attack flows, large size eddy dominated vortical flows with cross-flow separations have been simulated using CERANS for its validation and the results obtained generally compares well with the available results from literature such as wind tunnel experiments and or other carefully conducted numerical experiments. The code was executed even for a very fine near wall spacing of the order of 0.1 microns, for capturing finer details of flow. Such validation exercises have provided the confidence to move forward for solving real world applications as being carried out routinely at DRDL. It has to be mentioned that whenever possible, such validation exercises are still being carried out from the latest versions of the CERANS code for repeatability of results and assessing the integrity of the flow solver in its latest order.

5. RECENT APPLICATIONS OF CERANS TO DRDO MISSILE CONFIGURATIONS

A broad spectrum of flow simulation applications encompassing the missiles configurations of DRDO had been carried out using CERANS. The simulations involve aerodynamic characterization of missiles right from very low subsonic to hypersonic Mach numbers, angles of attack as high as 70°, effects of jet flows in complex geometries involving complex physics, effects of variation in Reynolds Numbers by several orders of magnitude at high Mach numbers, estimation of wall heat fluxes in high speed environment etc., to name a few. Application of CERANS for a few missile configurations is presented here.

5.1 ASTRA Missile

Several inviscid and viscous transonic-to-high-supersonic Mach number, low-to-high angles of attack flow simulations for several design versions of ASTRA missile configurations have been carried out using CERANS. Notable among the studies, is the one conducted for physical understanding of the cause of rolling moment at a roll orientation of 22.5° degrees for a Mach number of 2 at an angle of attack of 20° degrees with unit Reynolds number of 8.03x10^6. Fig.1 shows the formation of cross flow vortices over the body at various cross sections represented by the total pressure loss contours. It is observed that the cross flow vortices due to high angle of attack emanating from the body are split by the wing asymmetrically and these vortices of different strengths coalesce with wing tip vortices producing severe rolling moment. Gaining of such physical insight of complex flow phenomenon can be appreciated well by CFD simulations per se.

Figure 1. Vortex formation along the Astra missile at angle of attack=22.5°
are one with low aspect ratio wings and the other configuration with a high aspect ratio wing, the planform area being same for both. It had been found that the axial force due to the high AR wing is higher by about 20% due to higher wave drag than the low AR wing, whereas the changes due to normal forces and pitching moments are just about 7% higher for the high AR wing. The flow contours and its complex pattern around high AR configuration are shown in Fig. 2.

The Mach contours in pitch plane and at the fin cross-section are shown respectively in Figs 4 and 5. It was found from the CFD studies that the skin friction contribution to total axial forces is the major component of $C_A$, and the contribution of the pressure component of $C_A$ due to protrusions is also significant in the transonic Mach number range. In the supersonic flows, the skin friction was contributing to about 30-40% of the total $C_A$. Further high angle of attack study is presently underway for assessing the performance of SR SAM.

5.2 SR-SAM Missile

CFD analysis for SR-SAM configuration is being carried out using CERANS from transonic to supersonic Mach numbers and for angles of attack as high as about 40 degrees. Due to the complexity in geometry, such as the bluff launch shoes, wing joints, antennae protrusions etc., it was required to generate unstructured tetrahedral meshes with near wall polyhedral extruded grids. The grid size amounts to about 5 millions control volumes for each of the variant of missile. The Polyhedral-MUSCL option from CERANS is used for addressing the unstructured mesh computations. Typical surface pressure contours for transonic flow Mach number of 0.9 and angle of attack of 37° for a unit Reynolds number of 3.25 million shown in Fig. 3 depict the regions of high pressure around the protrusions.

5.3 HSTDV Launch Vehicle

5.3.1 Evaluation of Heat Flux

The heat flux distribution for the HSTDV launch vehicle forebody (up to 5m length) had been obtained using CERANS for various constant isothermal cold wall conditions. The
freestream Mach number considered is 7.89, angle of attack is 0°, freestream static temperature is 200K, and the unit Reynolds number is 29.34 Million. The first point spacing of grid point away from the wall is about 0.75 microns. AUTOELGRID\textsuperscript{6} had been used for grid generation. Typical results of flow solution in the form of Mach contours (overlaid upon the grid) in the pitch plane for a wall temperature of 700K is shown in Fig. 6. It had been observed from these simulations that the heat flux at 5m length varies between 0 and 20 Watts/cm\textsuperscript{2} for cold wall temperatures from 600 to 900K. Fig.7 presents the boundary layer profiles (U\textsuperscript{+} vs. Y\textsuperscript{+}) and Fig.8 presents the temperature profiles at an axial station of 4.2m. It can be observed that the velocity profiles are represented well and as the cold wall temperature increases, the trend of the velocity profile shifts towards the theoretical adiabatic wall profile. The results obtained by CERANS had been compared with results of other standard codes and the agreement is good.

5.3.2 Reynolds Number Variation Study

Aerodynamic characterization of HSTDV launch vehicle in hypersonic flows had been carried out using the CERANS for various Reynolds numbers viz., 10\textsuperscript{6}, 10\textsuperscript{7} and 10\textsuperscript{8} per unit length of the geometry, from Mach Nos. 4 to 8 with angles of attack of 2° and 4° respectively. This study was required to be carried out due to non-availability of experimental results at flight Reynolds number, and to table the results for both the experimental and flight Re in addition to an intermediate Re between the two. Structured hexahedral single block grid had been generated for the HSTDV launch vehicle configuration using AUTOELGRID. For high Mach numbers, the lower Re cases tend to be predominantly laminar and as the Reynolds number increases, flow was observed to be fully turbulent. The lower Re cases tend to produce more axial force than the higher Re counterparts. The critical aspect of the simulations is that there was as much a shift of about 0.22D in the center of pressure, with Reynolds number variation, occurring at Mach 7 and angle of attack 2 degrees.

5.4 B-05 Missile

5.4.1 Low Speed Flow Characterisation

This was for the first time, ‘incompressible’ flow simulations\textsuperscript{7} had been attempted for a practical flight vehicle configuration (B-05 Mk-II) using CERANS. The freestream Mach number considered is 0.0286 corresponding to a flow velocity of 10 m/s at sea level and the angle of attack was varied from -12° to +12°. The numerical stiffness associated with low speed incompressible flows while using a compressible CFD code had been overcome by the method of preconditioning. The Weiss and Smith preconditioner and all speed AUSM+ numerical flux formulae with modified numerical speed of sound had been used to evaluate the flow field. Laminar version of CERANS with explicit update procedure had been applied for all the simulations. DRDL SUMO and AUTOELGRID\textsuperscript{6} codes have been used for generation of
the grids. The agreement of aerodynamic coefficients obtained from the simulations is good when compared with experiment. Typical flow contours in pitch plane at angle of attack of 4 degrees are presented in Fig. 9. The aerodynamic coefficients when compared with experiment values are higher by about 15 to 30 per cent for the normal force and pitching moment coefficients. The axial force coefficients are lower than experimental values by as high as 30%. The difference in results between CERANS and the experiment can be mainly attributed to the differences in geometry between the experimental model which contained antennae protrusions and CFD model without antennae.

5.4.2 Launcher-plume Deflector Flow Analysis

To estimate the jet loads on the jet-deflector of the mobile launcher configuration of the B-05 missile during lift-off, three simulations were carried out with various offset condition of the nozzle axial locations, viz., 0mm, 30mm and 50mm. The height of the nozzle exhaust from the crown of the deflector is 1560mm. It was observed that the downward load is about 10.1 tonne with a side load of as high as 2.2 tonne for the maximum offset case. The temperature contours \((T/T_\infty)\) for the 50mm offset case is shown in Fig.10. The jet plume can be clearly seen from the Figure and its impingement with the deflector plate creating a detached inclined shock above the crown tip is depicted. These loads were used for the launcher design.

5.5 SF – A’ Missile

CFD simulations using CERANS were carried out to determine the safe separation distance and to study the complex flow field during the stage jettisoning process of A’ configuration. Two simulations with stage separation distances of 500 mm and 1000 mm were considered for simulation. AUTOELGRID was used for grid generation. It had been found from the simulation that there is a huge jet-upstream interference for the 500 mm case resulting in upstream flow separation. For the 1000 mm case, the flow is distortion free and the jet simply gets flushed downstream without any adverse interference with the upstream external flow. The flow field contours and stream line pattern for an upstream Mach number of 6 for 0 degrees angle of attack with the complex jet upstream interaction and the formation of Mach disc for the 500 mm case are shown in Figs. 11 and 12 respectively. The Mach contours clearly show the detached bow shock ahead of the nose, the Mach disc formed due to jet-free stream interaction and streamline plot show the flow separation ahead of the Mach disc. It had to be categorically emphasized here that for understanding and discerning a clear picture of such complex physical processes, CFD is the right analysis tool.

5.6 SF-A3 Missile

5.6.1 Launcher-plume Deflector Flow Analysis

Estimation of steady state lift-off loads of SF-A3 configuration due to lower stage nozzle exhaust jet on the
mobile launcher deflector plate had been carried out using CERANS. These loads were required to design the anchoring mechanism of the jet-deflector assembly. The nozzle exit plane was considered to be at 3.2 m (with 30 mm offset) and at 11.2 m (with 200 mm offset) height above the crown of the deflector. From the numerical simulations it was found that for the first case, the downward load on the deflector is about 79 tonne and the sideward load is about 7 tonne and for the second case, the downward load on the deflector is about 74 tonne and the side load is about 36 tonne. Typical Mach contours for both the simulations are shown in Fig. 13 and 14 respectively, and it can be observed that the flow is quite complex. These load estimates were used for designing the anchoring mechanism.

Figure 13. Mach contours for launcher plume Jet-Deflector flow analysis for SF-A3. (Case.1. Height 3.2m, 30mm offset).

5.7 AAD Missile

Numerical simulation of supersonic external flow past AAD missile configuration at Mach 4.5, angle of attack of zero degree with a unit Reynolds number of about 27 million had been carried out using CERANS. The prediction of necessary boundary layer inputs required for estimating the thermal environment around the launch-shoes is the motivation for the simulations. Unstructured polyhedral meshes were generated around the configuration and the Polyhedral MUSCL based flow simulation of CERANS had been used for obtaining the solution. It had been observed that the peak fluid temperature around the launch shoe is about 1370K. Typical Mach contours in pitch plane and around the launch shoes are shown in Fig.15 and 16.

Figure 15. Mach Contours for AAD at Mach 4.5, angle of attack=0° at pitch plane.

Figure 16. Temperature (T/T_¥) contours for AAD at Mach 4.5, angle of attack=0° near the launch shoe.

It can be clearly seen that there is a shock induced flow separation ahead of the launch shoe. The near wall fluid temperature contours are shown in Fig.17 and it clearly depicts the high fluid temperature zone and the
detached shocks ahead of launch shoes. Further, the boundary layer properties and the fluid temperature around the launch shoes were used as inputs for heat conduction model for estimating the airframe material temperature.

Figure 17. Near wall temperature (T/T) contours for AAD at Mach 4.5, angle of attack = 0° around the launch shoe.

5.8. Skin Temperature Prediction for Dhanush SLTGT Missile

Aerodynamic heating prediction for the Dhanush sea-launched target missile had been carried out for several flight trajectories. Inviscid version of CERANS was used for generation of data base required for the trajectory conditions. The Aeroheating analysis was carried out offline using a Skin Temperature Prediction Code (STPC). Heat conduction into the material and radiation of heat from the material surface were modeled. The necessary skin friction coefficient was obtained using van Driest method, the heat flux had been obtained using the Colburn’s analogy and the compressibility correction was provided using the Eckert’s reference enthalpy method. An ‘equivalent thickness’ analogy was used to correct the skin temperature. Material properties such as density, thickness, thermal conductivity and specific heat capacity were provided as input to the STPC. The simulated skin temperature contours over the missile are shown in Fig. 18. Skin temperature distribution over a nose-body-wing-fin generator is shown in Fig. 19. It can be observed that the skin temperature over the geometry hovered around 350 °C to 425 °C.

6. FUTURE SCOPE OF WORK

Any CFD code to compete at the highest level, requires to keep it tuned and abreast with the latest concurrent developments taking place elsewhere in the world for faster and high fidelity flow simulations. Keeping in mind such needs to stay put and for gaining the confidence level of the missile designers, the code developers have identified the some of the key technology areas that can be addressed by CERANS in the immediate and near future.

Figure 18. Skin Temperature (°C) contours over SLTGT missile at the end of the flight trajectory.

Figure 19. Skin temperature (°C) distribution at a nose-body-wing-fin generator.

6.1 Advanced Turbulence Models

CERANS in its application version is implemented with just the one-equation based SA model, hence it is required, as the industry norm, to provide more options for the users, from the point of view of turbulence models. Therefore the two equation Shear Stress Turbulence model of Menter is being implemented at present. Also it is proposed to implement an Algebraic Stress Model for improving the turbulence predictive capability, especially the adverse pressure gradient flows accurately.

6.2 Transition Models

Flows past hypersonic flight vehicles generally undergoes transition and sometimes tend to relaminarize also, leading to higher wall heat fluxes than the heat fluxes predicted by assuming fully turbulent flow. All the turbulent simulations carried out using CERANS so far have been without a transition model. viz, the flow is assumed fully turbulent. Addressing flow transition is a critical design issue for the designers and hence it is proposed to incorporate two
transition models in CERANS. The first one is an experimental correlation based transition model which solves for the intermittency and the momentum thickness Reynolds number using transport equations, which couples with the SST turbulence model through source terms. The second one is a physics based model where the underlying turbulence model is itself is tuned to mimic the transition behavior of flow. Both these models will be implemented in CERANS for enhancing further the predictive capabilities.

6.3 Higher Order Accuracy and Robustness

In order to obtain high quality, accurate results for futuristic proposals such as aeroacoustics and Large Eddy Simulations, it is necessary to incorporate higher order accurate models such as the discontinuous Galerkin (DG) methods. The DG method is gaining worldwide acceptance as the method to seek for, for very high order spatial accuracy and robustness, without enhancing the neighborhood stencil as in other higher order methods. Hence it is proposed to implement the same in CERANS.

6.4 Flow Evolution Based Grid Adaptation

Grid generation for complex geometries is laborious and daunting task involving several man-days of work and leads to exasperation if the final grid does not suit the needs of the flow solver. This can happen, for example, in case of changing the near wall clustering till desired Y’ is achieved or increasing the grid density in some selective region where some of the important flow features are expected to be missed out due to a coarser grid distribution. These problems can be alleviated by grid adaptation based on flow evolution whereby the grid is enriched or coarsened and or moved accordingly in and around the regions of interest. This area of activity will be considered for improving the modeling capability of CERANS further.

6.5 Real Gas Effects

Since it is necessary to model the air-chemistry for very high speed flows, especially at high hypersonic Mach numbers where dissociation and ionization effects of air molecules are predominant, it is necessary to address the non-equilibrium properties of flow and the chemical constituents of the fluid by modeling multiple species and reactions. In near future this aspect of code enhancement is planned to be implemented in CERANS.

6.6 Magneto Hydro Dynamics

For hypersonic flows with ionization and dissociation effects, the fluid around the vehicle can be considered as plasma. This plasma flow when subjected to an external magnetic field produces force called Lorentz force which can be expertly utilized for flow control. For hypersonic air-breathing propulsion system intake, the desired ‘shock-on-lip’ condition can be achieved all through the air-breathing engine operation even at off-design flight conditions using MHD flow control. Relaminarization of flow past hypersonic flight vehicles can be realized using MHD flow control. MHD can be used as an effective tool for creation of virtual fin/stabilizer which in turn can be used for control of missiles. Drag reduction have been realized by injecting high speed plasma jet ahead of a bluff body through the detached shock which in turn weakens the shock strength. It had been projected that all futuristic high speed flight vehicle projects, especially the hypersonic ones, will be essentially having MHD based flow control as a major tightly integrated subsystem unit. In order to simulate and realize such potential benefits, the MHD governing equations involving magnetic field is proposed to be included in CERANS.

6.7 Kinetic Heating Analysis

For the determination of missile wall temperature, whether it exceeds the critical temperature of the airframe material, it is necessary to couple the material heat conduction model with the fluid flow equations and solve for wall temperature while marching along the flight trajectory path. It is proposed to incorporate the kinetic heating analysis in CERANS for evaluation of airframe temperature which can provide valuable design inputs for proper material selection.

7. CONCLUSION

Details regarding the development, validation aspects, capabilities & limitations, application for missile configurations and future scope of work with respect to the CFD code CERANS had been brought out in this paper. Ability of CERANS for handling complex geometries and addressing complex flow physics had been brought out clearly. Overall, the potential of CFD as a right tool for aerodynamic design analysis and insightful physical understanding of complex flow phenomenon had been comprehensively highlighted. The enhancements proposed to be carried out in CERANS in the due course for maintaining the confidence of the designers also had been explained.

ACKNOWLEDGEMENTS

The authors wish to express their sincere thanks to Director, DRDL, Directors of Missile Complex Laboratories, Project Directors, Technology Director, DOCD, Head, CFD Division and Scientists of CFD Division for all their perpetual support and encouragement.

REFERENCES