Diyala Journal
of Engineering
Sciences

ISSN 1999-8716
Printed in Iraq

Vol. 06, No. 01, pp. 105-120, March 2013

SIMULATION OF BACK PRESSURE EFFECT ON BEHAVIOUR OF CONVERGENT DIVERGENT NOZZLE

Nazar Muneam Mahmood
Unityversity of Dyala\ Engineering college\ Electronic department
Email:Nasamatl@yahoo.com

(Received:30/11/2011 ; Accepted:26/6/2012)

ABSTRACT:- In this research a simulation of steady flow of a gas through a convergent divergent nozzle which has a varying cross sectional area will be considered. The nature of the flow can be explained by considering how the flow and its characteristics in the nozzle changes as nthe back pressure P_b is decreased.

The characteristics of gas flow i.e.(Mach number, static pressure, density, velocity magnitude and static temperature) distributions for the convergent divergent nozzle are implemented by using the ANSYS Fluent 12.1 software to solve the quasi-one dimensional nozzle flow.

The reductions in the back pressure cannot affect conditions upstream of the throat. The nozzle is, therefore, choked. The shock wave increases the pressure, density and temperature and reduces the velocity and Mach number to a subsonic value, and as back pressure is further reduced to a certain value, the extent of the supersonic flow region increases, the shock wave moving further down the divergent portion of the nozzle towards the exit plane.

Keywords :convergent divergent h , back pressure , shock wave , variable area flow , quasi-one dimensional nozzle flow.

INTRODUCTION

The one-dimensional inviscid isentropic flow in a convergent-divergent (CD) nozzle is a classical textbook problem, which has different flow regimes depending upon the nozzle pressure ratio (NPR). The inviscid theory predicts a simple shock structure consisting of a normal shock followed by a smooth recovery to exit pressure in the divergence part of a choked nozzle.(1)

Steady flow of a gas through a nozzle which has a varying cross sectional area i.e., compressible gas flows through a nozzle whose cross-sectional area is varying, occur in many engineering devices, e.g., in the nozzle of a rocket engine and in the blade passages in turbo machines. It will be assumed that the flow can be adequately modeled by assuming it to be one-dimensional at all sections of the nozzle, i.e., quasi-one-dimensional flow will be assumed. It will also be assumed in studying the effects of changes in area on the flow that the flow is isentropic everywhere except through any shock waves that may occur in the flow(2).

The term compressible flow is routinely used to define variable density flow which is in contrast to incompressible flow, where the density is assumed to be constant throughout. In many cases, these density variations are principally caused by the pressure changes from one point to another. Physically,the compressibility can be defined as the fractional change in volume of the gas element per unit change in pressure. It is a property of the gas.(3)

According to the assumptions introduced, are valid only for isentropic flow, since the ariation of the density is assumed to be isentropic. Mach number and pressure distribution can be determined for an area distribution prescribed in the streamwise direction. The flow in the nozzle solely depends on the pressure in the exit cross section.(4) This assumption is usually quite adequate since the effects of friction and heat transfer are usually restricted to a thin boundary layer in the types of flows here being considered and their effects can often be ignored. (2)

VARIABLE GEOMETRY CONVERGENT-DIVERGENT NOZZLE

The first nozzle considered here is an axisymmetric configuration employing a variable convergent divergent (C-D) internal flowpath. Modern military aircraft utilizing an afterburning engine most frequently use such a C-D nozzle with the variable geometry convergent and divergent sections provided by a flap and seal arrangement. The aerodynamic layout of the C-D nozzle began with consideration of the divergent section. In order to minimize nozzle divergence losses, it is desirable to set the divergent flaps as long as possible. However, the overall length of the nozzle could not exceed 26 in. (5)

CRITICAL PRESSURE (P*)(6)

Critical flow through a convergent-divergent nozzle is a phenomenon that occurs when the mass flow rate of a gas cannot be increased when the pressure drop across the nozzle is further increased. The ratio between the downstream and upstream pressures at which this occurs is called the critical pressure ratio.

FLOW BEHAVIOUR IN A CONVERGENT–DIVERGENT NOZZLE

The application of fluid mechanic theory to an ideal convergent-divergent nozzle leads to many main types of flow behaviors (7)

The operating characteristics of nozzles it will be assumed that the nozzle is connected to an upstream chamber in which the conditions, i.e., the upstream stagnation conditions, are kept constant while the conditions in the downstream chamber into which the nozzle discharges are varied. The pressure in the downstream chamber is termed the back pressure. The nature of the flow can be explained by considering how the flow in the nozzle changes as the back pressure P_b is decreased (2). When the gas exits the nozzle at supersonic speeds, it undergoes several flow phenomena depending on the nozzle pressure ratio **(8)**. The flow regimes are then as shown in Fig.(1) as follows: (9)

- Fig.1 (a) shows the flow through the nozzle when it is completely subsonic (i.e. nozzle isn't choked). The flow accelerates out of the chamber through the converging section, reaching its maximum (subsonic) speed at the throat. The flow then decelerates through the diverging section and exhausts into the ambient as a subsonic flow. Lowering the back pressure in this state increases the flow speed everywhere in the nozzle.
- Further lowering P_b results in Fig.1 (b). The flow pattern is exactly the same as in subsonic flow, except that the flow speed at the throat has just reached Mach equal to one. Flow through the nozzle is now choked since further reductions in the back pressure can't move the point of (M=1) away from the throat. However, the flow pattern in the diverging section does change as the back pressure is lowered further.
- As P_b is lowered below that needed to just choke, Fig.1 (c) the flow a region of supersonic flow forms just downstream of the throat. Unlike a subsonic flow, the supersonic flow accelerates as the area gets bigger. This region of supersonic acceleration is terminated by a normal shock wave. (5) The shock can occur only in steady state when there is a supersonic flow. the gas has to pass through a converging–diverging nozzle to obtain a supersonic flow. The location of the shock is determined by geometry to achieve the right back pressure. Obviously if the back pressure P_b, is lower than the critical value P* (the only value that can achieve continuous pressure) a shock occurs outside of the nozzle (10)
- The shock wave produces a near-instantaneous deceleration of the flow to subsonic speed. This subsonic flow then decelerates through the remainder of the diverging section and exhausts as a subsonic flow. In this regime if the back pressure is lowered or raised the

length of supersonic flow in the diverging section before the shock wave increases or decreases.

- If p_b is lowered enough the supersonic region may be extended all the way down the nozzle until the shock is sitting at the nozzle exit, Fig.(1) (d), because of the very long region of acceleration (the entire nozzle length) the flow speed just before the shock will be very large. However, after the shock the flow in the jet will still be subsonic.
- A further lowering of the back pressure changes and weakens the wave pattern in the jet. Eventually, the back pressure will be lowered enough so that it is now equal to the pressure at the nozzle exit. In this case, the waves in the jet disappear altogether, figure (e), and the jet will be uniformly supersonic. This situation, since it is often desirable, is referred to as the 'design condition', Pe=Pa.

MESH GENERATION

Mesh generation is often considered as the most important and most time consuming part of CFD simulation. The quality of the grid plays a direct role in the quality of the analysis, regardless of the flow solver used. Computational fluid dynamics can be used to reduce the experimental effort (wind tunnel) which is highly expensive and take a long time to collect all the data required. (11).

PROBLEM SPECIFICATION

Consider air flowing at high speed through a convergent divergent nozzle having a circular sectional area that varies with axial distance. the boundary conditions for the surfaces of nozzle were selected in inlet, outlet, centerline and wall.

the stagnation pressure P_o and stagnation temperature for air (as ideal gas) at the inlet had taken, (800kpa) and (313K) respectively. The static pressure at the exit P_b varies, with values equal to (780, 751, 700, 600, 500, 400, 360 and 270) kpa. The Mesh generation was done in ANSYS 12.1, Workbench software, and since the nozzle has a circular cross-section, it's reasonable to assume that the flow is axial, so the geometry to be created is tow-dimensional. In ANSYS Fluent, two solver technologies are available, pressure –based and density- based but the density based may give an accuracy advantage, therefore the density based solver was selected in present work for solving a high speed compressible flow. The axisymmertic form of the government equations was select, and since the energy equation is coupled to the continuity and momentum equations, in ANSYS fluent the energy equation needs to be turned. The Reynolds number is high so the viscous effects is expect to be confined to a small region close to the wall therefore, so it reasonable to model the flow a inviscid, this mean the viscous terms in the governing equations would neglect.

RESULTS AND DISCUSSION

The Mach number, static pressure, density, velocity and temperature distribution for the convergent divergent nozzle are implemented by using the ANSYS Fluent 12.1 software to solve the quasi-one dimensional nozzle flow.

Fig.'s (2 , 3 ,4 ,5 and 6) show the Contours of static pressure (Pascal) , Mach Number , density (kg/m^3) , velocity magnitude (m/s) and static Temperature (k) respectively, plot for different values of back pressure which are (780, 751, 700, 600, 500, 400, 360 and 270) kpa respectively. Table(1) shows the minimum and maximum values for characteristics of air along the nozzle. The behavior of fluid for static pressure (Pascal) , Mach Number , density (kg/m^3) , velocity magnitude (m/s) and static Temperature (k) are presented in fig.'s (7, 8, 9, 10 and 11) respectively.

When P_b (P_b =780kpa) is very nearly the same as stagnation pressure P_o (P_o = 800kpa) the flow remains subsonic throughout. the pressure dropping from P_o to a minimum value (P= 663kpa) which is greater than critical pressure P* (P* =422kpa form tables of one dimensional isentropic flow) at the throat, see fig.(8) and then increasing again to exit pressure P_e ($P_e = P_b$) at the exit, as the back pressure decreases, the throat pressure, which is lower than the back pressure, also decreases. This continues until P_b has dropped to a value (P_b = 751 kpa) at which the throat pressure becomes equal to the critical pressure P* and the Mach number at the throat approach to one. Further reductions in the back pressure (P_b = 700kpa) cannot affect conditions upstream of the throat, The nozzle is therefore, choked. As the back pressure is reduced below that value a region of supersonic flow develops just downstream of the throat. This region of supersonic flow is terminated by a normal shock wave. The shock wave increases the pressure, density and temperature see fig.'s (7, 9, 11) respectively and reduces the velocity and mach number to a subsonic value see fig.'s 8 and 10 respectively. The flow then decelerates subsonically until the pressure on the exit plane (p_e) is equal to the back pressure, As the back pressure is further reduced to values such as (600,500,400 and 360) kpa, the extent of the supersonic flow region increases, the shock wave moving further down the divergent portion of the nozzle towards the exit plane. When P_b equal to value in the range between 350 to 280 there are no solution because of diverging in iteration process that was done in (ANSYS fluent.12.1's software) see Fig.(12) as example for iteration to one case only. As P_b is decreased to and below 270kpa, conditions at all sections of the nozzle remain unchanged and the pressure on the exit plane p_e remains unchanged.

CONCLUSIONS

When the flow is compressible, density, cross-section area, flow velocity mach Number , static pressure and static temperature can all vary from section to section. The study had proceed to determine how characteristics of fluid change with axial location in a variable area nozzle when the fluid is an ideal gas and the flow through it, is steady and isentropic.

With the air entering the choked converging-diverging duct subsonically, only one isentropic solution exists for the converging portion of the duct. This solution involves an accelerating flow that becomes sonic at the throat of the passage. Two isentropic flow solutions are possible for the diverging portion of the nozzle—one subsonic, the other supersonic.

When back pressure very nearly the same as stagnation pressure the flow remains subsonic throughout. as the back pressure decreases, the throat pressure, which is lower than the back pressure, also decreases. Further reductions in the back pressure cannot affect conditions upstream of the throat. The nozzle is, therefore, choked. The shock wave increases the pressure, density and temperature and reduces the velocity and Mach number to a subsonic value. As back pressure is further reduced to a certain value, the extent of the supersonic flow region increases, the shock wave moving further down the divergent portion of the nozzle towards the exit plane. When P_b equal to value in the range between 280 to 350 there are no solution because of diverging in iteration process that was done in (ANSYS fluent.12.1's software) as example for iteration to one case only. As P_b is decreased to below 270kpa, conditions at all sections of the nozzle remain unchanged and the pressure on the exit plane p_e remains unchanged.

REFERENCES

[1] Khan, A. A. ; Shembharkar, T. R., (2008) ,"Viscous flow analysis in a convergent-divergent nozzle", Proceedings of the International Conference on Aerospace Science and Technology 26 - 28 June 2008, Bangalore, India,.

[2] Patrick H. O. ; William E. C. (1997), ”Compressible Fluid Flow”, The Mcgraw-Hill companies, INC..

[3] Kreith, F.; Berger, S.A.; et. al. (1999) , “Fluid Mechanics”, Handbook, Ed. Frank Kreith Boca Raton: CRC Press LLC,.

[4] Krause, E ; Aachen, R, (2005) "Fluid mechanics with problems and solutions and an aerodynamic laboratory " Aerodynamics Institute, Germany ,ISBN 3-540-22981-7 Springer Berlin Heidelberg New York.

[5] Nicholas J. Georgiadis, Teryn W. DalBello , Charles J. Trefny , and Albert L. Johns, (2006) "Aerodynamic Design and Analysis of High Performance Nozzles for Mach 4 Accelerator Vehicles " T44th AIAA Aerospace Sciences Meeting and Exhibit 9 - 12 January, 2006 Reno, Nevada

[6] Malley ,P. O. , Mattie, L. , Butts , P. (1999)"Determination of Critical Pressure Ratios In Convergent-Divergent Nozzles" May 7,

[7] Hubert, D., Sesmat, S., Giorgi, R. D., Gautier, D. and Bideaux, E. , (2008) , "Analysis of flow behaviour and characteristics of pneumatic components " Proceedings of the 7th JFPS International Symposium on Fluid Power, TOYAMA 2008 September 15-18, 2008

[8] Ekanayake, E. M. S. ; Gear, J. A. Y. Ding (2010) , " Flow simulation of a two dimensional rectangular supersonic convergent divergent nozzle" ANZIAM J. 51n (EMAC2009) PP.C377-C392,.

[9] Kirk, D. R., "Gas Turbine Engine Nozzles" mae 4261: air-breathing engines Mechanical and Aerospace Engineering Department Florida Institute of Technology.

[10]Genick Bar–Meir, "Fundamentals of Compressible Fluid Mechanics", Version (0.4.8.6

[11] Abaas, A. A., ,(2006)."Computational study of supersonic flow over aerodynamic configurations", University of Technology, Mechanical Engineering Department

Table (1): The minimum and maximum values for characteristics of air along the nozzle.

Back pressure P_b kPa	Distance from nozzle throat x(m)	Min static pressure (Pascal)	Max. Mach Number	Min. density (kg/m3)	Max. velocity magnitude (m/s)	Min. static Temperature (k)
780	0.1	663	0.523	11.4	149	202
751	0.11	411	1.02	-	-	-
700	0.4	202	1.55	4.9	373	144
600	0.55	126	11.86	3.48	419	126
500	0.7	88.3	2.09	2.7	447	114
400	0.85	57.1	2.37	1.98	476	100
360	0.9	49.7	2.45	1.79	484	96.7
270	1	39.4	2.6	1.51	496	90.5

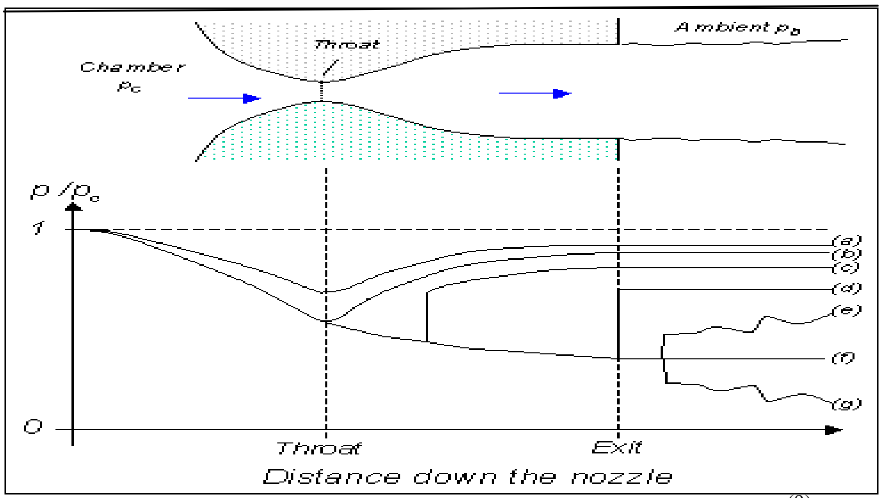

Fig.(1): the operation of converging-diverging nozzles. (9)

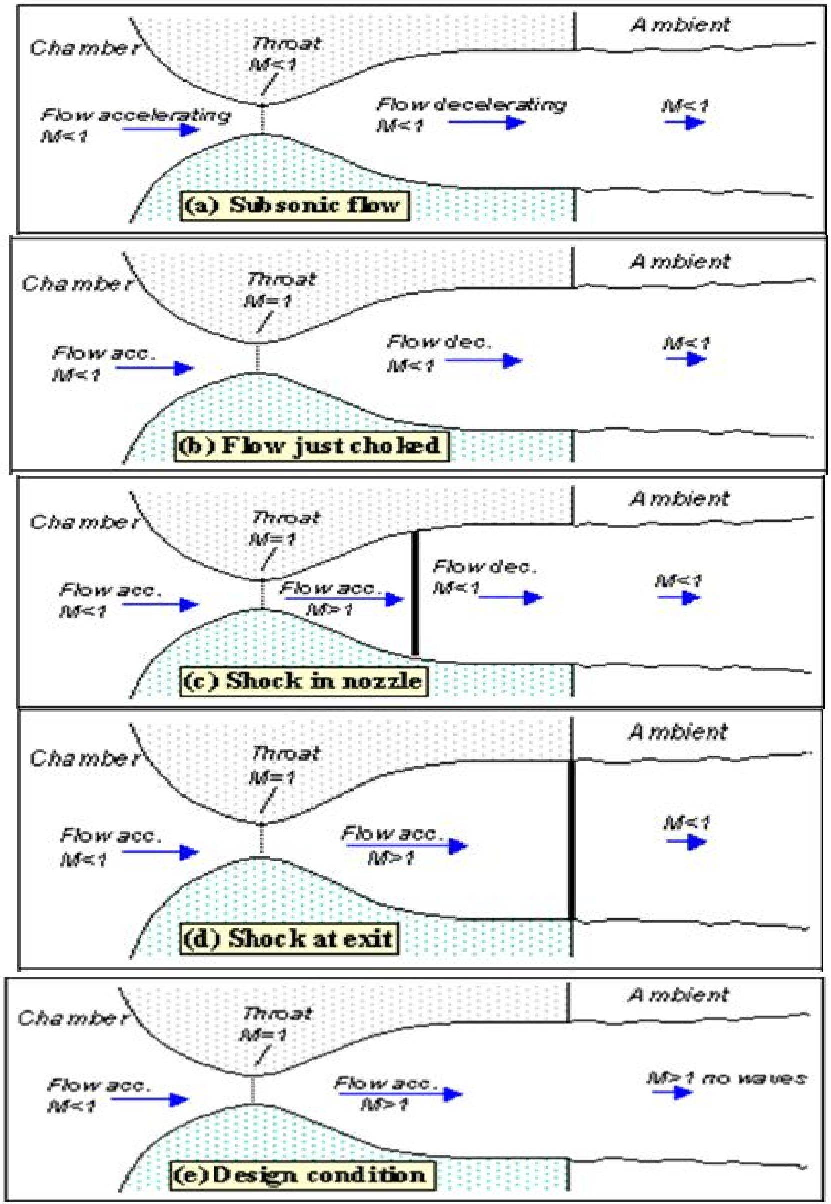

Fig.(1): the operation of converging-diverging nozzles. (9)

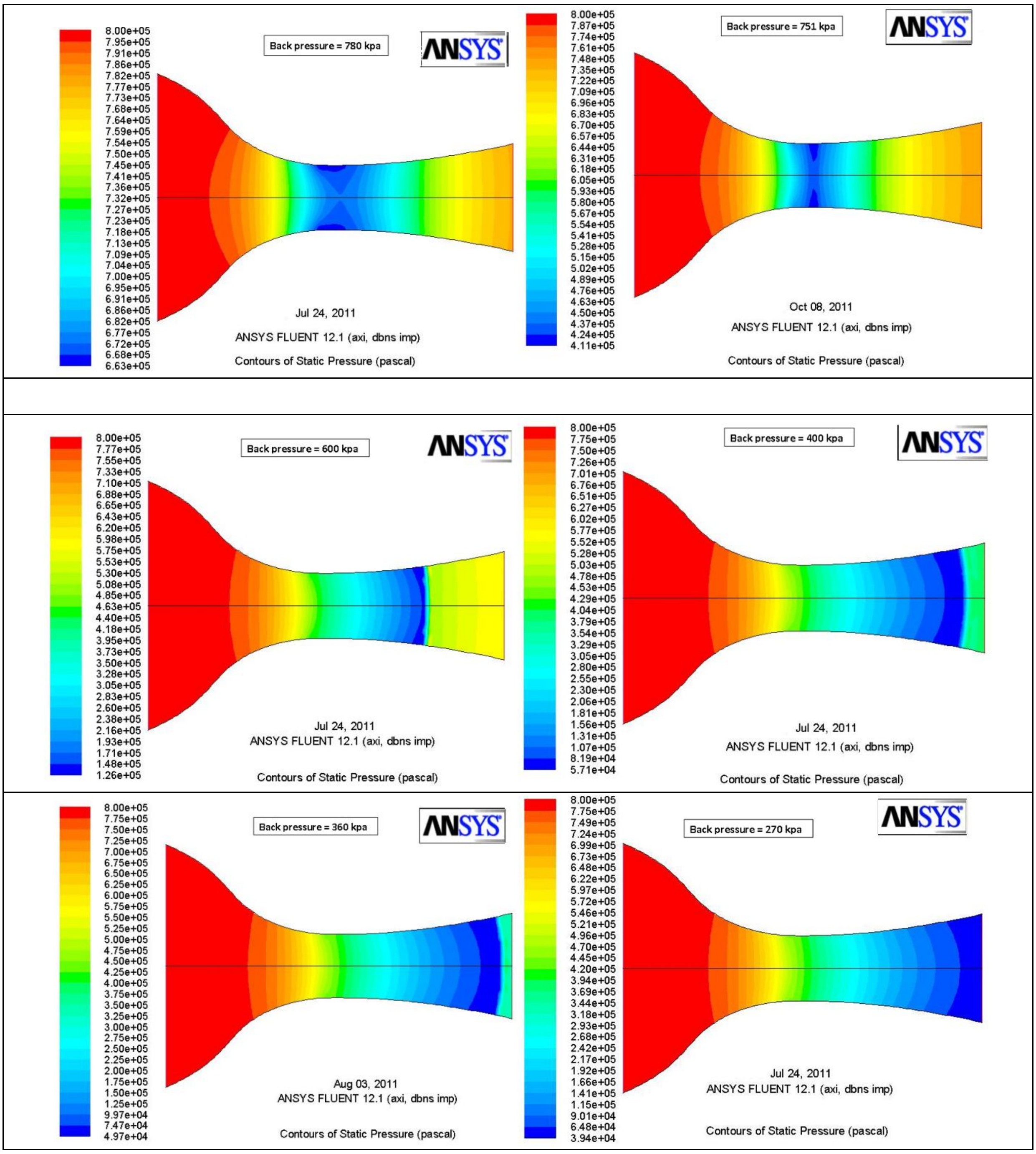

Fig.(2): Contours of static pressure (Pascal) , plot for different back pressure.

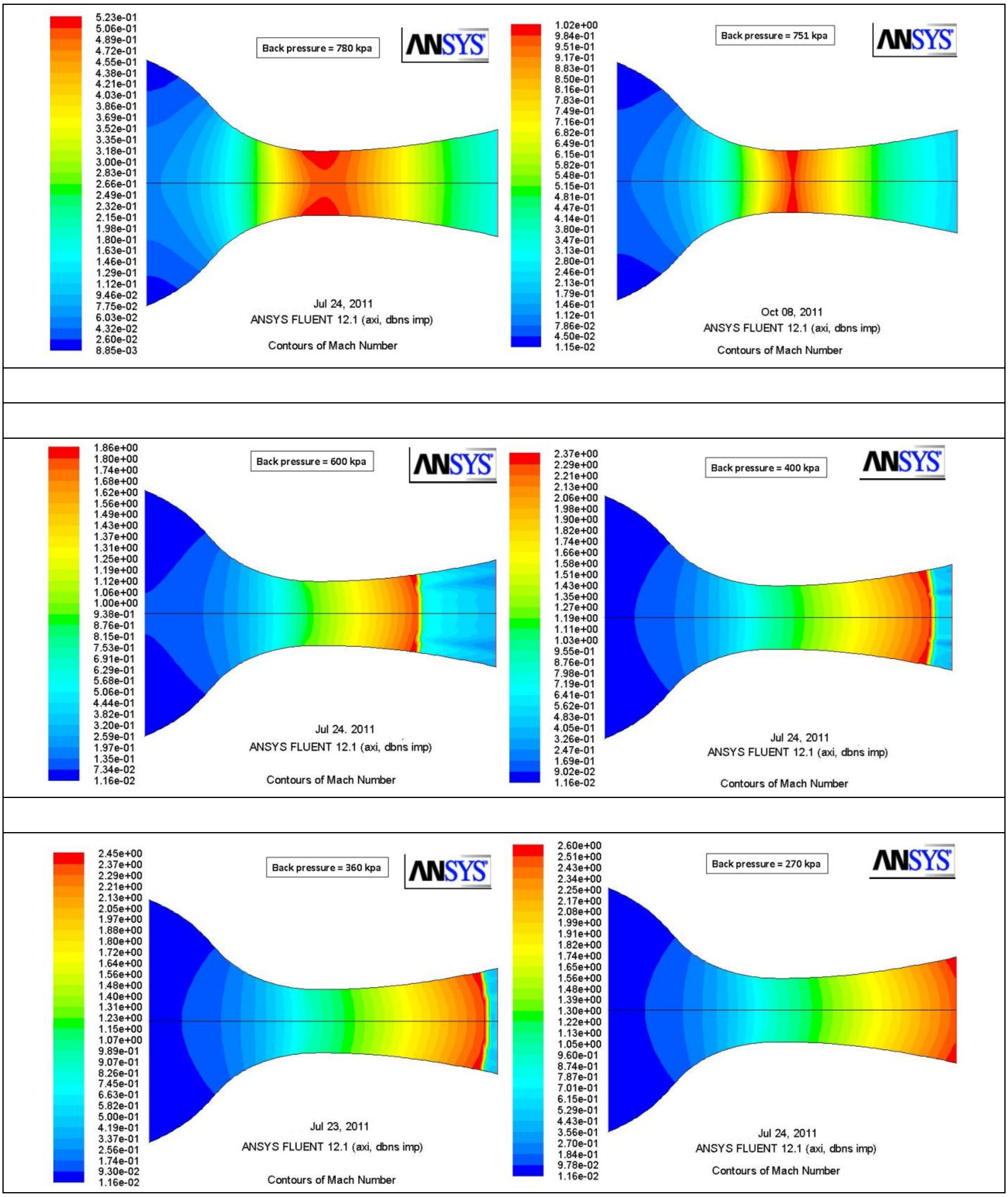

Fig.(3): Contours of Mach Number, plot for different back pressure.

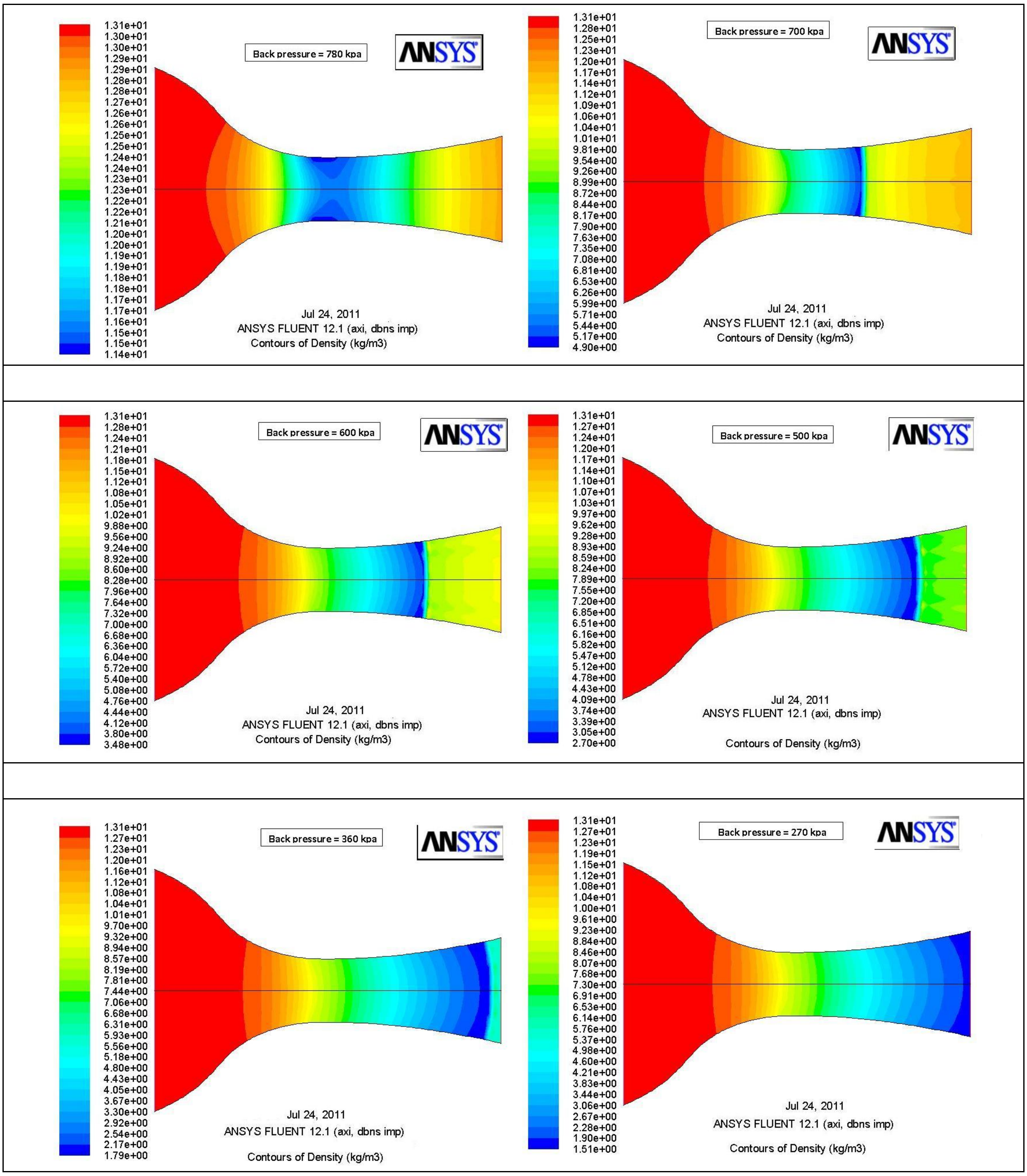

Fig.(4) : Contours of density (kg/m3) plot for different back pressure.

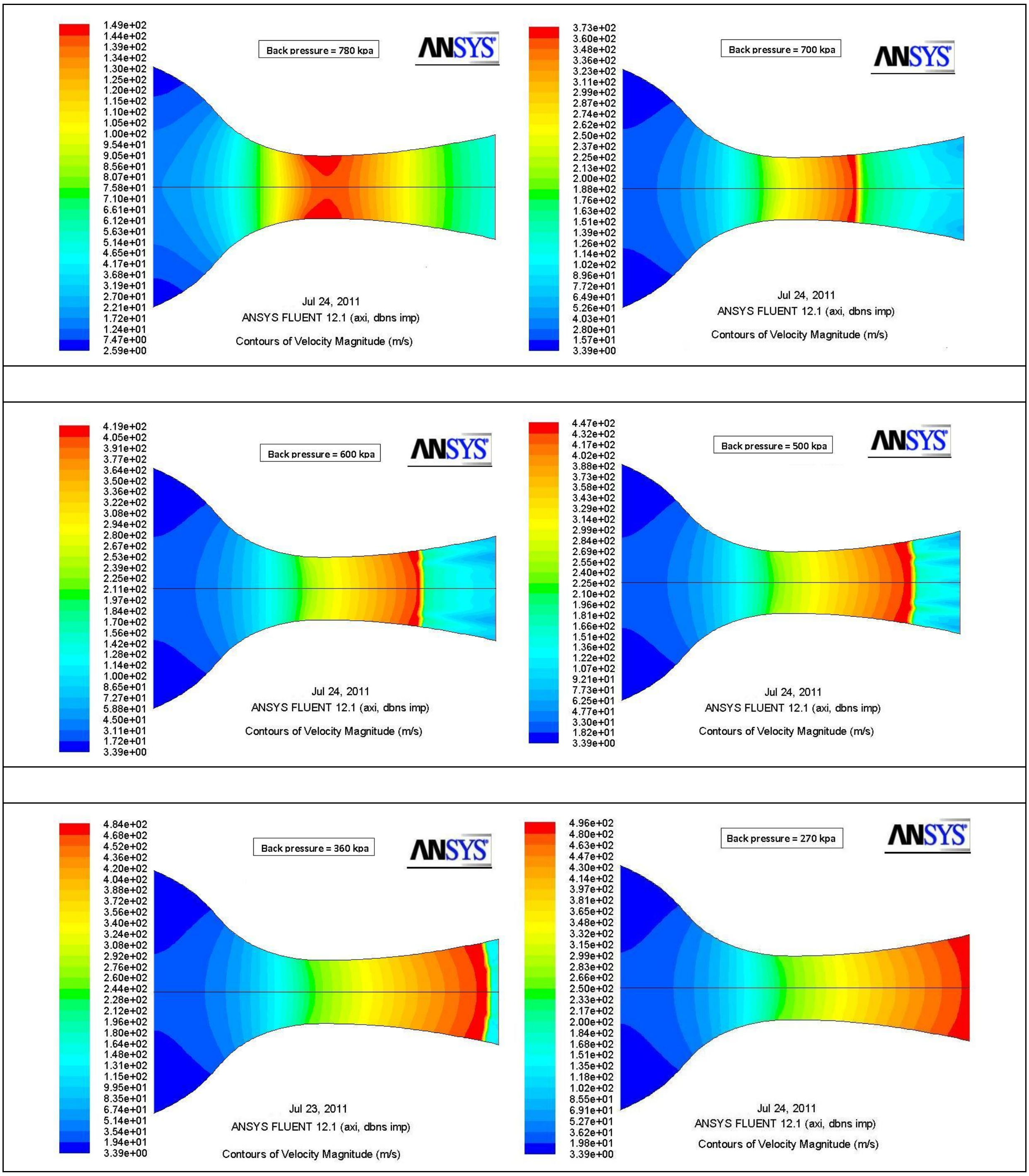

Fig.(5) : Contours of velocity magnitude (m/s) plot for different back pressure.

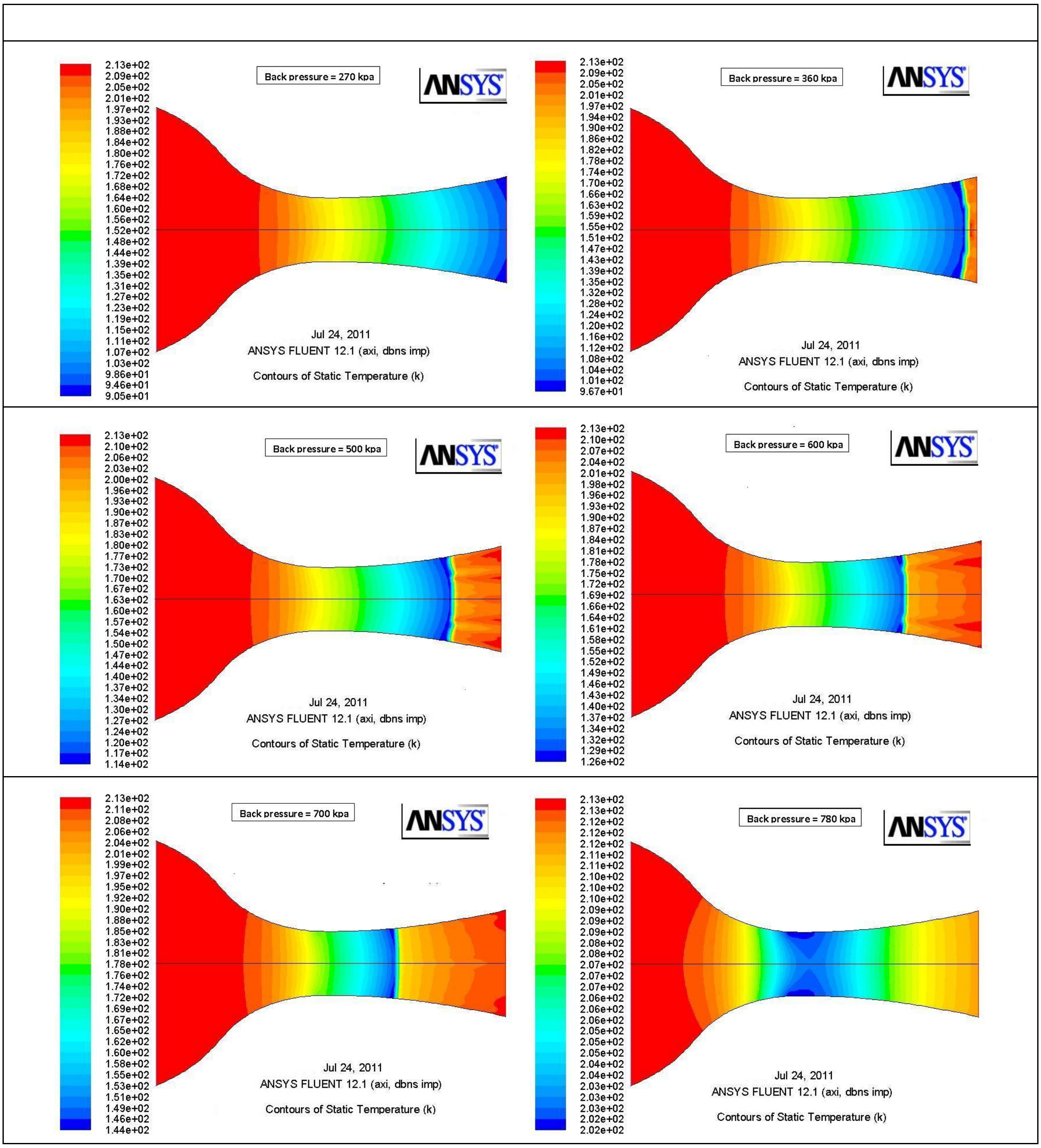

Fig.(6): Contours of static Temperature (k) plot for different back pressure.

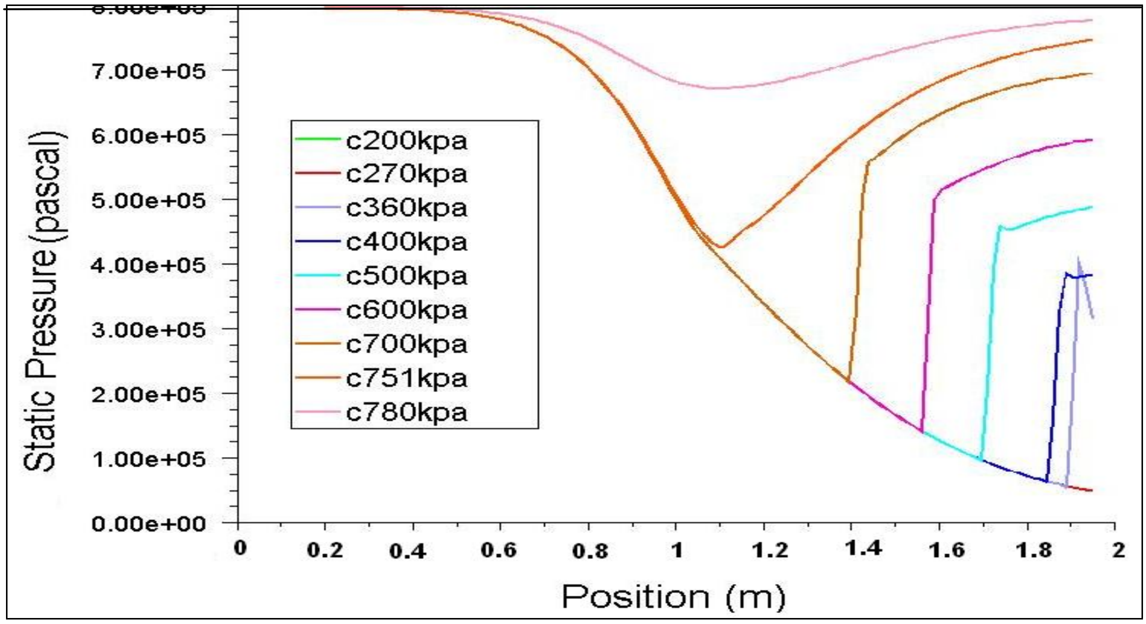

Fig. (7): Centerline Static Pressure plot for different back pressure.

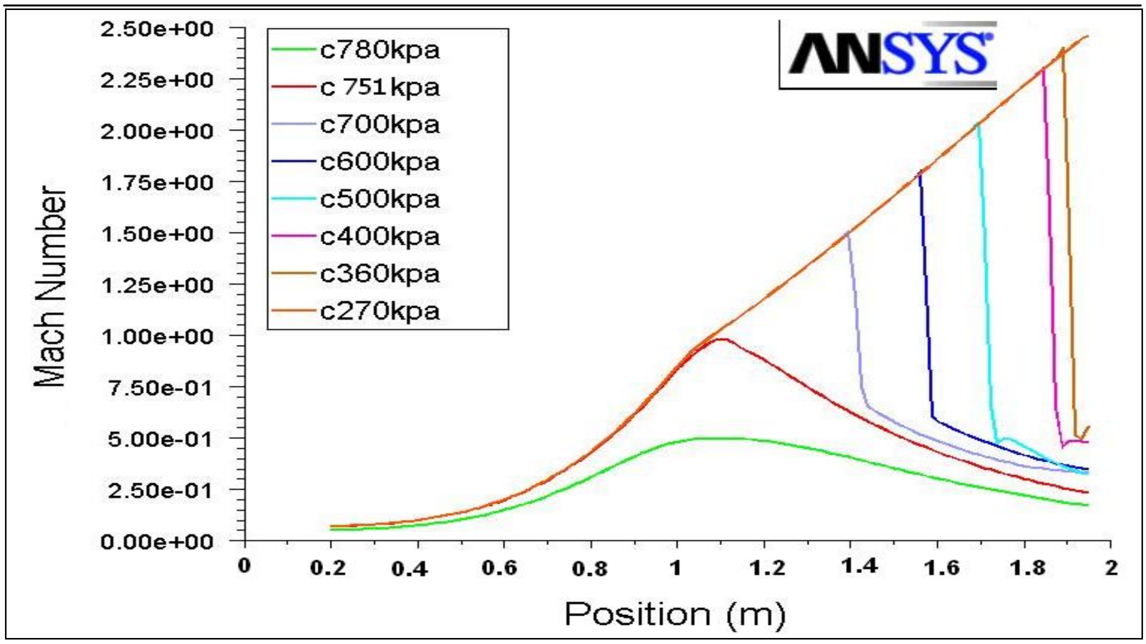

Fig.(8): Centerline Mach Number plot for different back pressure.

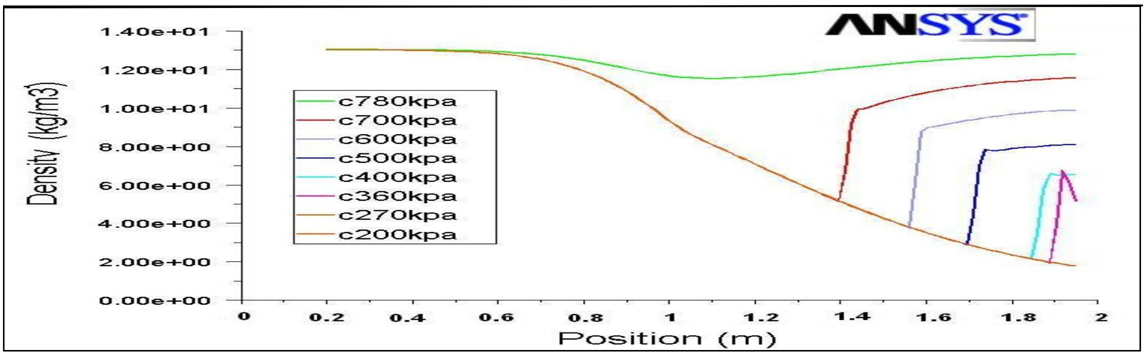

Fig. (9): Centerline Density plot for different back pressure.

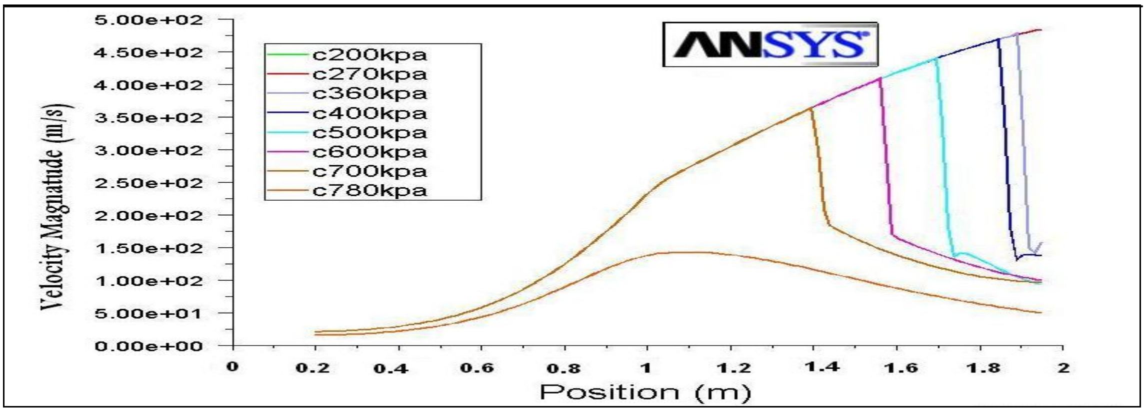

Fig. (10): Centerline Velocity magnitude plot for different back pressure.

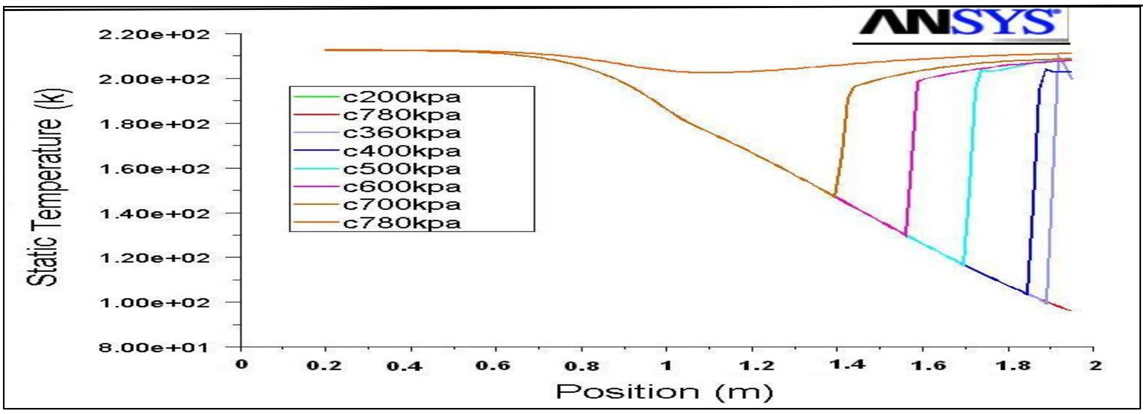

Fig. (11): Centerline Static Temperature plot for different back pressure.

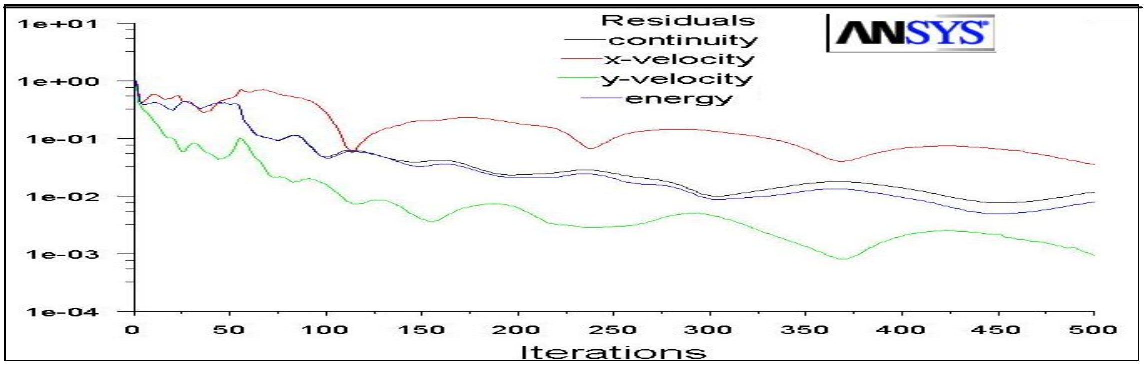

Fig.(12) : the iterations of processes to solve model's calculation (nozzle) that was done in (ANSYS fluent.12.1's software).

محاكاة تأثير الضغط الخلفي على سلوك المنفث الملتئم المنفرج

نزار منعم محمود
مدرس مساعد
كلية الهندسة_ جامعة ديالى

الخلاصـــة

تم في هذا البحث محاكاة الجريان المستقر للغازات خلال منفث ملتئم منفرج يمتلك مساحة مقطع متغيرة ، طبيعة الجريان تتوضح اذا اخذنا بعين الاعتبار كيفية تغير سلوك وخصائص الجريان عند تغير الضغط الخلفي عند مخرج المنفث.

تم استخدام برنامج الانسز . فلوينت الاصدار ١٢.١ لرسم توزيع خصائص الجريان (رقم ماخ ، الضغط الساكن ،كثافة الهواء ، مقدار سرعة الجريان ودرجة حرارة الجريان على طول المنفث لحل معادلات شبه احادية البعد اي متغير المساحة.

لا يوجد اي تأثير على سلوك الجريان عند انخفاض الضغط الخلفي كثيرا في المنطقة ما قبل عنق المنفث والتي تحدث ظاهرة الخنق عند العنق . ان وجود الموجة الصدمية يزيد الضغط والكثافة ودرجة الحرارة ويقلل السرعة ورقم ماخ الى قيمة الجريان تحت الصوتي ، وعندما يقل الضغط الخلفي اكثر ، فان امتداد منطقة الجريان الصوتي تزداد والموجة الصدمية تتحرك باتجاه مخرج المنفث .

الكلمات الدالة: المنفث الملتئم المنفرج ، الضغط الخلفي ، موجة صدمية ، جريان متغير المساحة ، الجريان شبه احادي البعد خلال المنفث