10th ANKARA INTERNATIONAL AEROSPACE CONFERENCE 18-20 September 2019 - METU, Ankara TURKEY AIAC-2019-196

DISCONTINUOUS GALERKIN FINITE ELEMENT METHOD SOLUTIONS OF EULER EQUATIONS

Osman Güngör¹ Middle East Technical University Ankara, Turkey Serkan ÖZGEN² Middle East Technical University Ankara, Turkey

ABSTRACT

This paper provides solution method and sample results for discontinuous Galerkin finite element method (DG-FEM) solutions of two dimensional Euler equations which govern the inviscid and adiabatic flows with a set of quasi-steady hyperbolic equations. DG-FEM discretization of Euler equations are presented with proper boundary conditions. Implementation of boundary conditions is discussed in detail. Choice of flux function and limiting are addressed through the discretization. An explicit time integration method is chosen. Since the DG-FEM solves each cell in a discontinuous manner and cells are only connected to their neighbors, it is highly suitable for an object oriented approach and parallelization.

INTRODUCTION

Particularly in aerospace industry, understanding the fluid dynamics has enormous importance in the design process of the products. Experimental and theoretical approaches are two historically major branches of fluid dynamics. Experimentation helps to understand and evaluate physical phenomena in fluid dynamics however it is highly expensive for design process. Solving governing equations (or simplified forms) is the part of theoretical approach, but analytical solutions exist for only a few simple flow problems. Thanks to growth in computation power over the last decades, a third approach, numerical approach has gained importance. Evolutionary works [Jameson, Schmidt & Turkel 1981], [Jameson 1983] by Jameson pioneered the solving governing equations numerically, rather than mimicking. Finite volume algorithms increased robustness for applications with strong shocks and resolution with viscous problems by incorporating up-winding mechanism. However, currently, most of the finite volume methods used in fluid dynamics is limited in accuracy to second order due to complications of extended stencils in higher orders. On the contrary, finite element methods can provide higher order accuracies by formulation but for smooth inviscid and viscous flows as shown by Venkatakrishnan [Ventekakrishnan, Allmaras, Kamenetskii, & Johnson, 2003]. Applications with strong shocks and under resolved flow features become challenging for continuous finite element methods. For discontinuous Galerkin methods, higher order accuracy achieved within elements as in finite element methods, and element to element coupling exist through flux at element boundaries similar to finite volume methods. Hence; DG discretization permits a high order solver that may provide achieving robust, reliably accurate

¹ MS Student in Aerospace Engineering Department, Email: osman.gungor@metu.edu.tr

² Prof. in Aerospace Engineering Department, Email: serkan.ozgen@ae.metu.edu.tr

and efficient simulations. This paper is devoted to the solution of two dimensional Euler equations with DG-FEM.

Euler Equations

In the differential form, 2 dimensional Euler equations are given below:

$$\frac{\partial U}{\partial t} + \nabla \cdot F_i(U) = 0$$

where U is the conservative state vector

$$U = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ E \end{pmatrix}$$

where ρ is the fluid density, (u, v) are the Cartesian components of the velocity, *E* is the specific total energy which is composed of specific internal, *e*, and specific kinetic energy

$$E = e + \frac{1}{2}(u^2 + v^2)$$

and F_i is the inviscid flux tensor

$$F_i = (F_i^x, F_i^y)$$

$$F_i^{x} = \begin{pmatrix} \rho u \\ \rho u^2 + P \\ \rho uv \\ \rho Eu + Pu \end{pmatrix}, F_i^{y} = \begin{pmatrix} \rho v \\ \rho vu \\ \rho v^2 + P \\ \rho Ev + Pv \end{pmatrix}$$

One more equations is required to close the system. Further relations under the thermally and calorically perfect gas assumption are given for the total enthalpy, *H*, as follows:

$$H = E + \frac{P}{\rho}$$

and the equation of state provides

$$\frac{P}{\rho} = \frac{\gamma-1}{\gamma}(H - \frac{1}{2}(u^2 + v^2))$$

where γ is the ratio of specific heats and assumed as constant value of 1.4.

Non-dimensional Form of Euler Equations:

When working with dimensional variables, all variables will resolve in different order of magnitudes. This will result in ill conditioned linear systems of implicit time marching schemes, precision errors, etc. hence equations are normalized in order to catch the same order of magnitude. Non-dimensional form of the Euler equations reads

$$\bar{t} = \frac{v_{ref}t}{L_{ref}}, \bar{\rho} = \frac{\rho}{\rho_{ref}}, \bar{P} = \frac{P}{\rho_{ref}v_{ref}^2}, \bar{e} = \frac{e}{v_{ref}^2}, \bar{H} = \frac{H}{v_{ref}^2}, \bar{a} = \frac{a}{v_{ref}}, \bar{u} = \frac{u}{v_{ref}}, \bar{v} = \frac{v_{ref}}{v_{ref}}, \bar{v} = \frac{v_{ref}}{v_{r$$

The flow related reference values are taken from free-stream values, dimension related reference values are chosen according to problem to be solved. Finally,

$$\frac{\partial \overline{U}}{\partial \overline{t}} + \overline{\nabla} \cdot F_i(\overline{U}) = 0$$

METHOD

The DG method used to solve Euler equations is described in this section. Euler equations are only discretized in space using DG method in the present work. In theory, it is possible to use a space-time DG method discretization. References [Klaij, Vegt, & Ven, 2006] can be referred for example works of space-time DG method. DG discretization uses basis functions that are continuous within an element but discontinuous between elements. A nodal hierarchical basis is chosen in the present work. Differential form of Euler equations is multiplied by a test function Φ and integrated over the domain Ω to obtain:

$$\int_{\Omega} \Phi\left(\frac{\partial U}{\partial t} + \nabla \cdot F_i(U)\right) d\Omega = 0$$

This form is known as weighted residual form. Performing integration by parts on the advection term, weak form of the problem, which is the basic form of DG, is obtained:

$$\int_{\Omega} \Phi \frac{\partial U}{\partial t} d\Omega + \oint_{\partial \Omega} \Phi F_i(U) \cdot \vec{n} d\Gamma - \int_{\Omega} (\nabla \cdot \Phi) \cdot F_i(U) d\Omega = 0$$

Dividing domain into non-overlapping elements *E* and summation of elements read:

$$\sum_{E} \left[\int_{E} \Phi \frac{\partial U}{\partial t} d\Omega + \oint_{\partial E} \Phi F_{i}(U) \cdot \vec{n} d\Gamma - \int_{E} (\nabla \cdot \Phi) \cdot F_{i}(U) d\Omega \right] = 0$$

Local solution is assumed as:

$$u_h(\vec{x},t) = \sum_{k=1}^{N_p} \hat{u}_h(\vec{x}_k,t) l_k(\vec{x}) = \sum_{k=1}^{N_p} (u_h(t))_k b_k(\vec{x})$$

The local solution is represented in a two way; first one is nodal representation and second one is the modal representation. The two representations are mathematically equivalent, however computationally different and have certain advantages and disadvantages (see [Hesthaven & Warburton, 2008] for details). Using Vandermonde matrix, *V*, direct transformation between modal and nodal representations can be achieved.

$$Vu = \hat{u}, \quad V_{kj} = b_j(\vec{x}_k)$$

where \vec{x}_k is the nodal points in each element. l_k is multidimensional Lagrangian polynomial based on nodal points and b_k is the multidimensional polynomial basis of *N*th order with $N_p = (N+1)(N+2)/2$. Finally, as in general Galerkin approach, choosing test and basis functions from the same space, semi-discrete form is reached:

$$\underbrace{\frac{\partial}{\partial t} \int_{E} b_{k} u_{h} d\Omega}_{I} + \underbrace{\oint_{\partial E} b_{k} F_{i}(u_{h}) \cdot \vec{n} d\Gamma}_{II} - \underbrace{\int_{E} (\nabla \cdot b_{k}) \cdot F_{i}(u_{h}) d\Omega}_{III} = 0, 0 \le k \le n$$

The part *I* of the semi-discrete form is generally written as $M \cdot U$ where *M* is the element mass matrix and *U* is the vector composed of the solutions degrees of freedom. The mass matrix for modal representation is defined as:

$$M_{kj} = \int_E b_k b_j \, d\Omega$$

For an orthonormal basis function set, mass matrix becomes diagonal.

$$M_{kj} = \int_E b_k b_j \, d\Omega = 0, k \neq j$$

The part *II* of the semi-discrete form, which is physical flux, is approximated using a numerical flux, $H_i(u_h^-, u_h^+)$, due to the fact that solution is allowed to be discontinuous between elements.

$$\oint_{\partial E} b_k F_i(u_h) \cdot \vec{n} d\Gamma \approx \oint_{\partial E} b_k H_i(u_h^-, u_h^+) \cdot \vec{n} d\Gamma$$

$$\oint_{\partial E} b_k H_i(u_h^-, u_h^+) \cdot \vec{n} d\Gamma = \sum_{\substack{e \in \partial E \setminus \partial \Omega \\ inner fluxes (numerical flux)}} \left[\oint_{\partial E} b_k H_i(u_h^-, u_h^+) \cdot \vec{n} d\Gamma \right] + \sum_{\substack{e \in \partial E \cap \partial \Omega \\ boundary fluxes (boundary conditions)}} \left[\oint_{\partial E} b_k H_i^b(u_h^-, u_h^-) \cdot \vec{n} d\Gamma \right]$$

Notation $(\cdot)^-$ and $(\cdot)^+$ represents the values are taken from interior and exterior solutions, respectively, as depicted in Figure 1.



Figure 1 Interior and Exterior Cell Notation and Normal Vector (Landmann, 2008)

The part *III* of the semi-discrete form is evaluated using Gaussian quadrature. Since basis functions span ideal elements, Gaussian integration is evaluated in computational space. Quadrature points and weights are computed on equilateral triangle with equidistant points using barycentric coordinates. Equidistant points are transformed into Legendre-Gauss-Lobatto points to achieve exact integration and well-conditioned Vandermonde matrix (Hesthaven & Warburton, 2008). LGL quadrature points are mapped to simplex and presented in Figure 2.



Figure 2 Legendre-Gauss-Lobatto Quadrature Points

4 Ankara International Aerospace Conference

Basis Functions

The discrete DG solution u_h is expanded in a series of basis functions. Number of modes/nodes are chosen such that complete basis of order *N* is obtained. A hierarchical set of basis function is chosen. The basis function set is the Legendre polynomials which are special types of Jacobi polynomials valid in [-1, 1] and orthonormal. Jacobi polynomials are evaluated using following recurrence relations [Szegö, 1939]:

$$xP_{n}^{(\alpha,\beta)}(x) = a_{n}P_{n-1}^{(\alpha,\beta)}(x) + b_{n}P_{n}^{(\alpha,\beta)}(x) + a_{n+1}P_{n+1}^{(\alpha,\beta)}(x)$$

where:

$$a_n = \frac{2}{2n+\alpha+\beta} \sqrt{\frac{n(n+\alpha+\beta)(n+\alpha)(n+\beta)}{(2n+\alpha+\beta-1)(2n+\alpha+\beta+1)}}$$
$$b_n = -\frac{\alpha^2 - \beta^2}{(2n+\alpha+\beta)(2n+\alpha+\beta+2)}$$

Initial values of Jacobi polynomials are:

$$P_0^{(\alpha,\beta)}(x) = \sqrt{2^{-\alpha-\beta-1}\frac{\Gamma(\alpha+\beta+2)}{\Gamma(\alpha+1)\Gamma(\beta+1)}}$$
$$P_1^{(\alpha,\beta)}(x) = \frac{1}{2}P_0^{(\alpha,\beta)}(x)\sqrt{\frac{\alpha+\beta+3}{(\alpha+1)(\beta+1)}}((\alpha+\beta+2)x + (\alpha-\beta))$$

where $\Gamma(x)$ is the classic Gamma function [Abromowitz & Stegun, 1972]. $P_n^{(0,0)}(x)$ type of Jacobi polynomials are known as Legendre polynomials. The Jacobi polynomials are 1 dimensional, hence extension to multidimensional case is needed and complicated. Details of extensions to multidimensional case are given in [Koorwinder, 1975], [Suetin, 1999]. A complete set of basis functions on simplex for an order of 3 element is given in Figure 3.



Figure 3 Complete 2 Dimensional Basis Set for Order of 3 Element

Boundary Conditions

Slip Wall:

Slip wall requires that flow should be tangent to the wall. Hence, velocity normal to the wall should be zero and velocity tangent to the wall should be preserved.

$$\vec{u}^b \cdot \vec{n} = 0$$
$$\vec{u}^b \cdot \vec{t} = \vec{u}^- \cdot \vec{t}$$

Solving for \vec{u}^b provides boundary state as follows:

$$U^{b} = \begin{pmatrix} \rho \\ \rho^{-} u^{b} \\ \rho^{-} v^{b} \\ \rho^{-} w^{b} \\ E^{-} \end{pmatrix}$$

Far-field:

Far-field boundary condition uses the Riemann solver at the far-field boundary. Riemann invariants should remain constant for boundary state. Conditions of characteristics for subsonic and supersonic flow regime are given in Figure 4 for inflow and outflow boundaries.



Figure 4 Inflow and Outflow Characteristic Information Propagation Directions for Subsonic and Supersonic Flow [Burgess, 2011]

For subsonic flow regime, the Riemann invariants used to determine boundary state are: For inflow $(\vec{u}^- \cdot \vec{n} < 0)$:

$$R_{R} = \vec{u}^{-} \cdot \vec{n} + \frac{2c^{-}}{\gamma - 1}$$

$$R_{L} = \vec{u}^{\infty} \cdot \vec{n} - \frac{2c^{\infty}}{\gamma - 1}$$

$$R_{L} = \vec{u}^{\infty} \cdot \vec{n} - \frac{2c^{\infty}}{\gamma - 1}$$

$$R_{L} = \vec{u}^{\infty} \cdot \vec{n} - \frac{2c^{\infty}}{\gamma - 1}$$

$$R_{L} = \vec{u}^{\infty} \cdot \vec{n} - \frac{2c^{\infty}}{\gamma - 1}$$

$$s^{b} = \frac{p^{-}}{(\rho^{-})^{\gamma}}$$

$$(\vec{u} \cdot \vec{t})^{b} = \vec{u}^{\infty} \cdot \vec{t}$$

$$(\vec{u} \cdot \vec{t})^{b} = \vec{u}^{-} \cdot \vec{t}$$

For supersonic flow regime, the Riemann invariants used to determine boundary state are: For inflow $(\vec{u}^- \cdot \vec{n} < 0)$:

$$R_{R} = \vec{u}^{\infty} \cdot \vec{n} + \frac{2c^{\infty}}{\gamma - 1}$$

$$R_{L} = \vec{u}^{\infty} \cdot \vec{n} - \frac{2c^{\infty}}{\gamma - 1}$$

$$s^{b} = \frac{p^{\infty}}{(\rho^{\infty})^{\gamma}}$$

$$(\vec{u} \cdot \vec{t})^{b} = \vec{u}^{\infty} \cdot \vec{t}$$

$$R_{R} = \vec{u}^{-} \cdot \vec{n} + \frac{2c^{-}}{\gamma - 1}$$

$$R_{L} = \vec{u}^{-} \cdot \vec{n} - \frac{2c^{-}}{\gamma - 1}$$

$$s^{b} = \frac{p^{-}}{(\rho^{-})^{\gamma}}$$

$$(\vec{u} \cdot \vec{t})^{b} = \vec{u}^{-} \cdot \vec{t}$$

Boundary states are calculated as follows:

$$(\vec{u}\cdot\vec{n})^b = \frac{1}{2}(R_R + R_L)$$

6

Ankara International Aerospace Conference

$$c^{b} = 4(\gamma - 1)(R_{R} - R_{L})$$
$$\rho^{b} = \left(\frac{(c^{b})^{2}}{\gamma s_{b}}\right)^{1/\gamma - 1}$$
$$p^{b} = \frac{\rho^{b}(c^{b})^{2}}{\gamma}$$

where *c* and *s* are speed of sound and entropy. Solving for \vec{u}^b provides boundary state as follows:

$$U^{b} = \begin{pmatrix} \rho^{b} \\ \rho^{b} u^{b} \\ \rho^{b} v^{b} \\ \rho^{b} w^{b} \\ E^{b} \end{pmatrix}$$

Exact:

The boundary condition value is simply obtained from analytic solution.

$$u_h^+ = f(x, y, t)$$

RESULTS AND DISCUSSIONS

Isentropic Vortex

Isentropic vortex problem is very popular in CFD field to assess the performance of high order schemes [Spiegel, Huynh, & DeBonis, 2015]. The reason is being relatively simple problem and there is a known analytical solution at any time. Hence, solution error can be obtained easily. The isentropic vortex problem is defined by the following equation:

$$u = 1 - \beta e^{(1-r^2)} \frac{y - y_0}{2\pi}$$
$$v = \beta e^{(1-r^2)} \frac{x - x_0}{2\pi}$$
$$\rho = \left(1 - \left(\frac{\gamma - 1}{16\gamma\pi^2}\right)\beta^2 e^{2(1-r^2)}\right)^{\frac{1}{\gamma - 1}}$$
$$p = \rho^{\gamma}$$
$$r = \sqrt{(x - t - x_0)^2 + (y - y_0)^2}$$
$$x_0 = 5, y_0 = 0, \beta = 5, \gamma = 1.4$$

A square computational domain is used. The vortex is located at the center of the computational domain. Domain boundaries have equivalent distance to the vortex center. Computational domain extends from 0 to 10 in x-direction and from -5 to 5 in y-direction. In literature, grid refinement methodology is used to verify the numerical accuracy as the main purpose of problem. The series of grids are generated in the computational domain having 16, 32 and 64 nodes on the boundaries. Initially, grids are generated in structured grid fashion using boundary elements. The structured domain is diagonalized to obtain unstructured triangular grid. The triangular grids used in convergence study are shown in Figure 5. Simulations are conducted with polynomial orders from 1 to 5 for each grids which results in total of 15 simulations. Exact boundary conditions are employed at the domain boundaries which mean that analytical solution is enforced at boundary elements at each time step. Local Lax-<u>Friedrichs</u> flux method is utilized due to its convenient nature to low subsonic flows. The density error for each simulation is evaluated at time, t = 1. The density contour is presented in Figure 6 at initial condition, t = 0.5 and t = 1 for various orders.







Figure 6 The Density Solution at Time = 1 for Various Orders

The L2-norm of the density error is plotted in Figure 7 for each polynomial order in order to show the accuracy of the scheme. The x-axis is the grid size while y-axis presents the density error in logarithmic scale. The convergence rate for each polynomial order computed using change of error with change of grid size. Convergence rate for polynomial order of 1,2,3,4 and 5 is 1.52, 2.61, 3.17, 3.95 and 4.92 respectively. Theoretically, optimal convergence rate is $\sigma(h^{N+1})$, however convergence rate is observed to be around $\sigma(h^{N+1/2})$ which is suboptimal convergence rate.



Figure 7 Convergence Rates for Solution Orders 1 to 5

Bump in a Channel

Inviscid smooth bump in a channel problem is a popular test case for high order CFD methods. Recently at 5th High Order CFD Workshop 2018, it is assigned to participants to test their solvers. The smooth bump test case aims to test high order CFD methods with curved boundary representation for the computation of internal flows. The flow through channel is subsonic with a Mach number of 0.5. The bump profile in the lower wall is given by an equation and has smooth variation. The analytical solution of the problem is unknown however; since the flow is subsonic and inviscid, entropy should be constant in the channel. Hence, L2 norm of entropy error given can be used as the indication of accuracy.



Figure 8 Numerical Model of Bump in a Channel Problem

The computational domain presented in Figure 8 is bounded between x = -1.5 and x = 1.5, and between y = 0 and y = 0.8. The inlet boundary is defined at x = -1.5 while outlet is defined at x = 1.5. The bump geometry is placed at the point [0,0] which is at equivalent distance to inlet and outlet boundaries. The subsonic inlet boundary condition is defined with total pressure, total temperature and Mach number. The static pressure is set at outflow boundary. Upper and lower boundaries are slip wall boundaries and bump geometry is represented with curved wall boundary approach. The flow is started from uniform M = 0.5 flow. HLL approximate Riemann

solver is selected as numerical flux function. Time integration is carried out by fourth order explicit Runge-Kutta method.



Figure 9 Sequence of Grids in Bump in a Channel Problem

Three meshes different in element sizes are generated to use in accuracy analysis. Coarse mesh has 667 elements. 1226 elements are utilized in medium mesh while fine mesh has 2444 elements. Coarse, medium and fine meshes are presented in Figure 9. The series of simulations are conducted with three meshes for polynomials order from 1 to 4 until convergence in entropy error achieved. Entropy error histories of coarse mesh solutions are presented in Figure 10 where x-axis is iteration number and y-axis is log scale of entropy error. The entropy error for all polynomial orders settled to a certain level. The low order simulations quickly converged however, error reduction capabilities are limited. The high order simulations took longer time to converge with oscillatory behavior.





The pressure contours from two different polynomial order on coarse mesh are presented in Figure 11. The pressure contour is ranged between 0.82 and 1.04. The outlet boundary condition is assumed to be undisturbed flow and has the pressure value of 1.0. Hence, pressure values lower than 1.0 represents suction regions. The solution presented in Figure 11.(a) uses elements with polynomial order of 1 while Figure 11.(b) presents the same solution for polynomial order of 4. The pressure field in Figure 11.(a) has discontinuities at regions high pressure gradient presents. Moreover, suction region at the bump peak is poorly captured. On the other hand, fourth order polynomials provide smooth well developed solution. Suction and high pressure regions are well captured. It can be summed up that high order solution can provide smooth solution on even coarse mesh while low order solution has disturbed regions.



(a) Second Order DG Scheme, N=1



(b) Fifth Order DG Scheme, N=4

Figure 11 Steady State Pressure Contours of (a) Second and (b) Fifth Order Solutions

The entropy error of each simulation is plotted with grid size in Figure 12. The convergence rates are observed between $\sigma(h^{N+1/2})$ and $\sigma(h^{N+1})$. The expected convergence rate in High Order CFD Workshop is stated as $\sigma(h^{N+1})$ hence it can be concluded that consistent results are obtained.



Figure 12 Convergence Rate for Bump in Channel Problem

Bump with Straight Cells:

In high order DG simulations, the use of curved wall boundary representation is stated as mandatory to achieve high order accurate solution. In order to experience importance of curved wall boundary condition, the bump in a channel solutions are computed with polynomial orders of 2 and 4 using straight sided cells on wall boundaries. The Figure 12 is re-drawn to compare curved and straight sided cells solution on Figure 13. As it is aforementioned, convergence rate is expected to be $\sigma(h^{N+1})$. For the third order accurate solution, convergence rate drops from 3.08 to 1.69 and for the fifth order solution, convergence rates are 4.88 and 1.80. The convergence rate of straight sided cell simulations is around 2 regardless of polynomial order. Other than first order solutions, geometry representation of straight sided cells is identical to second order boundary representation of curved cells. Hence, it can be said that order of accuracy is limited by representation of wall boundaries.



Figure 13 Convergence Rate Comparison of Curved and Straight Sided Cell Solutions

In order to further investigate the accuracy loss, entropy generation in solution domain is calculated for each simulation. The maximum entropy generation in each simulation occurred around the bump however, away from the bump geometry in solution domain, generation of entropy is vanished. The entropy contours of simulations with curved and straight sided wall are compared in Figure 14 zooming at bump geometry. The maximum and minimum values of each contour plot is set equivalent. The grid and Legendre-Gauss-Lobatto nodes are also activated. The grid nodes are connected with solid lines while LGL nodes are connected dashed lines to show deformation of cell edges and node blending. In each simulation, maximum entropy generation occurs at the bump peak; however, simulations with straight sided cells have greater entropy generation.

Another important observation is that, entropy error of curved cell simulations have smooth variation and diminish away from bump. However, in straight sided cell simulations, entropy error generated at bump geometry convected downstream and disturbed solution field.



Figure 14 Entropy Error Contours, (a) Curved Cell with Polynomial Order of 2, (b) Straight Sided Cell with Polynomial Order of 2, (c) Curved Cell with Polynomial Order of 4, (d) Straight Sided Cell with Polynomial Order of 4

RAE2822 Airfoil

The RAE2822 airfoil is a transonic airfoil which has a maximum camber of %2 positioned at %80 chord length and %22 maximum thickness to chord length ratio. The RAE2822 airfoil has a well-documented test campaign which made it popular test case in computational fluid dynamics literature. The upper and lower surface coordinates of the airfoil and experimental results are obtained from test document [Cook, McDonald, & Firmin, 1979]. The airfoil geometry is plotted in Figure 15.



Figure 15 RAE2822 Airfoil Geometry

The solution domain for RAE2822 is generated between airfoil geometry and farfield boundary. Leading edge of the airfoil is located at [0,0]. Curved wall boundary is applied to airfoil geometry. Farfield is generated as circular geometry with radius equal to 40 chord length and center at [0,0]. The number of elements placed on the airfoil geometry is 120 while farfield boundary is divided into 40 equal elements. The domain between airfoil and farfield boundaries

contains 1846 triangular elements. The generated grid is presented in Figure 16.(a). The zoomed viewed of grid around airfoil geometry is plotted on Figure 16.(b). HLL approximate Riemann solver is employed as numerical flux function.



Figure 16 Solution Grid Used in RAE2822 Airfoil Simulations (a) Solution Grid, (b) Zoomed View on Airfoil Geometry

RAE2822 Airfoil at Mach 0.3:

The RAE2822 test campaign does not include low Mach number flows. However, an inviscid shock-free flow around RAE2822 airfoil at zero angle of attack should produce no drag force due to fact that pressure forces cancel in stream-wise direction. Hence, verification study can be carried out.

Verification analyses are conducted with polynomial order of 3. Simulation is run until convergence achieved. The density residual and drag force history is plotted in Figure 17. Iteration number is used as x-axis while density residual and drag force are plotted on y-axis. Logarithmic scale is applied to the y-axis of density residual plot. The density residual is reduced to around order of -11. Similarly, drag force initially oscillates and converges to value of 1.18E-05 which is almost zero.



(a) Density Residual Plot

(b) Drag Force Convergence History

Figure 17 Convergence History for RAE2822 Airfoil Solution at M = 0.3 (a) Density Residual Plot, (b) Drag Force Convergence History

The pressure contour of converged solution is presented in Figure 18.(a). Pressure contour is ranged between 0.97 and 1.06 and spaced equally 21 lines. Figure 18.(b) shows Mach number contours which is also equally spaced 20 lines between values of 0.0 and 0.38. The contour

plots show smooth variation of flow variables in solution domain. Moreover, drag force is obtained near zero as it is aimed. It can be concluded that numerical approach is verified.



Figure 18 RAE2822 Solution at M = 0.3, (a) Pressure Contour, (b) Mach Number Contour

RAE2822 Airfoil at Mach 0.73:

The 2 transonic flow test cases of RAE2822, Case9 and Case10 [Cook, McDonald, & Firmin, 1979], has actually become a standard test case for turbulence modelling. Case9 is run at Mach number of 0.73 and 2.8 degrees angle of attack. The case9 is evaluated as subcritical flow condition where little to no separation occurs due to shock. However, Case10 is supercritical flow condition where massive separation is observed downstream of shock. Since the fidelity present work is limited by governing equations, Case9 can be used as validation case.

The same grid and numerical flux used in low Mach number verification case is utilized. However, since there is a presence of shock, high gradient values in the solution field are expected. Therefore, slope limiter is utilized to stabilize the solution.

The transonic test case simulations are conducted with polynomial orders of 1. The density residual history is plotted in Figure 19.(a). Pressure values on the airfoil geometry are extracted from converged solution. The pressure values are converted to pressure coefficient and compared with experimental results in Figure 19.(b). The simulation and experimental results match quite well in the lower surface However, results are less successful in capturing the upper surface vales. Nonetheless, shock positon is correctly captured. Leading edge region of upper surface shows notable discrepancy which may appear due to low resolution of surface curvature. Downstream of the shock, results are satisfactory.



Figure 19 RAE2822 Solution with Slope Limiter at M = 0.73 and α = 2:82, (a)

Density Residual, (b) Pressure Coefficient [Cook, McDonald, & Firmin, 1979]

The pressure and Mach number contours of the converged solution around the airfoil geometry are presented in Figure 20.(a) and Figure 20.(b), respectively. The pressure contour is divided into equal 20 levels between 0.6 and 1.4. Mach number contour is consists of 25 levels with maximum value of 1.2 and minimum value of 0.0. The air accelerates on the upper surface resulting in shock formation. However, lower surface has smooth variation in flow field.



(a) Pressure Contour

(b) Mach Number Contour

Figure 20 RAE2822 Solution with Slope Limiter at M = 0.73 and α= 2:82, (a) Pressure Contour, (b) Mach Number Contour

References

- Abromowitz, M., & Stegun, I. A. (1972). Handbook of Mathematical Functions. New York: Dover Publications.
- Burgess, N. K. (2011). An Adaptive Discontinuous Galerkin Solver for Aerodynamic Flows. University of Wyoming.
- Cook, P., McDonald, M., & Firmin, M. (1979). Aerofoil RAE 2822 Pressure Distributions, and Boundary Layer and Wake Measurements.
- Hesthaven, J. S., & Warburton, T. (2008). Nodal Discontinuous Galerkin Methods. Springer.
- Jameson, A. (1983). Solution of the Euler Equations for Two Dimensional Transonic Flow by a Multi Grid Method. Applied Mathematics and Computation, 13:327-355.
- Jameson, A., Schmidt, W., & Turkel, E. (1981). Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes. AIAA 14th Fluid and Plasma Dynamic Conference. Pablo Alto.
- Klaij, C. M., Vegt, J. J., & Ven, H. v. (2006). Space-Time Discontinuous Galerkin Method for the Compressible Navier-Stokes Equations. *Journal of Computational Physics*, 217(2), 589-611.
- Koorwinder, T. (1975). Two-Variable Analogues of the Classical Orthogonal Polynomials in Theory and Application of Special Functions. New York: Academic Press.
- Landmann, B. (2008). A Parallel Discontinuous Galerkin Code for the Navier-Stokes and Reynolds-Averaged Navier-Stokes Equations. University of Stuttgart - Institute for Aerodynamics and Gas Dynamics.
- Murman, S. M., Diosady, L. T., Garai, A., & Ceze, M. (2016). A Space-Time Discontinuous-Galerkin Approach for Separated Flows. *AIAA Aerospace Sciences Meeting*. San Diego, California.
- Spiegel, S. C., Huynh, H. T., & DeBonis, J. R. (2015). A Survey of the Isentropic Euler Vortex Problem using High-Order Methods. Cleveland: NASA Glenn Research Center.
- Suetin, P. K. (1999). Orthogonal Polynomials in Two Variables. Amsterdam: Gordon and Breach Science Publishers.
- Szegö, G. (1939). Orthogonal Polynomials. Rhode Islands: American Mathematical Society Providence.
- Ventekakrishnan, V., Allmaras, S. R., Kamenetskii, D. S., & Johnson, F. T. (2003). Higher Order Schemes for the Compressible Navier-Stokes Equations. AIAA Computational Fluid Dynamics Conference . Orlando.