A COMPARATIVE STUDY FOURTH ORDER RUNGE KUTTA-TVD SCHEME AND FLUENT SOFTWARE CASE OF INLET FLOW PROBLEMS

MOHAMED ABDALWAHAB ALTAHER MOHAMED

A project report submitted in partial fulfilment of requirement for the award of the Degree of Master of engineering

Faculty of mechanical and manufacturing engineering

University Tun Hussein Onn Malaysia

JANUARY 2011

ABSTRACT

Inlet as part of aircraft engine plays important role in controlling the rate of airflow entering to the engine. The shape of inlet has to be designed in such away to make the rate of airflow does not change too much with angle of attack and also not much pressure losses at the time, the airflow entering to the compressor section. It is therefore understanding on the flow pattern inside the inlet is important. The present work presents on the use of the Fourth Order Runge Kutta – Harten Yee TVD scheme [1, 2] for the flow analysis inside inlet. The flow is assumed as an inviscid quasi two dimensional compressible flow. As an initial stage of computer code development, here uses three generic inlet models. The first inlet model to allow the problem in hand solved as the case of inlet with expansion wave case. The second inlet model will relate to the case of expansion compression wave. The last inlet model concerns with the inlet which produce series of weak shock wave and end up with a normal shock wave. The comparison result for the same test case with Fluent Software [3] indicates that the developed computer code based on the Fourth Order Runge Kutta – Harten – Yee TVD scheme are very close to each other. However for complex inlet geometry, the problem is in the way how to provide an appropriate mesh model.

ABSTRAK

Masukan merupakan sebahagian kompenan pada enjin pesawat yang memainkan peranan penting dalam mengawal kadar aliran laju udara yang masuk ke dalam enjin. Rekabentuk masukan yang digunakan sebagai aliran masuk udara ke bahagian pemampat harus direka sedemikian untuk memastikan kadar aliran laju udara tidak berubah terlalu banyak terhadap sudut yang bertindak dan juga kehilangan tekana pada masa tersebut dapat dikurangkan. Oleh kerana itu pemahaman yang lebih mendalam tentang pola aliran di dalam masukan harus difahami dengan lebih lanjut. Dalam kajian ini, analisis yang digunakan untuk mengkaji aliran dalam masukan menggunakan analisis skima Fourth Order Runge Kutta – Harten Yee TVD. Aliran diandaikan sebagai aliran kebolehmampat dua demensi kuasi tidak likat. Pada peringkat awal dalam pembanguanan kod komputer, tiga jenis model generik masukan digunakan. Bagi model inlet jenis pertama, ianya digunakan untuk menganalisis bahagian masukan yang menggunakan gelombang pengembangan manakala modul jenis yang kedua pula menggunakan gelombang mampatan pengembangan dan akhir sekali iaitu modul jenis ketiga yang akan menghasilkan siri gelombang kejut lemah dan ianya akan berakhir dengan gelombang kejut normal. Setelah kajian ini selesai dijalankan, keputusan analisis yang diperolehi akan dibandingkan dengan keputusan analisis yang menggunakan Fluent Software dan didapati bahawa kedua-dua keputusan ini mempunyai keputusan yang hampir sama. Walaubagaimanapun, masalah yang wujud dalam menggunakan analisis skima Fourth Order Runge Kutta – Harten Yee TVD ialah kaedah untuk menghasilkan model mesh yang sesuai.

LIST OF CONTENTS

LIST OF TABLES

LIST OF APPENDIXS

LIST OF FIGURES

LIST OF SYMBOLS

∂	Differentiation
ρ	Static density
t	Time
υ, ω, u	Cartesian velocity components
p	Pressure
T	Temperature
E	Flux vector
Q	The conserved flow variables
x, y	distance along the reference x, y-axis
F, G	Flux in Cartesian coordinates
e	Total energy
$\boldsymbol{\tau}$	Non dimensional time in transformed coordinate system
η, ξ	Local coordinates.
J	Jacobin transformation
M	Mach number
v	Velocity
ɑ	Speed of sound
β	oblique shock angle
γ	specific heat ratio
δ	deflection angle
θ	Prandtl – Meyer angle
TV	tot.al variation
ϕ	Vector of linearized characteristic variables
α	Difference of characteristic variable

ψ	Numerical viscous derivative function
ε	Entropy correction parameter.
R	Eigen vector
A, B	Jacobian matrix

CHAPTER 1

INTTRODUCTION

The intake is important part in the aircraft engine because it will give the engine a proper supersonic air flow, the air intake is that part of an aircraft structure by means of which the aircraft engine is supplied with air taken from the outside atmosphere. The air flow enters the intake and is required to reach the engine face with optimum levels of total pressure and flow uniformity. These properties are vital to the performance and stability of engine operation. Depending on the type of installation, this stream of air may pass over the aircraft body before entering the intake properly.

During flight, there is various flight manoeuvres had to be done such as take off, landing stage or flight turning for flight change directions. Such flight conditions had made the aircraft are operated at different angle of attack. As result, the airflow which entering to the engine may make a certain angle of attack with respect to the main engine axis system. In appropriate inlet design may bring to the situation the rate of mass flow entering to the engine is not sufficient and the engine loss the thrust. Dropping rate of mass flow may due to a strong flow separation occurred in the inlet or may due to the presence of strong shock wave (Jack D. Mattingly).

The presence work aim the aerodynamics analysis in the presence of shock phenomena. . To capture shock phenomena for the case of flow past through a complex geometry one has to use at least two dimensional Euler equations as its governing equation of fluid motion for the flow problem in hand. Then the question is what the Euler equations are? The Euler equations first appeared in published form in Eulers article “Principes generaux du mouvement des fluides,” published in Mémoires de l'Academie des Sciences de Berlin in 1757(Batchelor, G. K.). They were among the first partial differential equations to be written down. At the time Euler published his work, the system of equations consisted of the momentum and continuity equations, thus it was underdetermined except in the case of an incompressible fluid. An additional equation, which was later to be called the adiabatic condition, was supplied by Pierre-Simon Laplace in 1816.

During the second half of the 19^{th} century, it was found that the equation related to the conservation of energy must at all times be kept, while the adiabatic condition is a consequence of the fundamental laws in the case of smooth solutions. With the discovery of the special theory of relativity, the concepts of energy density, momentum density, and stress were unified into the concept of the stress-energy tensor, and energy and momentum were likewise unified into a single concept, the energy-momentum vector.

In fluid dynamics, the Euler equations govern inviscid flow. They correspond to the Navier–Stokes equations with zero viscosity and heat conduction terms. They are usually written in the conservation form (Batchelor, G. K.).

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \mathrm{u}) = 0$$

$$\frac{\partial \rho \mathrm{u}}{\partial t} + \nabla.\big(u \times (\rho \mathrm{u})\big) + \nabla p = 0$$

$$\frac{\partial E}{\partial t} + \nabla.\big(\mathrm{u}(E + p)\big) = 0$$

to emphasize that they directly represent conservation of mass, momentum and energy. The equations are named after Leonhard Euler. The Euler equations can be applied to compressible as well as to incompressible flow using either an appropriate equation of state or assuming that the divergence of the flow velocity field is zero There are various numerical method had been developed for solving such kind equation such as Beam Warming Scheme, Mac Cormack Scheme, Steger Warming Scheme. The present work will use a Total Variation Diminishing - Runge Kutta scheme as suggested by Yee-Harten (C.HIRSCH).

This method treats the steady flow problem as unsteady flow problem in order to make the Euler Equation to behave as a hyperbolic partial differential equation with respect to time. Their result will be compared to the result from the

Fluent software. Fluent is the CFD solver which able to solve complex flows ranging from incompressible (low subsonic) to mildly compressible (transonic) to highly compressible (supersonic and hypersonic) flows. This software also provides various options in term of type of solver [explicit/implicit temporal discretization, upwind], will in term of grid generation this software also offer the use of multi grid method to enhance the convergence level. Beside that the FLUENT software also is to deliver an optimum solution efficiency and accuracy for a wide range of speed regimes. The wealth of physical models in FLUENT allows to accurately predict laminar and turbulent flows, various modes of heat transfer, chemical reactions, multiphase flows, and other phenomena with complete mesh flexibility and solution-based mesh adaption.

1.1 Objective:

To develop a computer code for aerodynamics analysis related to shock phenomena inside of the engine inlet.

1.2 Scope of Study:

(i) Developing computer code for grid generation of flow domain by using an Algebraic grid generator approach.

(ii) Develop computer code Euler solver based on Total variation Diminishing – Runge-Kutta scheme.

(iii) Generate Mesh Flow Domain for Fluent Software by using Gambit Mesh Generator Software.

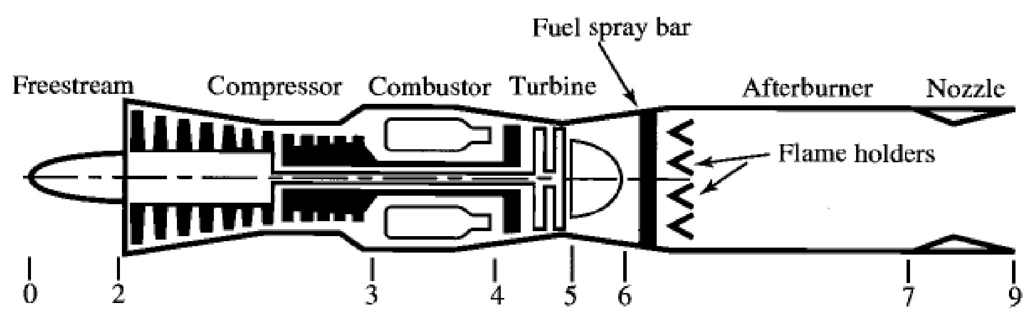
(iv) Comparative study between the result of developed computer code result and Fluent for several test case related to the inlet flow problems.

CHAPTER 2

THE GOVERNING EQUATION OF FLUID MOTION

2.1 Physical Flow Phenomena of Flow Inside Inlet Aircraft Engine.

An engine's air inlet duct is generally considered as an airframe part. During flight operation, it is very important to the engine performance. Engine thrust can be high only if the inlet duct supplies the engine with the required airflow at the highest possible pressure. (Figure 2.1) Show a physical configuration of the turbojet engine of aircraft, where from the position 0 to 2 represent the inlet part of engine will be placed.



. Figure 2.1: 2D engine of air craft.

Figure (2.2) show rather detail of inlet in two dimensional drawing

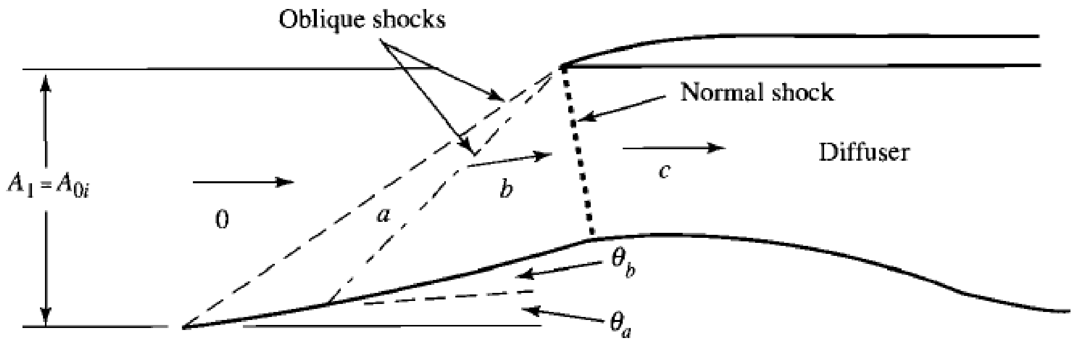


Figure 2.2: Technical drawing of the inlet in two dimensional drawing

As part of aircraft engine, the inlet duct has two engine functions and one aircraft function, namely:

(i) It must be able recover as much of the total pressure of the free air stream as possible and deliver this pressure to the front of the engine compressor.

(ii) The duct must deliver air to the compressor under all flight conditions with a little turbulence.

(iii) The aircraft is concerned, the duct must hold to a minimum of the drag.

The duct also usually has a diffusion section just ahead of the compressor to change the ram air velocity into higher static pressure at the face of the engine. This is called ram recovery. The inlet duct is built generally in the divergent shape (subsonic diffuser). However when the aircraft begins to fly at or near the speed of sound., the shock waves are developed in which, if it is not controlled, it will give a high pressure loss and decreasing of mass flow rate, and beside that it will also set up vibrating conditions in the inlet duct called inlet “buzz “. Buzz is an airflow instability caused by the shock wave rapidly being alternately swallowed and expelled at the inlet of the duct. Air enters the compressor section of engine must be slow down to subsonic velocity. At supersonic speeds the inlet does the job by slowing the air with minimize energy loss and the temperature rise. At transonic speeds the inlet duct is designed to keep shock waves out of the duct (Jack D. Mattingly).

This is done by locating the inlet duct behind a spike or probe which creates the shock wave in front of inlet duct. This normal shock wave will produce a pressure rise and velocity decrease to subsonic speeds. At higher Mach numbers, the single very strong normal shock wave may create and causes a great reduction in the total pressure in the duct inlet can be recovered and excessive air temperature rise inside the duct can be avoided. To avoid the presence of single strong normal shock wave, the inlet may be designed to generate a series of oblique shock waves. Hence the reduction of flow speed can be carried out gradually without loss to much in the stagnation pressure drop. To keep that the flow at the moment entering to the compressor section is a subsonic, a normal shock is created just in front of the compressor.

2.2 Governing Equation of Fluid Motion of Two dimensional Unsteady Inviscid Compressible Flows.

To ovoid the excessive pressure drop, inlet duct is normally designed to have a streamline surface, as result the flow pass through inlet can be considered as inviscid flow. However considering that the most airplanes are operated at relatively high speed the compressible effect have to be taken account. The real physical flow phenomena around the inlet engine of the aircraft are actually three dimensional flow phenomena. However this flow problem can be considered as the flow problem of a simple flow over a complex geometry. To reduce the complexity of the flow problem, especially, in the stage of computer code development, the flow problem around the inlet may be considered as the case of two dimensional planar flows.

The governing equation of motion for the two-dimensional inviscid flow is:

$$\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} = 0 \qquad (2.1)$$

Where:

$$Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho e_t \end{bmatrix} \qquad (2.2)$$

$$E = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho u v \\ (\rho e_t + p)u \end{bmatrix} \qquad (2.3)$$

$$F = \begin{bmatrix} \rho v \\ \rho v u \\ \rho v^2 + p \\ (\rho e_t + p)v \end{bmatrix} \qquad (2.4)$$

The equations of motion were transformed from physical space(x, y) to computational space$(\xi, \eta)$by the following relations. With the generalized coordinate transformation:

$$\tau = t \qquad (2.5)$$

$$\xi = \xi(t, x, y) \qquad (2.6)$$

$$\eta = \eta(t, x, y) \qquad (2.7)$$

The chain rule of partial differentiation provides the following expressions for the Cartesian derivative:

$$\frac{\partial}{\partial t} = \frac{\partial}{\partial \tau} + \xi_t \frac{\partial}{\partial \xi} + \eta_t \frac{\partial}{\partial \eta} \qquad (2.8)$$

$$\frac{\partial}{\partial x} = \xi_x \frac{\partial}{\partial \xi} + \eta_x \frac{\partial}{\partial \eta} \qquad (2.9)$$

$$\frac{\partial}{\partial y} = \xi_y \frac{\partial}{\partial \xi} + \eta_y \frac{\partial}{\partial \eta} \qquad (2.10)$$

Where:

$$\xi_t = \frac{\partial \xi}{\partial t}, \eta_t = \frac{\partial \eta}{\partial t}$$
$$\xi_x = \frac{\partial \xi}{\partial x}, \eta_x = \frac{\partial \eta}{\partial x}$$
$$\xi_y = \frac{\partial \xi}{\partial y}, \eta_y = \frac{\partial \eta}{\partial y}$$

The Euler equation in the transformed coordinate, Eq. (2.1) can be written as:

$$\frac{\partial \bar{Q}}{\partial \tau} + \frac{\partial \bar{E}}{\partial \xi} + \frac{\partial \bar{F}}{\partial \eta} = 0 \qquad (2.11)$$

Where:

$$\bar{Q} = \frac{Q}{J} \qquad (2.12)$$

$$\bar{E} = \frac{1}{J}\left[\xi_t Q + \xi_x E + \xi_y F\right] \qquad (2.13)$$

$$\bar{F} = \frac{1}{J}\left[\eta_t Q + \eta_x E + \eta_y F\right] \qquad (2.14)$$

2.3 Overview Various Methods For Solving The Euler Equations.

The Governing equation in the form of unsteady Compressible Euler equation can be classified as a hyperbolic partial differential system equation. There are various method had been developed for solving such type equation. They are namely: Mac Cormack Scheme, Beam Warming Scheme and lax Wendroff scheme.

Mac Cormack's (K_A.Hoffmann) technique is a variant of the Lax-Wendroff approach but is much simpler in its application. Like the Lax-Wendroff method, the Mac Cormack method is also an explicit finite-difference technique which is second-order-accurate in both space and time. First introduced in 1969, it became the most popular explicit finite-difference method for solving fluid flows for the next 15 years. Today, the Mac Cormack method has been mostly supplanted by more sophisticated approaches. However, the Mac Cormack method is very "student friendly;" it is among the easiest to understand and program. Moreover, the results obtained by using MacCormack's method are perfectly satisfactory for many fluid flow applications. It is an excellent method for introducing the fresh learner to the joys of CFD. For various study of solving Euler Equation the study will cover some of it as:

T. H. Pulliam "*Solution Methods in Computational Fluid Dynamics*" for Implicit finite difference schemes for solving two dimensional and three dimensional Euler and Navier-Stokes equations. In this study they concentrated on the Beam and Warming implicit approximate factorization algorithm in generalized coordinates. And they had some examples for 2-D inviscid and viscous calculations (e.g. airfoils with a deflected spoiler, circulation control airfoils and unsteady buffeting) and also 3-D viscous flow are included. When they used the Beam and Warming implicit approximate factorization scheme or variants of that scheme such as the diagonalization. The codes employ improvements to enhance accuracy, (grid refinement, better boundary conditions, more versatile artificial dissipation model) and efficiency (diagonal algorithm, implicit treatment of artificial dissipation terms, variable time steps). Results for a wide variety of cases substantiate the accuracy and efficiency claims.

H.C.Yee and P.Kutler "Application of Second-Order-Accurate Total Variation Diminishing (TVD) Scheme to the Euler Equations in General Geometries" they fined the TVD schemes have the property of not generating spurious oscillations for one dimensional nonlinear scalar hyperbolic conservation laws and constant coefficient hyperbolic system, so the applications of these methods to one and two-dimensional fluid flows containing shocks (in Cartesian coordinates) yields highly accurate non oscillatory numerical solutions, so the goal of this study was to extend these methods to the multidimensional Euler Equations in generalized coordinate systems then they got from numerical experiments the scheme is stable in a strong nonlinear sense the calculation with an incident shock Mach number of 10 and the report is the first attempt to apply the TVD scheme to non-Cartesian coordinates. It is preliminary in nature.

J.Shi and E.F.Toro(1993)"Fully Discrete High-Order TVD schemes for A scalar Hyperbolic Conservation Low" The investigated fully discrete high-order TVD schemes for a scalar hyperbolic conservation law and they used flux limiters, then they fined the courant-number dependent TVD region for second and third-order scheme have been theoretically established. Flux limited have been proposed and tested via numerical experiments. For methods of m-th order accuracy (m≥4) they proposed a semi-empirical limiting procedure that appears to work well. Test on the case of m=4 give very satisfactory results.

Dongfang Liang, Roger A. Falconer and Binliang Lin "Comparison between TVD-MacCormack and ADI-type solvers of the shallow water equations" in this study they comparison between the TVD-MacCormack model and an alternating direction implicit (ADI) model for cases involving steep-fronted shallow flows. It is demonstrated that the ADI model is unable to predict trans-critical flows correctly, and artificial viscosity has to be introduced to remove spurious oscillations. The TVD-MacCormack model reproduces all flow regimes accurately.

Finally, the TVD-MacCormack model is used to predict a laboratory-scale dyke break undertaken at Delft University of Technology. The predictions agree closely with the experimental data, and are in excellent agreement with results from an alternative Godunov-type model. Thin they got the difference between the two formulations can be significant in the numerical solutions for flows over uneven bottom topographies. The conventional formulation was found to be less accurate than the deviatoric formulation. However, local disturbances in the discharge could not be totally eradicated using the present TVD-MacCormack model for steady flows with abrupt changes. And the performance of the TVD-MacCormack model was also compared with a typical 2-D ADI model. The results showed that the 2-D ADI model did not have the shockcapturing capability, which is particularly necessary for modeling dam-break, levee-breach and steep riverine flows. Spurious oscillations arose in the solutions near sharp variations, and the steady hydraulic jump could not be predicted using the 2-D ADI model.

These numerical difficulties could not be overcome by simply increasing the artificial viscosity. Through comparisons between different formulations of SWEs and with the ADI model, the present TVD-MacCormack model has been validated for a range of flow conditions, including 1-D and 2-D, steady and unsteady, hypothetical and realistic, sub-, super- and trans-critical flows. Besides, a further validation was undertaken against a complicated dyke-break experiment. The predicted hydrographs compared well with the measured data. It should also be noted that the computational grids used in this paper were relatively coarse. Since the computational results were encouraging, no finer grid resolutions were considered. The simulations were all completed within 1 minute on a Pentium 4 personal computer (3.2 GHz CPU, 2G RAM) even for the dyke-break case with 315 · 81 grid cells. The high efficiency and robustness of the present model make it a powerful tool for real-time flood predictions.

2.4 Solution of simple inlet problem by the use of Fluent Software

Present work will study three cases of geometries which related to the inlet problem, they namely (1) a simple compression wave inlet model , (2) simple compression/expansion inlet model and (3) simple inlet configuration with wedge nose model. To allow this problem can be analyzed by use of Fluent software, firstly one has to create a mesh flow domain. The mesh generation can be made by using Gambit software. The combination of Gambit and Fluent had made the flow analysis for this inlet problem can be divided into six steps. Those steps can be explained as follows:

2.4.1 Simple inlet model compression wave.

2.4.1.1 Create geometry in Gambit:

Creating geometry for channel as shown in figure (2.3)

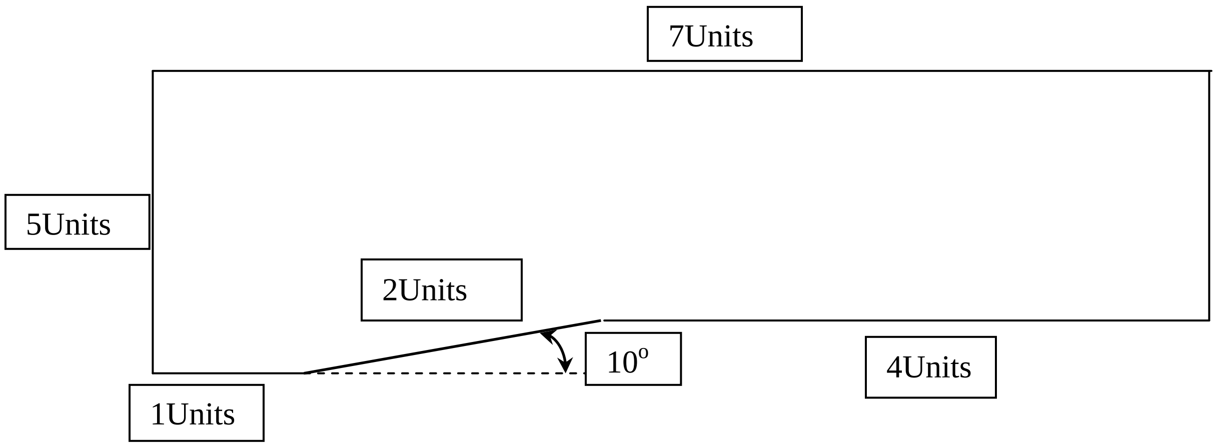


Figure 2.3: Geometry of simple inlet of airplane engine.

Open Gambit software first to create this simple geometry, so define the coordinates needed first from the table (2.1) as shown in figure (2.4),

Table 2.1: Coordinates of the geometry.

Label	X	Y	Z
A	0	0	0
B	0	5	0
C	1	5	0
D	3	5	0
E	7	5	0
F	7	0.353	0
G	3	0.353	0
H	1	0	0

Fig 2.4 coordinates of the geometry.

This coordinates need to be as wedge so link up those coordinates to make the edges as shown in figure (2.5)

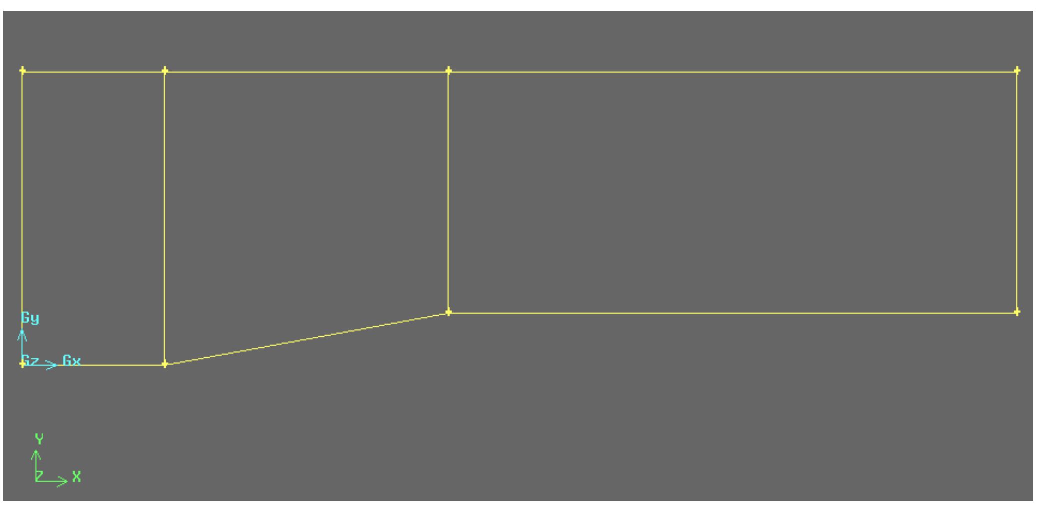


Fig 2.5: Link up the coordinates of the geometry.

Now while creating the edges, those edges need to define as faces. So define the (ABCH) as face1, (CDGH) as face2 and (DEFG) as face3 now after created the faces as in fig (2.6) the geometry ready to mesh.

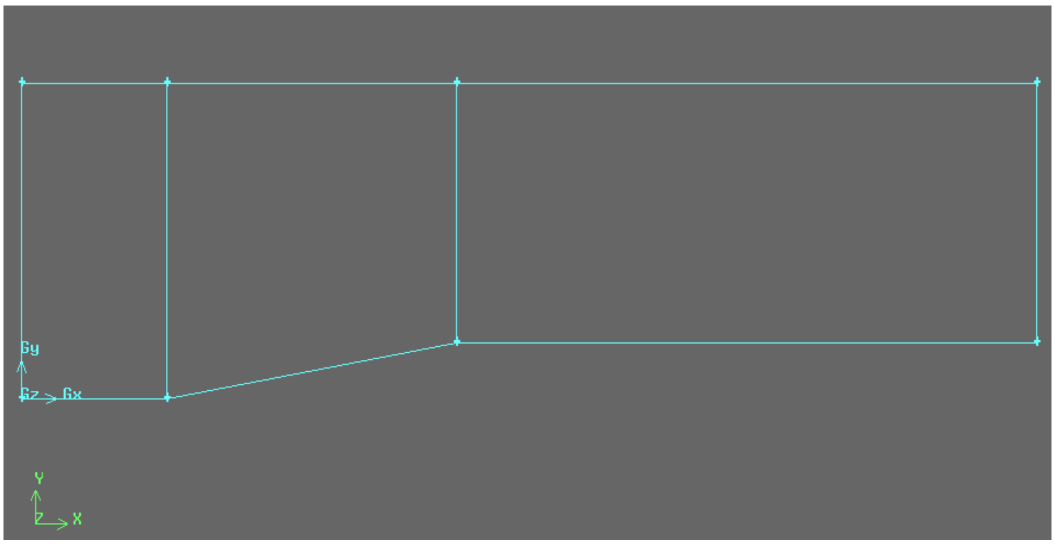


Fig 2.6: Creating the faces of the geometry of the geometry.

2.4.1.2 Mesh geometry in Gambit:

Now the geometry needs to mesh each of the 3 faces separately to get the final mesh. Before meshing the faces, need to define the point of distribution for each of the edges that form the face. firstly select the first wedge from the first face and define the point of distribution on the wedge then select the second wedge from the face one and define the points distribution as the first wedge and for all faces as the face one as shown in figure (2.7) .then mesh the three faces as shown in figure (2.8).

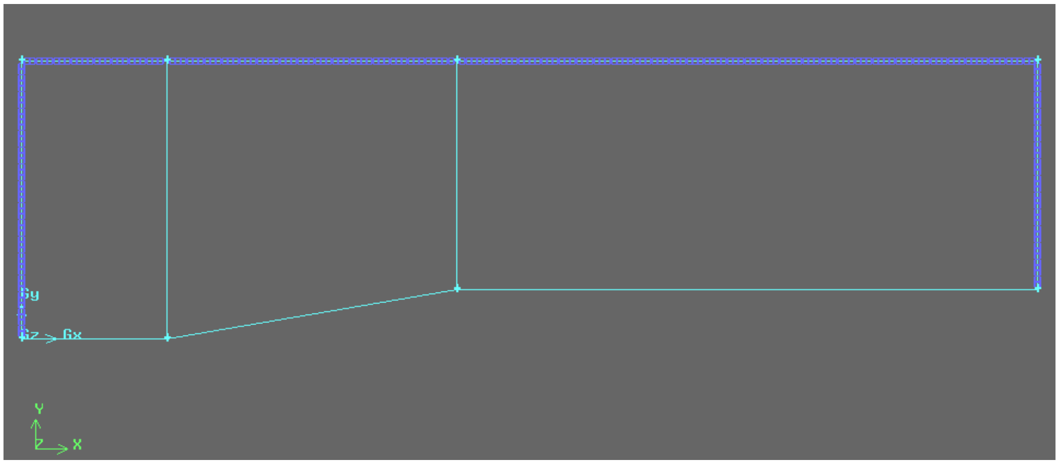


Fig 2.7: Start to mesh the geometry in Gambit.

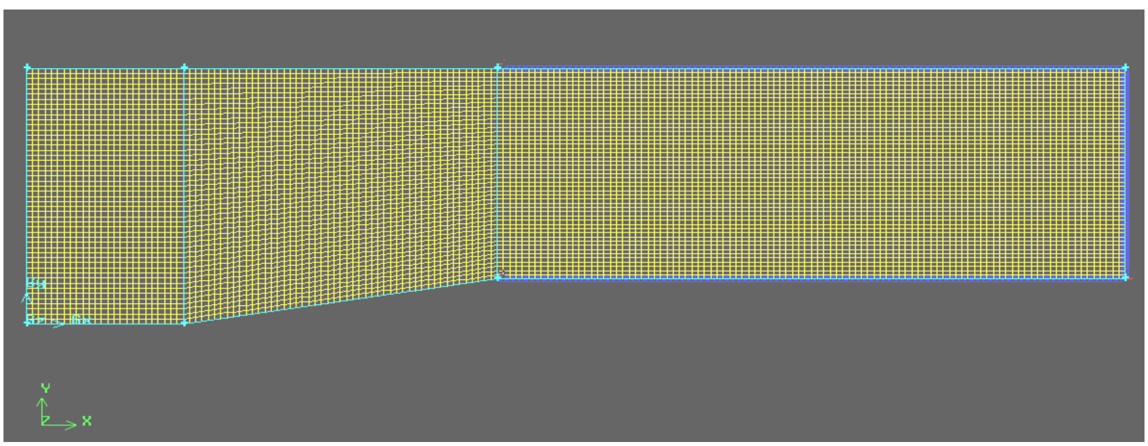


Fig 2.8: Creating the mesh in Gambit.

2.4.1.3 Specify Boundary Types in GAMBIT:

To specify the boundary type the boundary need to label (ABCDEF) as far-field, (HG) as wedge and (FG, HA) as symmetry. Recall that these will be the names that show up under boundary zones when the mesh is read into FLUENT. At the first create groups of edges and then create boundary entities from these groups. First, so the group it will be as AB, BC, CD and DE together and define it as far afield group, and then group the other edges as shown in the table (2.2), now the defined of all wedges as groups the geometry can define the boundary type in Gambit as shown in table (2.3), then save the geometry in Gambit and save it as mesh file.

Table 2.2: Defining the wedges as group.

Group name	**Edges in group**
Far-field	AB, BC, CD, DE, EF
Wedge	GF
Symmetry	FG, HA

Table 2.3: Defining the groups to the boundaries.

Group name	**Boundary entities**
Far-field	Pressure far-field
Wedge	Wall
Symmetry	Symmetry

2.4.1.4 Set Up Problem in FLUENT:

In this case firstly open the Fluent software then read the case which have done in the previous steps then make check on this case to make sure there are no problems in the case then define the properties of the problem, for define the solver turn on a density based and an explicit because the problem expect an oblique shock then for define the viscous turn on an inviscid flow This means the solver will neglect the viscous terms in the governing equations.

In compressible flow, the energy equation is coupled to the continuity and momentum equations. So the problem need to solve the energy equation for our problem, so turn on the energy equation from defining the energy, and then make sure the material is air from defining material and we will set the density to ideal gas, CP is constant and equal to 1006.43 j/kg-k and also the Molecular Weight is constant and equal to 28.966 kg/kgmol. For selecting the ideal gas option means that FLUENT will use the ideal gas equation of state to relate density to the static pressure and temperature. And then will go to define the boundary conditions as far afield to pressure far-field and set the (Gauge Pressure to 101325, the Mach number to 2, X-Component of Flow Direction to 1 and the temperature to 300K. We are assuming ambient temperature.) , Set wedge to wall boundary type and symmetry to symmetry type.

2.4.1.5 Solve:

To solve the problem use a second-order discretization scheme. Then set the initial guess values for the iterative solution. Use the far-field values (M=3, p=1 atm, T=300 K) as the initial guess for the entire flow field. Then set all the equations to 1e-6 and set the iteration number to 5000 then solve the problem, after finish the iterations save the data and analysis the results which we get from the Fluent software.

2.4.1.6 The Analytic Solution of Compression and Expansion Wave:

The flow problem as described in the previous sub chapter actually can be solved analytically. Refer to the Figure 2.6, For a given supersonic flow condition at entry station, let say, for example, the Mach number M, pressure P and temperature T are given as 2, 101.325Kpa, and 300K^{o} respectively. The bottom surface deflected at point A and B. At point A such positive deflection with respect to the flow direction create an oblique shock wave, while at the point B, the surface deflection sees with respect to the flow direction is a negative angle as result an expansion wave occurred at that point. For simplify the explanation, let the flow domain in before point A is Region (1), in between region A and B as Region (2) and after B as flow region (3). For a given flow condition in region (1), one can determine the flow condition region (2) by solving as oblique shock wave and for the flow in region (3) as expansion wave. This problem can be described schematically as shown in the Figure 2.9 bellows

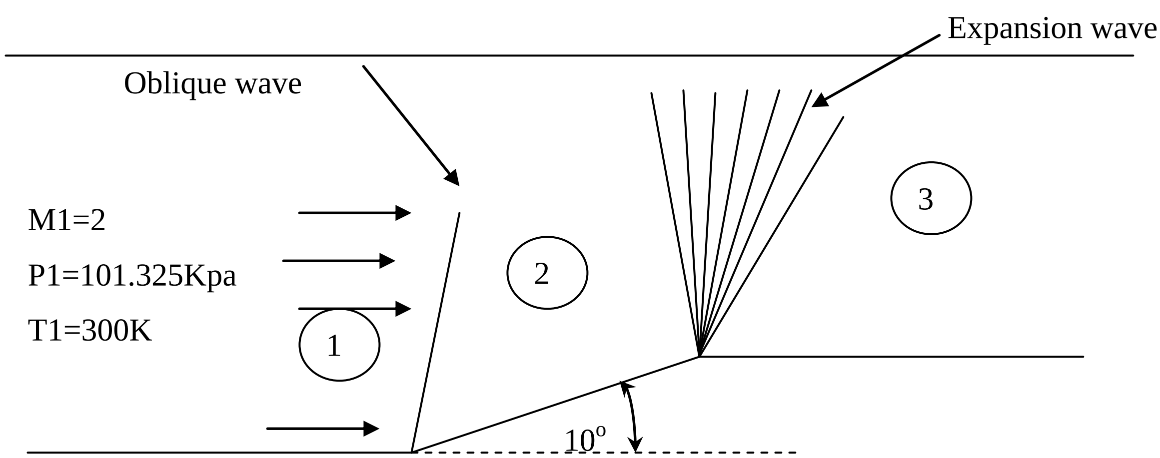


Fig 2.9: Simple geometry for inlet of airplane engine.

Flow solution for region (2) as an oblique shock wave problem for a given Mach number $M_{1,}$ deflection angle δ_1, and then the oblique shock angle β can be obtained through solving the M-δ-β equation defined as:

$$M = \frac{V}{a} \qquad (2.17)$$

Where (V) is velocity on one − dimensional flow, and (a) is Local speed of sound δ is deflection angle

Where:

$$tan\delta = \frac{2cot\beta(M_1^2 sin^2\beta - 1)}{2 + M_1^2(\gamma + cos2\beta)} \qquad (2.18)$$

β is oblique shock angle

Above equation is non linear algebraic equation, hence an iteration process is required. However one can use another method called as Graphical method. Equations the M-δ-β equations have already available in graphical form, so for given Mach and deflection angle δ, by use those graph which the solution for oblique shock $\beta=39^o$: Knowing this oblique shock, then other flow properties at the flow region (2) can be obtained sequentially as:

$$M_{N1} = M_1 sin\beta$$
$$= 2 \times sin39 = 1.258$$

To find the properties at M_{N1}=1.258, there is table called (Normal Shock Wave) can find those properties. So now can get:

$$M_{N2} = 0.808, \qquad \frac{P_2}{P_1} = 1.6793, \quad \frac{T_2}{T_1} = 1.16433$$

From Eq:

$$M_2 = \left(\frac{M_{N2}}{\sin(\beta - \delta)}\right) = 1.666$$

And from:

$$\frac{P_2}{P_1} = 1.6793 \ \rightarrow P_2 = 170.124Kpa$$
$$\frac{T_2}{T_1} = 1.16433 \ \rightarrow T_2 = 300.16433K^o$$

The previous steps showed the properties at the regain 1 and the regain 2, and before regain 3 there is expansion wave occurred, so to find the properties in regain 3 there is table called (Prandtl-Meyer Flow Table) from this table can find the θ_2 at M=1.666 which fined as:

$$\theta_2 = 16.8088°$$

From this value can find V_3 from the Equation:

$$\theta_3 = \theta_2 + \delta = 16.8088 + 10 = 26.8088°$$

Now from the same table can find M_3 which equal (2.016) and to find the properties at regain 3 can find it from table called (Isentropic flow tables for γ=1.4) at M_3=2.016 can find:

$$\frac{T_o}{T} = 1.81286 \rightarrow T_3 = 165.5K, \qquad \frac{P_o}{P} = 8.2697 \rightarrow P_3 = 12.253Kpa$$

CHAPTER 3

3.0 TOTAL VARIATION DIMINSHING SCHEME

3.1. Basic Idea TVD – Runge-Kutta Scheme

Numerical TVD schemes have been developed by various investigators over the last several years and more are being introduced at present (K.A.Hoffmann). The various TVD schemes can be broadly categorized and subcategorized. First, TVD schemes may be classified as first-order TVD schemes, second-order TVD schemes which are usually referred to as high resolution schemes, and predictor-corrector type TVD schemes. Furthermore, in each category, the formulation may be explicit or implicit. In terms of finite difference approximation, the resulting formulation may be classified as symmetric or upwind. Furthermore, for each formulation, different functions may be available for flux limiters. Thus, within each category, numerous formulations can be written. To familiarize with the TVD scheme, it can be done by a prototype of Euler Equation in the form of scalar hyperbolic partial differential equation which can be written as:

$$\frac{\partial u}{\partial t} + \frac{\partial E}{\partial x} = 0 \qquad (3.1)$$

In above equation, (u) is dependent variable, (E) is flux function and (x) and (t) are independent variable in spatial and temporal. There are various approaches can be used to convert from a continue differential equation into a discrete equation. The approximate solution u will not increase as the computational time in progress if fulfill the condition:

$$TV(u^{n+1}) \leq TV(u^n) \qquad (3.2)$$

Where:

$$TV(u^n) \equiv \sum_{-\infty}^{+\infty} |u_{i+1}^n - u_i^n| \qquad (3.3)$$

In implementing the Runge-Kutta fourth order time integration scheme combined with the spatial discretization which satisfies the TVD condition (Eq. 3.2) can be done through the normal use of Runge-Kutta scheme and then apply TVD formulation the last step. For a given a scalar partial differential Eq (3-1) the Fourth Order Runge-Kutta scheme can be implemented in the form:

$$u_i^{(1)} = u_i^n \qquad (3.4)$$

$$u_i^{(2)} = u_i^n - \frac{\Delta t}{4}\left(\frac{\partial E}{\partial x}\right)_i^{(1)} \qquad (3.5)$$

$$u_i^{(3)} = u_i^n - \frac{\Delta t}{3}\left(\frac{\partial E}{\partial x}\right)_i^{(2)} \qquad (3.6)$$

$$u_i^{(4)} = u_i^n - \frac{\Delta t}{2}\left(\frac{\partial E}{\partial x}\right)_i^{(3)} \qquad (3.7)$$

$$(u_i^{n+1})_{RK} = u_i^n - \Delta t\left(\frac{\partial E}{\partial x}\right)_i^{(4)} \qquad (3.8)$$

Where:

$$\left(\frac{\partial E}{\partial x}\right)_i^{(m)} = \frac{E_{i+1}^{(m)} - E_{i-1}^{(m)}}{2\Delta x}\ , where \quad m = 1,2,3,4 \qquad (3.9)$$

In combining with the TVD scheme, so the method called as the Fourth Order Runge-Kutta – TVD scheme, the dependent variable which at the last stage time integration completed by (Eq.3.8),is added with one additional stage for calculating u at the time level $t=t^{n+1}$ as:

$$u_i^{n+1} = (n_i^{n+1})_{RK} - \frac{\Delta t}{2\Delta x}\left(\Phi_{i+\frac{1}{2}}^n - \Phi_{i-\frac{1}{2}}^n\right) \qquad (3.10)$$

Where:

$$\Phi_{i+\frac{1}{2}}^n = \sigma\left(\alpha_{i+\frac{1}{2}}\right)(G_{i+1} + G_i) - \psi\left(\alpha_{i+\frac{1}{2}} + \beta_{i+\frac{1}{2}}\right)\Delta u_{i+\frac{1}{2}}^n \qquad (3.11)$$

$$\Phi_{i-\frac{1}{2}}^n = \sigma\left(\alpha_{i-\frac{1}{2}}\right)(G_i + G_{i-1}) - \psi\left(\alpha_{i-\frac{1}{2}} + \beta_{i-\frac{1}{2}}\right)\Delta u_{i-\frac{1}{2}}^n \qquad (3.12)$$

With:

$$\psi(y) = \begin{cases} |y| & \text{for } |y| \geq \varepsilon \\ \dfrac{(y)^2 + \varepsilon^2}{2\varepsilon} & \text{for } \quad |y| < \varepsilon \end{cases} \qquad (3.13)$$

Where:

$0 \leq \varepsilon \leq 0.125,$ and

$$\sigma\left(\alpha_{i+\frac{1}{2}}\right) = \frac{1}{2}\psi\left(\alpha_{i+\frac{1}{2}}\right) + \frac{\Delta t}{\Delta x}\left(\alpha_{i+\frac{1}{2}}\right)^2$$

And

$$\beta_{i+\frac{1}{2}} = \sigma\left(\alpha_{i+\frac{1}{2}}\right)\begin{cases}\dfrac{G_{i+1} - G_i}{\Delta u_{i+\frac{1}{2}}}, \Delta u_{i+\frac{1}{2}} \neq 0 \\ 0, \Delta u_{i+\frac{1}{2}} = 0\end{cases}$$

And

$$G_i = minmod\left(\Delta u_{i-\frac{1}{2}}, \Delta u_{i+\frac{1}{2}}\right) \qquad (3.14)$$

$$G_i = \frac{\Delta u_{i+\frac{1}{2}}\Delta u_{i-\frac{1}{2}} + \left|\Delta u_{i+\frac{1}{2}} + \Delta u_{i-\frac{1}{2}}\right|}{\Delta u_{i+\frac{1}{2}} + \Delta u_{i-\frac{1}{2}}}$$

If $\Delta u_{i+1/2} + \Delta u_{i-1/2} = 0, then\ G_i = 0$

$$G_i = \frac{\Delta u_{i-\frac{1}{2}}\left[\left(\Delta u_{i+\frac{1}{2}}\right)^2 + \omega\right] + \Delta u_{i+\frac{1}{2}}\left[\left(\Delta u_{i-\frac{1}{2}}\right)^2 + \omega\right]}{\left(\Delta u_{i+\frac{1}{2}}\right)^2 + \left(\Delta u_{i-\frac{1}{2}}\right)^2 + 2\omega}, 10^{-7} \leq \omega \leq 10^{-5} \qquad (3.15)$$

$$G_i = \text{minmod}\left[2\Delta u_{i-\frac{1}{2}}, 2\Delta u_{i+\frac{1}{2}}, \frac{1}{2}\left(\Delta u_{i+\frac{1}{2}} + \Delta u_{i-\frac{1}{2}}\right)\right] \qquad (3.16)$$

$$G_i = S * \max\left[0, \min\left(2\left|\Delta u_{i+\frac{1}{2}}\right|, S * \Delta u_{i-\frac{1}{2}}\right), \min\left(\left|\Delta u_{i+\frac{1}{2}}\right|, 2S * \Delta\Delta u_{i-\frac{1}{2}}\right)\right] \qquad (3.17)$$

Recall that

$$minmod\ (a, b, c, \ldots, n) = S * \max[0, min(|a|, S * b, S * c, \ldots, S * n)]$$

REFERENCES

Batchelor. G. K (1967). *An Introduction to Fluid Dynamics. Cambridge UniversityPress.*

C.HIRSCH (1990). *Numerical Computation of Internal and external flows.*

Dongfang Liang, Roger A. Falconer and Binliang Lin (2006). *Comparison between TVD-MacCormack and ADI-type solvers of the shallow water equations.*

Edisson Sávio de Góes Maciel (2007). *Comparison Among Structured First Order Algorithms in the Solution of the Euler Equations in Two-Dimensions.*

H.C.Yee and P.Kutler (1983). *Application of Second-Order-Accurate Total Variation Diminishing (TVD) Scheme to the Euler Equations in General Geometries.*

HILEN MARIE FELICI (1992). *A coupled Eulerian /Lagrangian method for the solution of three dimensional vertical flows.*

K_A.Hoffmann, Staven T Chiang (2004). *Computational fluid dynamics*.

Jack D. Mattingly (2006). *Elements of Propulsion: Gas Turbines and Rockets*.

J.Shi. and E.F.Toro (1993). *Fully Discrete High-Order TVD schemes for A scalar Hyperbolic Conservation Low.*

Mark Anthony Driver. B.A.E.. M.S. (1991). *High-Resolution TVD schemes for the analysis of 1.Inviscid supersonic and transonic flows. 2. Viscous flows with shock-induced separation and Heat transfer.*

Patrick H.Oosthuizen (1997). *Compressible fluid flow*.

Raph Levien (2008). *The Euler spiral: a mathematical history*.

T. H. Pulliam (2000). *Solution Methods in Computational Fluid Dynamics.*