International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-3 Issue-4, April 2014

Numerical Simulation of the Interaction Shockwave / Turbulent Boundary Layer: Interference RR/MR

Benderradji R, Gouidmi H, Beghidja A

Abstract - This study focuses on both the development of the turbulent boundary layer in supersonic flow over a flat plate, the distance required for it to invade the entire section of the plate, and the effects of the size of the interaction area on the development of the boundary layer. Increasing the strength of the interaction is an increase in size of the areas of interaction leading to the formation of a recirculation bubble which is an area of the head losses. For this reason, it can increase and decrease the Mach number for a reflection or Mach explains the impact of the incident shock wave with strong boundary layer. The increase and decrease the Mach number has caused the appearance of a hysteresis loop which is represented by the contours of iso fields' density. These studies are in agreement with respect to the trial that was presented by J. Delery et al (2009). We were given another contribution and investigations of the phenomenon of wave interaction of shock / turbulent boundary layer. The model used in this study is the kw-SST model, it is considered as the most suitable for this kind of problem, with special treatment of the near-wall region. Numerical simulations were performed using FLUENT software.

Keyword: Interaction of shockwave/ boundary layer-Interference shockwaves-Polar shock- RR Reflection- Reflection MR.

I. INTRODUCTION

Generally, the boundary layer perturbation is due to several variants, particularly of the Reynolds number, the wall temperature. The interaction between an oblique shock wave with a boundary layer, a further aspect is defined disturbance. This phenomenon is of great importance in aerodynamics. On the particular meeting at the air intakes, the wings of supersonic aircraft and launchers. This interaction process is used to define the characteristics of the structure of the boundary layer. The interaction area is disrupted (unstable), caused by the shock wave intensity, which determined the structure of the inner region of the boundary layer. It is changed. The dynamic interaction between the shocks - wave boundary turbulent layer on an adiabatic plate, allows to introduce a separation zone of the flow with the wall, which causes another oblique shock wave. This area resulted the appearance of a bulb with the recirculation phenomenon interference shock waves (RR - MR).

Studies that were carried out, giving great importance in aerodynamic characteristics of the boundary layer. Experiences such as J. Deleuze (1995) [1] , H. Lawrence (1996) [2] , J. Delery and Dussauge JP (2009) [3] , Dolling (2001) [4] , and simulations of A. Hadjadj et al (2010) [5] , S. Deck et al (2011) [6] , K. Sinha et al (2004) [7] , J. R. Edwards (2008) [8] , J. E. Choi et al (1998) [9] S. Pirozzoli et al (2006) [10] , E. Touber et al (2009) [11] S. Teramoto (2005) [12] , B. Morgan et al (2010) [13] Mr. Lagha et al (2011) [14] MF Shahab et al (2009) [15] , S. Dubos [16] , C. Hadad (2005) [17] . They showed that the interaction area and the outer portion of the boundary layer are disrupted (unstable).

II. GEOMETRY AND FLOW PARAMETERS

A. Creating the geometry and mesh

The creation of the geometry and the mesh are due to software «Gambit «2.3.16. Several methods allow the creation of this geometry, or one based on predefined geometries, or simply enter the coordinates of the points (x , y) in 2D , create boundaries and finally create the surface . However, for our case, two main choices mesh arose. In this case, a mesh of quadrilaterals or based cells or triangular cells based. The use of a triangular mesh induce a surplus in the number of cells compared with cells quadrilaterals, hence the need for more resources and computing time. However, this is relatively simple geometry in which the flow follows substantially the form of the geometry. So using a quadrilateral mesh cells, we have an alignment of the flow with our mesh, then it will never be the case with triangular cells. Particular attention should be paid to the subsequent verification of mesh refinement near the walls to ensure that all phenomena are captured. FLUENT offers three methods for treating turbulence near the walls: standard (Standard Wall- Functions) functions walls, the functions of unbalanced walls (Non- equilibrium wall-function) and improved treatment laws walls (Enhanced wall treatment). In each of these cases, a subsequent verification must be done in order to verify the mesh. This is done by looking at the + y takes values . We will use the first method is the simplest. For this method it is necessary that the centre of each cell lies within the logarithmic region ie 30 <y + < 300. It is best to approach the lower limit: y + = 30.

B. Control of mesh sizes

To verify that it is included in 30 and 300, we get the following result which confirms that we are in this range there. Note that we had to make an adaptation of the mesh to reduce the values of y + as our mesh were not fine enough near walls. One can see that there is between + 30 and 44. The grid resolution is very good for the problem studied.

Manuscript published on 30 April 2014.
***** Correspondence Author (s)
Dr. R. Benderradji Laboratory of renewable energies and sustainable development (LERDD), University of Constantine1, Algeria.
Dr. H. Gouidmi, Laboratory of renewable energies and sustainable development (LERDD), University of Constantine1, Algeria.
A Beghidja: He is an engineer in 1983 University of Constantine. Ph.D. 1990 ENSMA Poitiers, France. He Is a professor in 2006 at the University of Constantine**.** Algeria.

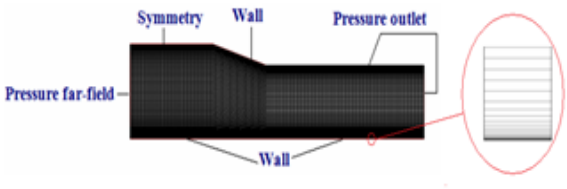

FIG.1: Geometry and mesh of the problem

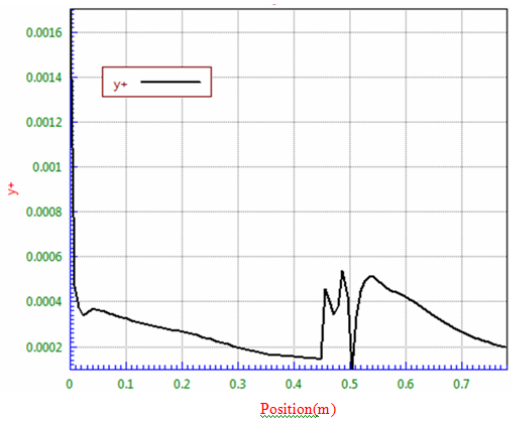

FIG 2 : Evolution y^+ along the wall to kw-SST model.

III. MODELLING AND NUMERICAL SOLUTION

For the numerical solution of supersonic flow inside the nozzle, two commercial codes were used, FLUENT 2D version 6.3.26 (solver) and GAMBIT 2.3.16. (Mesh generator), both supplied by Fluent. Inc. In this study the turbulence model is used URANS, kw- SST gives the quantitative properties of the turbulence, in the entire flow. The computer code used, solved the Navier-Stokes Reynolds averaged to explicit second order and instant formulations. These equations , momentum , energy and turbulence model are discredited by the finite volume method , and using the cantered second order Upwind scheme based on flow - Roe FDS a CFL = 0.5 . The system of equations for each time step was solved by an iterative method developed classical Gauss-Seidel iterative. For the sake of accelerating the convergence, a step of pseudo-relaxation time was used in each time step with a suitable expansion factor. The fluid used is air, considered as ideal gas. Admission requirements (initial condition of the flow) are shown in Table.1. The density is calculated using the ideal gas law (isentropic flow). Sutherland's law [20] was chosen to calculate the molecular viscosity μ describes the variations of the viscosity with respect to temperature, because in a supersonic flow, there are large temperature gradients.

$$\mu(T) = \mu_0 \left(\frac{T}{T_0}\right)^{1.5} \frac{T_0 + 110.56}{T + 110.56}$$

Table1. Physical parameters of the flow upstream.

Definition	Value and unit
Number of current local imposes cfl	0.5
Mach number at upstream infinity (ideal gas)	Vari
Value of the energy upstream infinity	1683 (Pa)
Value of temperature upstream infinity	76.5 (K)

IV. RESULTS AND DISCUSSION

Calculation results were obtained on a cluster machine with a CPU time of about 20 hours to complete convergence. A dihedral angle (impulse generator), 17 ° has been fixed, and the angle of the incident shock varied by decay and growth of the upstream Mach number in the initial conditions .The sequence of the Figure.3. This left curves iso densities, and right form of the interaction zone (bubble recirculation). The separation of the viscous layer in the turbulent boundary layer under the effect of the impact of the shock wave on an adiabatic plate , allows to introduce a separation zone of the wall with the flow which causes a shock compression (incident shock) . There will therefore be the phenomenon of interference of shock waves RR -MR (shock wave / shock compression). First, a regular reflection (RR). This configuration has been obtained starting from a uniform field equal to Mach = 3,2. Then, the Mach number is decreased and every time a stable stationary solution has been reached, based on the initial field converged to the previous number of Mach. The calculations were performed until a Mach reflection (MR) , the transition to the regular reflection of the reflection is performed for a Mach number equal to Mach 2. The figure clearly shows the sudden appearance of a quasi- normal shock and recirculation bubble becomes unstable.

Figure 4. A. Presented changes in the parietal pressure undergo a sudden growth, crossings interaction, due to compression waves, present below the reflected shock. The pressure level of the upstream flow. This decrease is caused by the intensity of the beams of relaxing the trailing edge of the shock generator incident. Figure shows that there is a similar approximation of the wall pressure fluid than perfect, but offset toward the upstream of the point of impact of the incident shock. It is caused by the intensity of incident shock. The results presented show that the simulation URANS per kw-SST predicts correctly the pressure rises, as well as levels of downstream interaction. This remark was observed by J. Deleuze [1] A. Hadjadj et al. [5]. The sequence of Figure.4 B. represents the distribution coefficient of friction along the nozzle. This figure shows that the presence of the solid wall causes slow particles. This slowdown is due to the fact that the viscosity tends to produce by viscous friction between the wall and the flow. By the action of this friction, the flow velocity near the wall decreases. Therefore, the thickness of the neck heli mid increases aufur you and me su re that the flow develops to invade the whole section of the nozzle. This figure shows that at the entrance of the nozzle, the coefficient of friction in the area close to the wall decreases to the interaction zone, at the impact of the shock wave with the boundary layer and increases in the region after the recirculation zone but remains positive. Just from the point of detachment of the birth of interaction zone the friction coefficient becomes negative along the entire area up to the point of reattachment. This phenomenon accompanying the development of the boundary layer at the wall where the turbulence levels are high. After this distance, the flow continues to develop until reaching the final establishment to constant values. The shape of this curve shows that there is a disturbance in the region of the interaction area (bubble of recirculation)

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

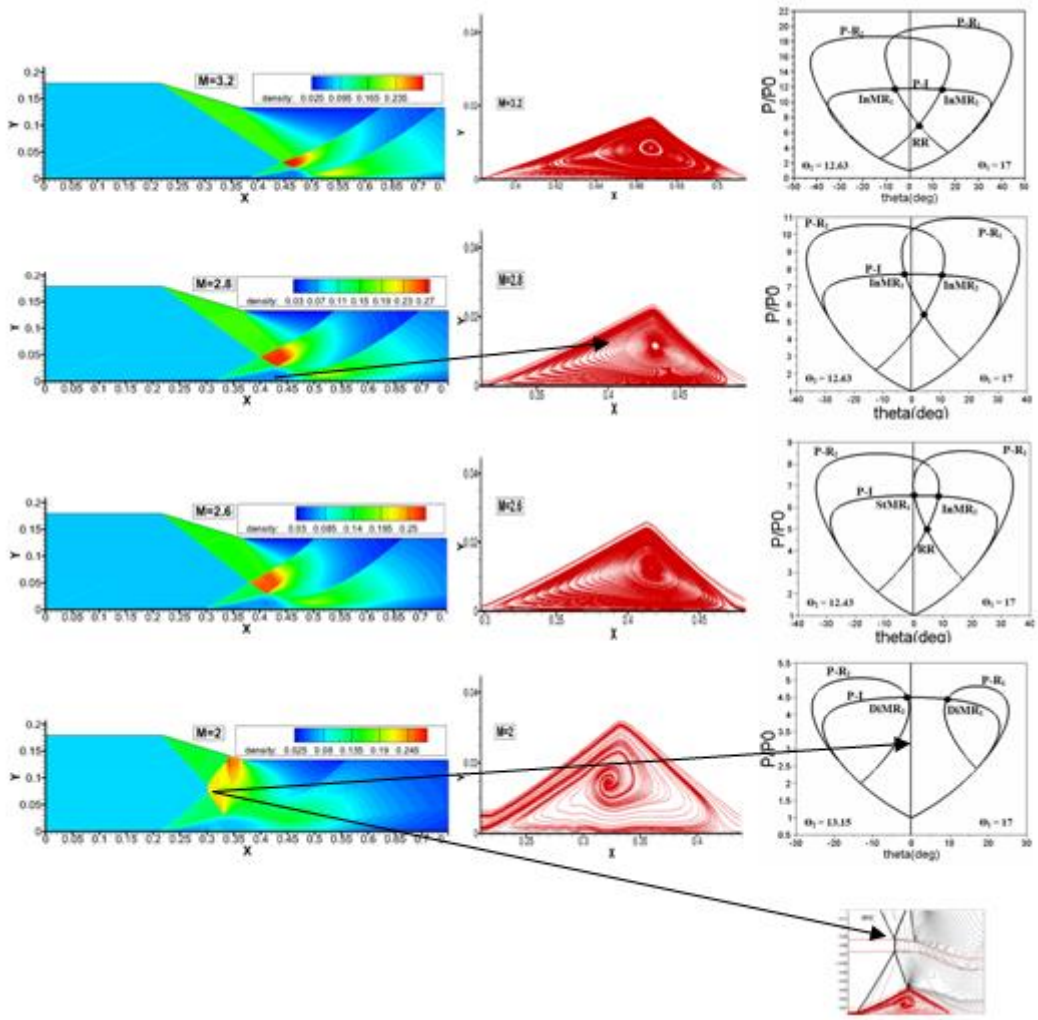

FIG 3: Iso contours fields density with the shape of the interaction area (bubble recirculation) for different Mach number.

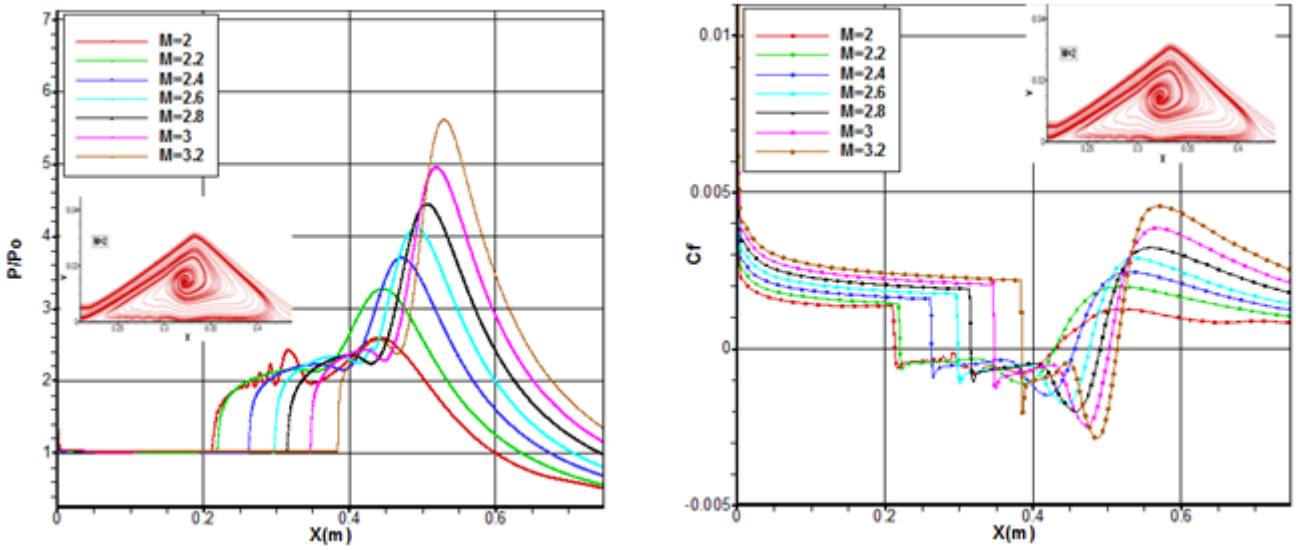

FIG 4: A - Longitudinal changes in wall pressure. B - Longitudinal changes in coefficients of friction.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

A. The effect of Mach number on the detachment /reattachment, and the height of the interaction area

The figures 5 and figure 6. Respectively represent the evolution of separation point's detachment / reattachment, and the height of the interaction area with the number of upstream Mach. Both figures show that: when the Mach number decreases the impact of a shockwave in a boundary layer is stronger and the thickening zone of the subsonic boundary layer, the thickness (height) is increased, because the shock is more intense. It is low in the opposite case. The decrease of the Mach number also influences the formation of a bubble recirculation i.e., the distance between the detachments / reattachment is greater than if the shock wave is strong.

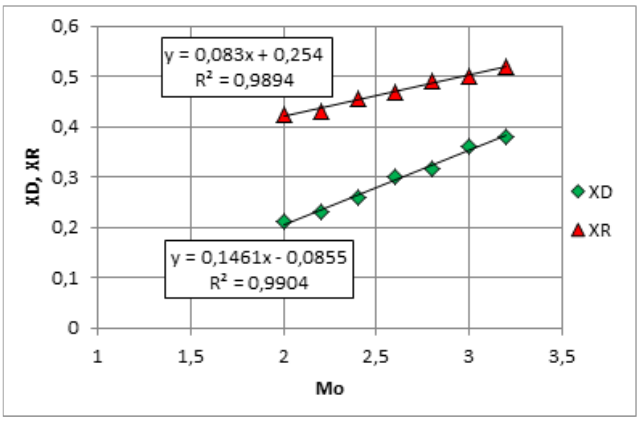

FIG 5: Changes in point's detachment and reattachment with the number of upstream Mach.

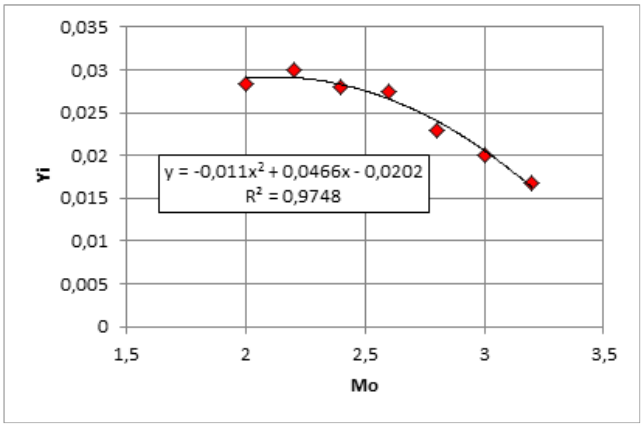

FIG 6: Height of the interaction area with the number of upstream Mach.

V. CONCLUSION

The transition regular reflection ⇔ Mach reflection was simulated numerically by solving Navier Stokes equations, using the code FLUENT calculation for a two-dimensional compressible turbulent flow. Through this study we have shown the interest and importance of the phenomena of wave interactions shock / boundary layer in supersonic nozzles. Thus, numerical simulations in a 2D nozzle could highlight certain phenomena:

1- hysteresis in the transition RR-MR, due to the memory effect of the flow, and by the interference between a shock wave generated by the generator of impact / shock caused by the compression of the sub décolement viscous layer of the boundarylayer.

2- In line with experience (J. Deleuze. [3]), and a calculation based on the finite volume method CFD. Bubble recirculation in MR solution is more volatile than in the RR solution.

REFERENCES

[1] J. Deleuze, "Structure d'une couche limite turbulente soumise à une onde de choc incident. Thèse doctorat. Université AIX-Marseille II, No. 2079588 (1995).

[2] H. Laurent, Turbulence d'une interaction onde de choc/couche limite sur une paroi plane adiabatique ou chauffée, thèse de Doctorat, Université AIX-Marseille II, (1996).

[3] J. Délery and J. P. Dussauge, Some physical aspects of shock wave/boundary layer interactions, Shock wave, No. 19 (2009) 453-468.

[4] D. S. Dolling, Fifty years of shock-wave/boundary layer interaction research: what next?, AIAA J. No. 39 (8) (2001).

[5] A. Hadjadj J, Larsson, B. E. Morgan, J. W. Nichols and S. K. Lele, Large-eddy simulation of shock/boundary layer interaction. Center for turbulence research-proceeding of the Summer ¨Program (2010).

[6] S. Deck, P. E. Weiss, M. Pamiès and E. Garnier, Zonal detached Eddy Simulation of spatially developing flat plate turbulent boundary layer, Computer§ Fluids No. 48 (2011) 1-15.

[7] K. Sinha, K. Mahesh and G. Candler, Modeling the effect of shock unsteadiness in shock-wave/turbulent boundary layer interactions, AIAA (2004).

[8] J. R. Edwards, Numerical simulations of shock/boundary layer interactions using time-dependent modeling techniques: A survey of recent results, Progress in Aerospace Sciences No. 44 (2008) 447-765.

[9] J. Y. Choi, I. S. Jeung and Y. Yoon, Scaling effect of the combustion induced by shock-wave boundary layer interaction in premixed Gas, Twenty-seventh symposium (international) on combustion (1998) 2181-2188.

[10] S. Pirozzoli and F. Grasso, Direct numerical simulation of impinging shock wave/turbulent boundary layer interaction at M=2.25, Physics of fluids No. 18 065113 (2006).

[11] E. Touber and N. D. Sandham, Large-eddy simulation of low-frequency unsteadiness in a turbulent shock-induced separation bubble, Theor. Comput. Fluid Dyn. No. 23 (2009) 79-107.

[12] S. Teramoto, Large-Eddy simulation of shock wave/Boundary layer interaction, Trans. Japan Coc. Aero. Space sci. Vol. 47, No. 158 (2005) 268-275.

[13] B. Morgan, S. Kawai and S. K. Lele, Large-Eddy simulation of an oblique shock wave impinging on a turbulent boundary layer, 10th fluid dynamics conference and exhibit, AIAA No. 4467 28 june-1 july Chicago, IIIinois (2010).

[14] M. Lagha, J. Kim, J. D. Eldredge and X. Zhong, A numerical study of compressible turbulent boundary layers, Physics of fluids No. 23 015106 (2011).

[15] M. F. Shahab, G. Lehnasch, T. B. Gatski and P, Comte, DNS of a spatially developing, supersonic turbulent boundary layer flow over a cooled wall, Turbulence, Heat and Masse Transfer 6, (2009).

[16] S. Dubos, Simulation des grandes échelles d'écoulement turbulent supersoniques, These Doctorat, INSA de Rouen (2005).

[17] C. Hadad, Instationnarités, mouvements d'onde de choc et tourbillons à grandes échelles dans une interaction onde de choc/couche limite avec décollement, Thèse Doctorat, Université de Provence Aix-Marseille I (2005).

[18] V. S. Murthy and W. C. Rose, Wall shear stress measurements in shock wave boundary layer interaction, AIAA J. No. 7, 16 (1978) 667-672.

[19] E. R. Van Driest, Turbulent boundary layer in compressible fluids, J. Aero. Sc. No. 3, 18 (1951) 145-160.

[20] Sebastian Deck, Philippe Duvaeu, Paulo d'Espincy et Philippe Guillen: Development and application of Spalart-Allmaras one equation turbulence model to three dimensional supersonic complex configurations. Aerospace Science and Technology. (2002) 171-183.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

R. BENDERRADJI (Dr), Laboratory of renewable energies and sustainable development (LERDD), University of Constantine1, Algeria.

H. GOUIDMI (Dr), Laboratory of renewable energies and sustainable development (LERDD), University of Constantine1, Algeria..Email address: gouidmi@yahoo.fr

A BEGHIDJA: He is an engineer in 1983 University of Constantine. Ph.D. 1990 ENSMA Poitiers, France. He Is a professor in 2006 at the University of Constantine.His research interests include numerical optimization, heat transfer, thermodynamics and fluid mechanics and their application on renewable energy, energy management, wind energy, solar energy and, Laboratory of renewable energies and sustainable development (LERDD), University of Constantine1, Algeria.
.Email address: abeghidja1@hotmail.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)