Eng. & Tech. Journal, Vol.31, Part (A), No.6, 2013

Numerical Investigation on the Performance of an External Compression Supersonic Air Intake Using By-Pass Technique

Dr. Jalal M. Jalil
Electromechanical Engineering Department, University of Technology/ Baghdad
Email:jalalmjalil@yahoo.com
Dr. Abdullateef A. Jadallah
Electromechanical Engineering Department, University of Technology/ Baghdad
Ahmed F. Mahdi
Al Khawarsmi College for Engineering& Science, University of Baghdad/ Baghdad

Received on: 10/6/2012 & Accepted on: 8/11/2012

ABSTRACT

Two-dimensional, supersonic flow field computation with by-pass holes has been conducted using Euler equation. An algorithm based on finite difference McCormack's technique is used to solve the subsonic-supersonic flow problem with and without by-passing. Trials on by-pass locations were also made. Operation without bypass shows the movement of the normal shock wave upstream which refers to an off-design operating regime. At a distance of 1.271 m measured from the cowl lip, the region of normal shock wave was localized near the throat which approximately represents the on-design perform 0ance condition (cells 8-9). When the flow is bypassed from the rear passages, the normal shock wave is sucked inside the flow passage save poor performance. Results show that by-passing provides a fast reaction to maintain a steady performance which meets the requirements of on-design operation. The Mach number variation corresponding to each case is presented. Results show that the tendency to unity Mach number near the cowl lip sector can be achieved using by-pass expelled. The influence of by-pass on the total pressure recovery and relative pressure is also discussed.

Keywords: Air intake, bypass, CFD, McCormack’s technique, Pressure recovery, Shock wave.

تفسير عددي لأداء آخذة هواء فوق صوتية ذات انضغاط خارجي باستخدام تقنية تمرير الهواء

الخلاصة

تناول هذا البحث مناقشة عددية لأداء آخذة هواء ثنائية الابعاد باعتماد تقنية تمرير الهواء. تم استخدام خوارزمية تعتمد الفروقات المحددة تقنية McCormack لحل الجريان الفوق صوتي بدون

https://doi.org/10.30684/etj.31.6A1
2412-0758/University of Technology-Iraq, Baghdad, Iraq

ومع تمرير الهواء خارج الآخذة لأثبات تأثير تقنية التمرير اختبرت منافذ متعددة لتمرير الهواء لمسايرة اداء الاخذة للوصــول الى الاداء التصميمـي. للهيكلية المعتمدة, المنفذ الذي يبعد على مسافة 1.712 m من مقدمة المخروط اعطى احسن اداء بإيصال عدد ماخ الى واحد عند منطقة العنق وللخلايا (8-9). كلما ابتعدت منافذ التمرير فان الموجة الصدمية تنسحب داخل الاخذة مؤدية الى اداء سيء للآخذة. أظهرت النتائج ان تقنية تمرير الهواء هي وسيلة سريعة لإعادة أداء الآخذة الى النظام المستقر وقد أضيف تاثير التمرير على نسبة استرداد الضغط الكلية للآخذة.

NOMENCLATURE

Symbol	Definition	Units
E,F,V	Column vector in body fitted coordinates	
I	Identity matrix	
J	Jacobian of coordinate transformation.	
$\dot{m}$	Mass flow rate	Kg/sec
M	Mach number.	
P	Static pressure.	N/m^2
P_o	Stagnation pressure.	N/m^2
R	Gas constant .	J/kg.K
t	Time	Sec.
T	Static temperature.	K
u, v	Velocity component in x and y Coordinate directions	m/s
$\bar{U}$, V	Conservation velocity component in η and ξ coordinate directions.	
x, y	Cartesian coordinates.	
n	Unit normal vector.	

INTRODUCTION

Designers of air intakes look for minimizing drag and providing maximum total pressure recovery. Furthermore; it must insure controllable flow matching for all operating conditions. To attain the objective of maximum pressure recovery; normal shock wave should form at the minimum Mach number location i.e.; when it formed nearest to the throat. A shock train, a series of shock waves during compression, can occur in the supersonic region of the inlet and is eventually terminated by a normal shock. The location of the shock train categorizes the type of inlets as internal, external, or mixed compressions [(1)].

CFD will make full use of every increase in computer power of this century. Increasingly, computational fluid dynamics will compete with flight tests, not just with wind tunnels, and will be validated by flight tests. Integration with other disciplines will allow predicting crucial phenomena such as flutter, sonic fatigue, and pilot-induced oscillations[2]. Experimental and computational investigations have been made by S. Das J. K. Prasad [3] to obtain the details of the flow field of a supersonic air-intake with different cowl deflection angles and back pressures at the exit. External compression inlets can maintain a balance between an acceptable total pressure ration and cowl drag up to flight Mach number of 2.5[4]. Sanjay M. et al [5] presented a numerical simulation of a 2-D mixed compression supersonic inlet by solving time dependent compressible Euler's equations. The investigation of a performance simulation method of the intake and fan component of a two-shaft high-bypass turbofan is represented by in an axi-symmetric 2D fashion [6]. The design has been done to insure adequate air capture for all regimes of operation and the effect of boundary bleed on the pressure recovery was also studied. Biringen [7] outlines a time-implicit, finite-difference solution procedure for Euler equations to calculate two-dimensional inlet flow fields. Moretti[8], presented an efficient Euler computational technique for two-dimensional Euler equation at any Mach number of any shape and type, whose interaction can be treated by this technique. The two dimensional inner steady flow of hypersonic inlets was numerically simulated in different free stream conditions and back pressures, and two different inlets nonstart phenomena were analyzed [9].

The specification of back pressure at the exit of the intake leads to flow reversal and, therefore, difficulties in carrying out the numerical computations. Interests on the full integration of two-dimensional component models with a low fidelity cycle program were recently concerned[10]. Two-dimensional models were used in the engine cycle analysis to provide a more accurate, physics- and geometry-based estimate of intake and fan performances. Flow treatment is evaluated in off-design point condition. An analysis of fluid dynamic flow filed is done with solving of Euler's equations and by means of CFD codes and compare with analytical and parametrical analysis [11]. Implementation of a CFD code assessed for the verification of steady and unsteady flows associated with supersonic inlets. Verification procedures include grid convergence studies and the behavior of boundary condition.

The present work mainly focuses on use of bypass technique in external compression supersonic air intake. Bypass flow concept was originally proposed for attenuating the sonic boom strength by mean of enforcement of flow to take the pattern of unity Mach number at the cowl lip.

The computation of inlet flow field has been the subject of number of investigations. The governing equations of mass, momentum, energy, and state equation have been solved to obtain the complete flow field. The improvement in performance of the intake is achieved by one of the conventional techniques which is the by-pass as in Figure (1). The main issue of the present work is to study the two-dimension ramp inlet flow field for an air breathing in a two-shock system using bypass under the conditions specified in the next articles. The flow domain typically terminates near the compressor face to avoid the complexity of modeling the geometry and dynamics of the compressor.

THEORETICAL CONSIDERATIONS

External compression supersonic diffuser with normal shock placed in cowl lip, create a normal shock with variable locus that move superior position and create instability in flow filed. The performance of the air intake is characterized by operation regimes. When the mass flow rate captured by the intake is the same as required by the compressor; the air intake is then in the so called critical operation regime. If the mass flow rate delivered is greater than required(subcritical operation) which will not be accepted by the compressor; the normal shock wave moves upstream and the flow will spilled over which leads to a spillage drag and poor performance sequentially. The supercritical operation regime is reached when the mass flow required by the compressor is greater than that delivered by the intake. In this case the shock wave will be sucked inside the convergent part. The performance is then deteriorates and this mean lower thrust and higher thrust specific fuel consumption.

COMPUTATIONS

The calculation of the inlet flow field is of considerable importance to the efficient design of air breathing engines. CFD models have become a focal point for the improvement of performance simulation tools in terms of introducing higher fidelity to some or all the engine components [6]. It is an external compression inlet, including two external shocks and one normal shock. A developed technique employs time-marching solution technique for numerically solving the time dependent Euler Equation. The main factors that influence this choice are the ability to use the same difference operator in both subsonic and supersonic regions of the flow field. To solve the finite difference pertaining, the MacCormack's predictor-corrector technique was used. The main geometric parameters of the supersonic inlet are referred to Figure (2).

The solution procedure employs the Euler equation, which is written in a conservation-law form for two-dimensional flows of a perfect gas. Because the governing equation in fluid dynamics contains partial differentials and it is too difficult in most cases to solve analytically, these partial differentials are generally replaced by the finite difference terms. This procedure discretizes the field into finite number of states. These states, when plotted, form a grid or mesh. It is at these states, or field points, that the solution is found.

In order to numerically solve the governing partial differential equations PDEs of fluid mechanics, approximations to the partial differentials are introduced. These approximations convert the partial derivatives to finite difference expressions, which are used to rewrite the PDEs as algebraic equation. The approximate algebraic equations, referred to as finite difference equations FDEs, are subsequently solved at discrete points within the domain of interest. Therefore, a set of grid points is found within the domain, as well as the boundaries of the domain, must be specified. The creation of such a grid system is known as grid generation.

Governing Equations

For high Reynolds number flow, viscous effects are confined to the vicinity of the surface, where large velocity gradients exist. This region is known as the boundary layer. Outside the boundary layer, the velocity gradients are negligible resulting in zero shear stresses. This region is called the inviscid region, and solution procedures for the inviscid flow region are governed by the Euler equations and the solution of this research depends on it, which is written in conservation-law form for two-dimensional flows of a perfect gas [22].

The general compact vector form is given as:-

$$\frac{\partial U}{\partial t}+\frac{\partial E}{\partial x}+\frac{\partial F}{\partial y}=0$$

where: $$U=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho e_t\end{bmatrix},\quad E=\begin{bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ u(e_t+p)\end{bmatrix},\quad F=\begin{bmatrix}\rho v\\ \rho uv\\ \rho v^2+p\\ v(e_t+p)\end{bmatrix} \qquad \ldots(1)$$

u and *v* are the velocities along the *x* and *y* coordinates, respectively, *p* is the pressure, ρ is the density, and e_t is the total energy per unit volume. And *U*, *E*, *F* are the fluxes vectors.

To transform the Euler Equation (1) into curvilinear coordinates (ξ,η), an independent variable may be written as follows:-

$$\frac{\partial \bar{U}}{\partial t}=-\frac{\partial \bar{E}}{\partial \xi}-\frac{\partial \bar{F}}{\partial \eta}$$

Where: $\bar{U}=U/J$, $\bar{E}=\dfrac{E\xi_x+F\xi_y}{J}$, $\bar{F}=\dfrac{E\eta_x+F\eta_y}{J}$

The time step (*Δt)* cannot be arbitrary, rather it must be less than some maximum values for stability, it was stated that *Δt* must obey the Courant-Friedriches-Lowry criterion *CFL*. The *CFL* criterion states that physically the explicit time step must be not greater than the time required for a sound wave to propagate from one grid to next. The maximum allowable value of *CFL* factor for stability in explicitly time dependent finite difference calculation can vary from approximately 0.5 to 0.1[12]. To determine the value of time step, the following version of the *CFL* criterion [20]is used. Where $a_{i,j}$ is the local speed of sound in meters per second, and *C* is the constant number.

$$(\Delta t_{CFL})_{i,j}=\left[\left|\frac{u_{i,j}}{\Delta\xi}\right|+\left|\frac{v_{i,j}}{\Delta\eta}\right|+a_{i,j}\sqrt{\frac{1}{\Delta\xi^2}+\frac{1}{\Delta\eta^2}}\right]^{-1} \qquad \ldots(2)$$

and $\Delta t = \min\,[C\,(\Delta t\ CFL)\ i,j]$.

Boundary Conditions

The Euler equation has an unlimited number of solutions. What makes a solution unique is the proper specification of initial and boundary conditions for a given PDE (Euler equation). A set of boundary conditions must be specified, it referred to as the “analytical boundary condition” Once the PDE is approximated by a FDE, Thus the FDE will require additional boundary conditions. This boundary condition will be referred to as “numerical boundary condition”. In order to develop a proper boundary condition, the following points must be considered:

1. The physics of a particular problem must be modeled correctly.
2. The physical conditions must be represented correctly by mathematical expressions.
3. Additional numerical boundary conditions may be required. These boundary conditions are usually specified by extrapolation from interior solution.
4. The manner in which boundary conditions are specified must be considered in overall stability and accuracy of the numerical scheme used to solve the system.
5. The boundary condition may be applied explicitly or implicitly [24].

As for the problem under consideration, there are five types of boundaries: solid, bypass, inflow, outer and outflow.

Solid Boundary Condition

For the three solid boundary conditions (two ramp (at 5^o & 9^o), upper cowl surface and the lower cowl surface), the tangency grid body surface must be satisfied for inviscid flow. The components of the momentum equation for the two-dimension flow may be expressed as:

In x-direction :

$$\frac{\partial}{\partial t}(\rho u)+\frac{\partial}{\partial x}(\rho u^2+P)+\frac{\partial}{\partial y}(\rho uv)=0 \qquad \ldots (3)$$

In y-direction:

$$\frac{\partial}{\partial t}(\rho v)+\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial x}(\rho v^2+P)=0 \qquad \ldots (4)$$

from mass flow rate definition

$$m^{\bullet}=\rho.V.\bar{n} \qquad \ldots (5)$$

Let $\bar{U}=v\xi_y+u\xi_x$; $\bar{V}=v\eta_y+u\eta_x$

m. = 0. at the surface of a nonporous surface. The grid system has been assumed to be independent of time with some mathematical steps, and after transformation to body fitted coor

$$\left\{ \begin{array}{l} \eta_x \left(\frac{\rho u \bar{U}}{J}\right)_\xi + \eta_y \left(\frac{\rho u \bar{U}}{J}\right)_\xi + \eta_x \left(\frac{\rho v \bar{V}}{J}\right)_\eta + \eta_x \left(\frac{\rho u \bar{V}}{J}\right)_\eta \\ + \eta_x \left(\frac{\xi_x P}{J}\right)_\xi + \eta_x \left(\frac{\eta_x P}{J}\right)_\eta + \eta_y \left(\frac{\xi_y P}{J}\right)_\xi + \eta_y \left(\frac{\eta_y P}{J}\right)_\eta = 0 \end{array} \right\} \quad \ldots (6)$$

A finite difference equation for the upper equation is obtained equation.6, as a second order central difference approximation for the ξ derivatives and a second order forward difference approximation for η derivatives are used as illustrated in Figure (3).

For the lower surface of cowl intake V=0, and from equation 5, a second order central difference approximation for ξ derivatives and second-order backward difference approximation for η derivatives are used**,** except the bypass. It is shown on Figure (4).

Bypass Boundary Condition

In bypass section there are two nodes that made the air exiting from the intake to the ambient air then the calculation of the influence of high pressure inside the inlet must be entering in the substantial nodes calculation as inviscid flow region that governing by the Euler equations. A second order central difference approximation for ξ derivatives and second-order central difference approximation for η derivatives are used, in order to emulate air impulsion from the high subsonic pressure (low air speed) region to the low supersonic pressure (high air speed) region, with deferent nodes selected from the cowl lip point to the end of cowl body.

To carry out the bypass during the numerical simulation, a series of holes were considered measured from the leading edge of the cowl lip. Table (1) shows the location of pairs of nodes which represent the bypass gates.

Table (1) Location of Bypass Measured from the Cowl Lip.

Bypass Location	**Distance from the cowl lip**	**Bypass Location**	**Distance from the cowl lip**
Bypass 2-3*	1.016-1.101 (m)	Bypass 16-17	1.611-1.696 (m)
Bypass 4-5	1.011-1.186 (m)	Bypass 18-19	1.696-1.781 (m)
Bypass 6-7	1.186-1.271 (m)	Bypass 20-21	1.781-1.866 (m)
Bypass 8-9	1.271-1.356 (m)	Bypass 22-23	1.866-1.951 (m)
Bypass 10-11	1.356-1.441 (m)	Bypass 24-25	1.951-2.036 (m)
Bypass 12-13	1.441-1.526 (m)	Bypass 26-27	2.036-2.121 (m)
Bypass 14-15	1.526-1.611 (m)	Bypass 28-29	2.121-2.206)

*Two apparent cells are considered as the flow passage and its distance measured from the cowl lip.

Outer Flow Boundary

The upper outer flow boundary is the air flow out from the numerical simulation of two-dimensional three shock inlet system at +0.8 meter in the positive *y*-direction. To calculate the properties at this boundary, first order backward transformation derivatives are used. The lower outer flow boundary is the air flow out from the numerical simulation of two-dimensional three shock inlet system at -0.8 meter in the negative *y*-direction, the first order forward transformation derivatives are used to calculate the properties.

Out Flow Boundary

The outflow boundary illustrated in Figure (2) represents air flow out from the numerical simulation of two-dimension ramp inlet 2.2 meter far from the original point O above the inlet duct and the airflow out from the ramp inlet duct.

a. Using background derivatives to obtain the air flow properties above the inlet duct.

b. For critical and supercritical cases (subsonic outflow boundary) the back pressure at the duct outflow boundary ($Po= 0.9\ Po_{inflow}$) was set to a value high enough to ensure subsonic outflow boundary and the other properties are obtained from the background transformation derivatives [12]. To do the abovementioned computations, a computer program developed and adapted to solve the flow with bypass technique. The following flow chart represents the steps of solution.

Discussion

The main task of the air intake is to deliver the flow that meets the requirements of the compressor. The extra flow is to be expelled form the inlet by bleeding or bypassing. The bypass technique is numerically investigated. Passages for bypassing are proposed at certain locations. This was conducted by utilizing two apparent cells along the upper surface of the air intake up to the rear of it to expel with same flow conditions at these locations. The flow is processed with and without bypassing. To do so, a computer program written in Fortran was built. The algorithm is based on finite difference McCormack's technique is used to solve the subsonic-supersonic flow problem. Figure (7a) shows the mach contour in case of without bypassing under the specified conditions. The region of unity Mach number is moved upstream and the flow reaches the entrance of the inlet with subsonic flow which indicates deterioration in the inlet performance and the flow is spilled out. The increase of spillage air leads to an increase in spillage drag. Figures (7-b, c, d,) and f show the Mach contours for a number of the processed cases. They reveal that as the flow was passed from flow passages, the region of unity Mach number is pulled near to the throat. The location of 8-9 may considered as the nearest case to the on-design condition. This is because of the vicinity of unity Mach number from the cowl lip which may regards as the critical operation regime that gives best performance. The aft holes of bypass lead to sucking the normal shock inside the inlet. When the rear bypass passages were employed, the region of normal shock moves up stream because of the effect of the ram installed in front of the compressor. To give a well illustration, a full flow pattern was presented for with and without bypass as shown in Figure (8). It reveals the effect of total pressure on the flow distribution inside the air intake. The flow pattern is constrained by the pressure recovery interplayed with other operating conditions. Similar figures are presented for cases of with and without bypass. The history of pressure recovery across the air intake for all the cases considered is revealed in Figure (10). Results show the worst case is without bypassing as the pressure recovery reaches the lowest level in front of the inlet entrance. The variation of Mach number along the air intake with and without bypass is presented on Figure (11). It shows the movement Mach number toward unity near the cowl lip with utilizing the bypass technique which is desired. The influence of bypass on the relative pressure is shown in Figure (12). Employing bypass leads to a reasonable range of pressure recovery in front of the air intake that meets the requirements of recent technologies of external compression supersonic air intakes.

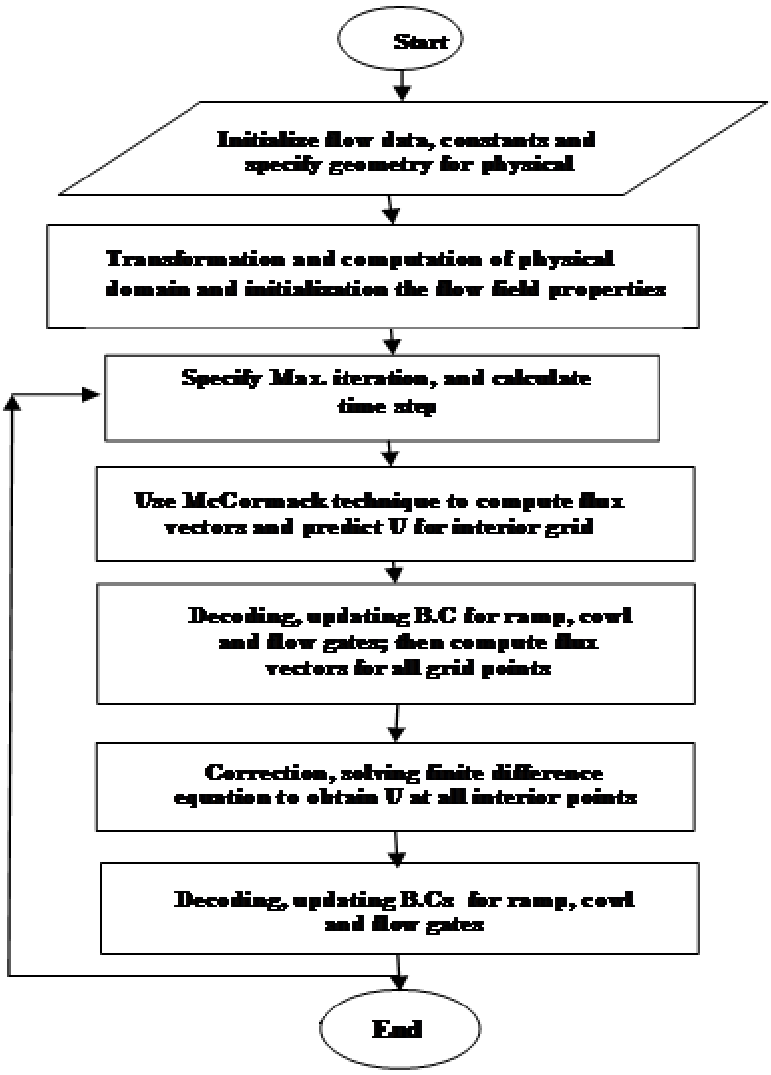


Figure (6) Flow Chart of the Computer Program.

CONCLUSIONS

The bypass technique for improving the air intake performance is numerically investigated.

A computer program written in Fortran based on McCormack's technique was used to perform the objectives of this work. Proposed holes at certain distances on the upper surface of the air intake have experienced to reflect the use of by pass on the performance of the air inlet which in turn affects the performance of the air breathing engine as whole. The effect of various total pressure recovery on the flow distribution inside the air intake and its duct is also conducted. The variation of Mach number, Pressure recovery and the relative pressure were presented along the air intake in order to give a self explanatory and understanding phenomena on the behavior of the flow under the specified operating regimes with and without bypass. The bypass technique is a fast reacting way in getting the on-design performance of the air intake and the air breathing engine consequently.

REFERENCES

[1]. Hyoung Jin Kim et al., Flow Simulation of Supersonic Inlet with Bypass Annular Duct, Journal of propulsion and power, Vol. 27, No. 1, January, 2011.

[2]. Spalart and D. R. Bogue, P. R. The role of CFD in Aerodynamics, Off-design, the aeronautical journal, June, pp. 323-329, 2003.

[3]. Das J. K. Prasad, S. Starting Characteristics of a Rectangular Supersonic Air-intake with Cowl Deflection, The aeronautical journal, March 2010 Volume 114 NO 1153.

[4]. Jack D. Mattingly et al., *Aircraft Engine Design*, AIAA Series, 1987.

[5].Sanjay M. etal., Numerical Investigation of Supersonic Mixed Compression Inlet Using Euler's Equation, National conference of Air Breathing Engines and Aerospace Propulsion; India; Hyderabad; 2000, pp 561-571.

[6]. Mund and P. Pilidis, F. C. Performance Simulation of a High-bypass Turbofan with a 2D Representation of the Intake and Fan Components, The aeronautical journal, Nov. 2008 VOLUME 112 NO 1137

[7]. Shaikh M. V. Wind Tunnel Testing of 2-D Supersonic Air Intake, National conference of Air Breathing Engines and Aerospace Propulsion; India; Hyderabad; 2000.

[8]. Moretti, and Gino, Efficient Euler Solver with Many Applications, AIAA Journal, Vol. 26, No.6, p.p. 755-757, June 1988.

[9]. Bao, Bao, Bao, W. Z. Xie and Y. Fan., A CFD Assessment of Classifications for Hypersonic Inlet Start/Unstart Phenomena, The aeronautical journal, APRIL 2008 VOLUME 113 NO 1142

[10]. Farajpour Khanaposhtani et al., V. A New Approach for Supersonic Diffuser Design, Journal of Applied Sciences Research, 6(5): 401-414, 2010.

[11]. Pachidis, V. P. Pilidis and L. Marinai;,"Towards a full two dimensional gas turbine performance simulator" The aeronautical journal, JULY 2007 433.

[12]. John. D. Anderson, TR., Computational Fluid Dynamics, Department of Aerospace Engineering University of Maryland, McGraw-Hill, New York, International Editions 1995.

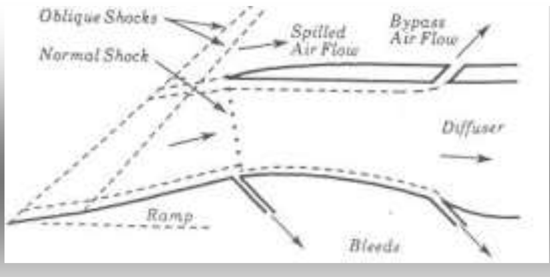


Fig. 1. External Compression Air Intake with Bypass and Bleeding[4]

Figure (1) External Compression Air Intake with Bypass and Bleeding[4].

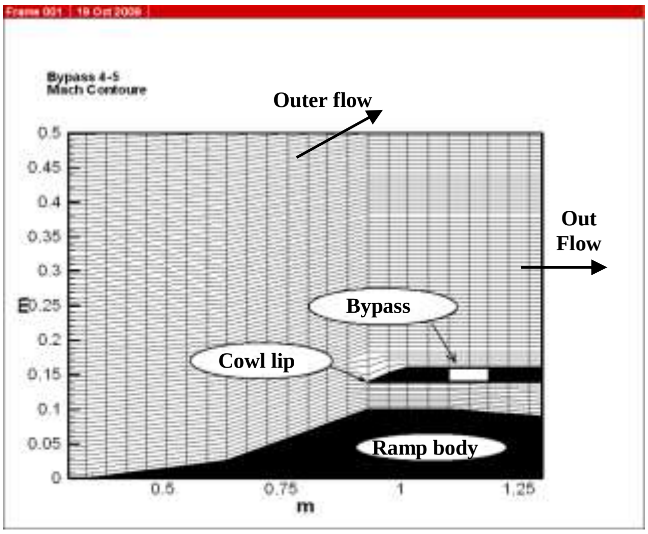


Figure (2) the air intake geometry with the types of boundaries.

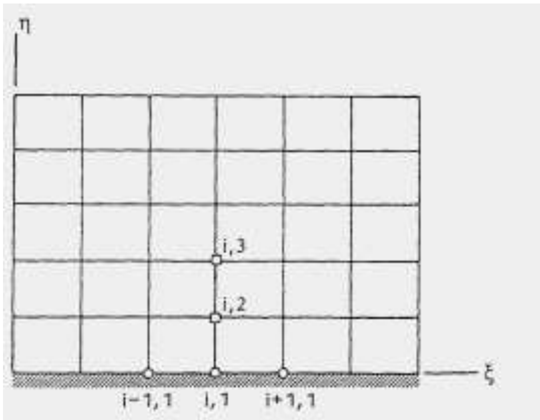


Figure (3) Grid points for upper cowl surface in computational domain.

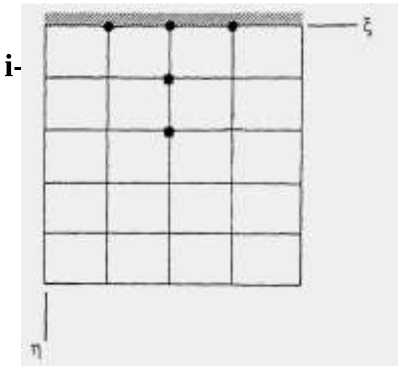


Figure (4) Grid points for lower cowl surface in computational domain.

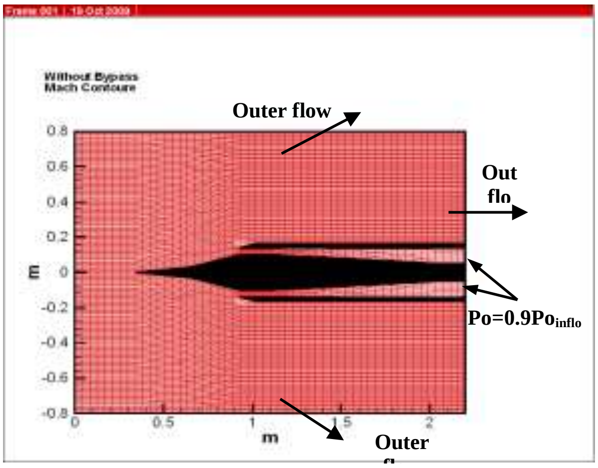
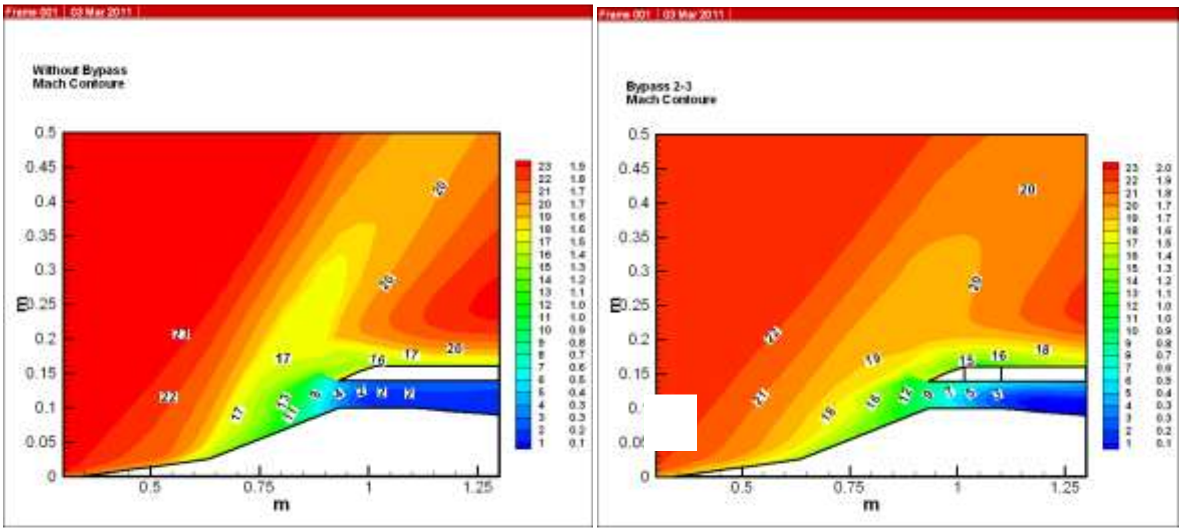
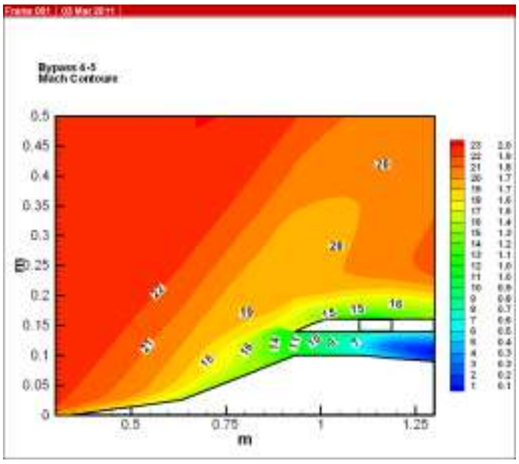


Figure (5) the outer flow boundary shown on the air intake.

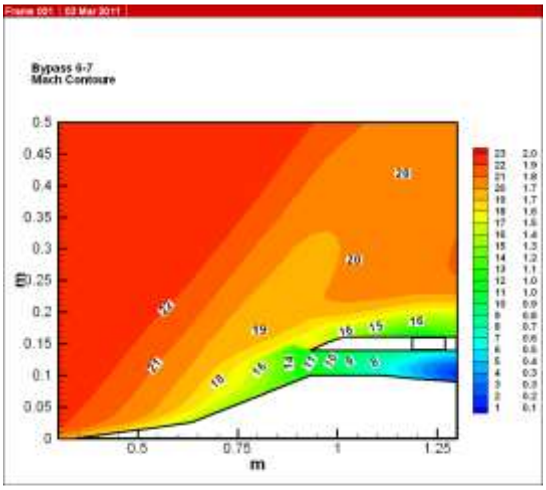


(a) Without bypass Technique

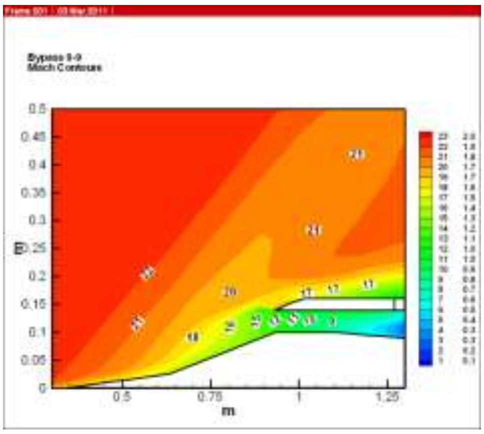
(b) With bypass at nodes 2-3.



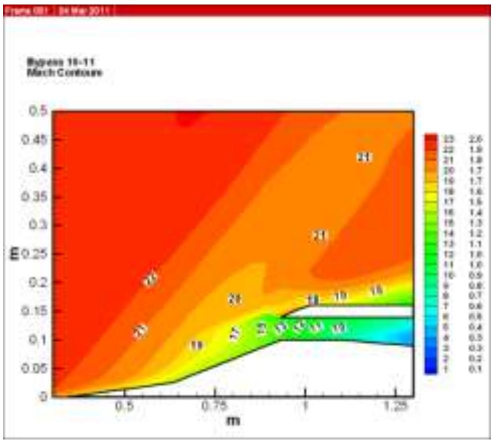
(c) With bypass at nodes 4-5



(d) With bypass at nodes 6-7



(e) With bypass at nodes 8-9



(f) with bypass at nodes 10-11

Fig ure (7) Mach Contour (a) without bypass, (b, c, d, e, and f) with bypass.

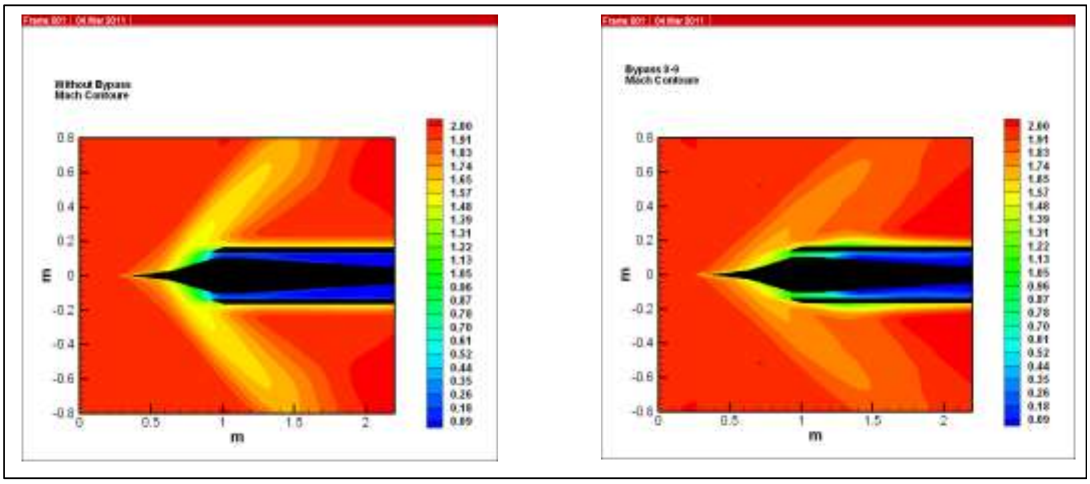
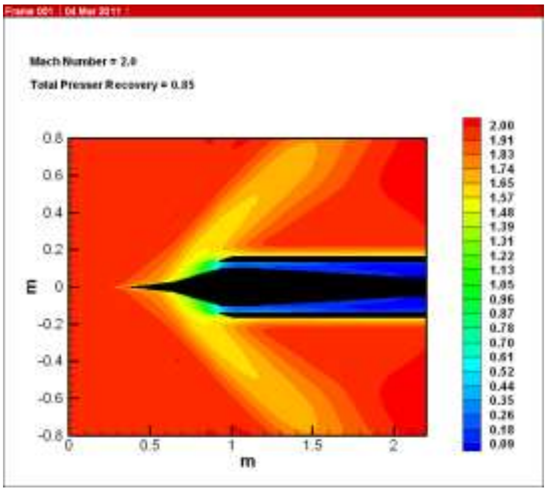
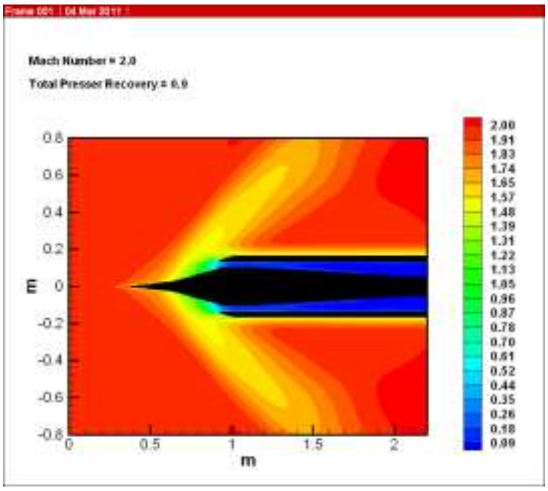


Figure (8) Mach contours for the full geometry inlet with and without bypass Technique



(a) Mach contour with total pressure ratio of 0.85



(b) Mach contour with total pressure ratio of 0.90

Figure (9) The effect of total pressure ratio on the flow geometry a & b.

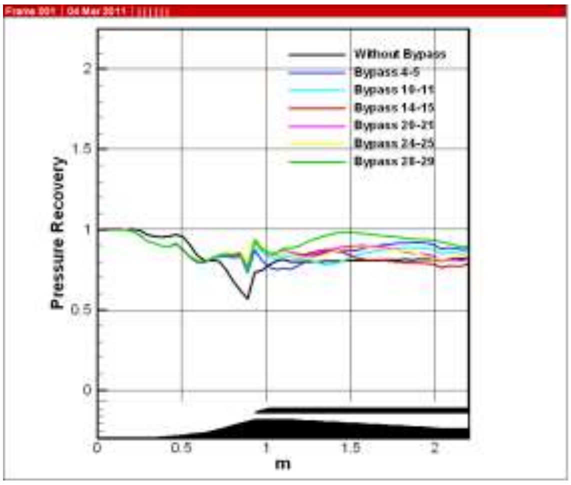


Figure (10) Pressure recovery variation along Intake axis for all cases.

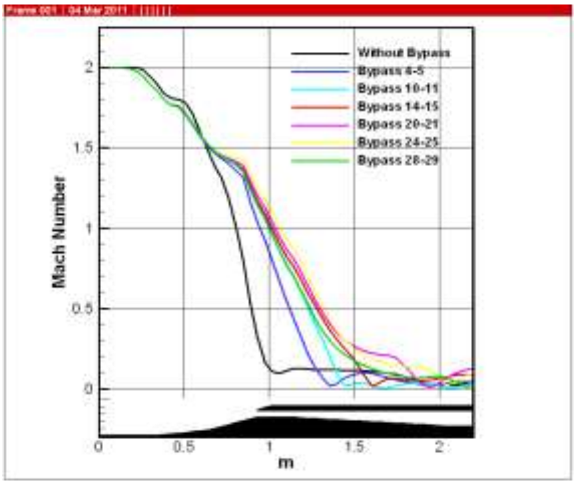


Figure (11) Mach number variation along Intake axis for all cases

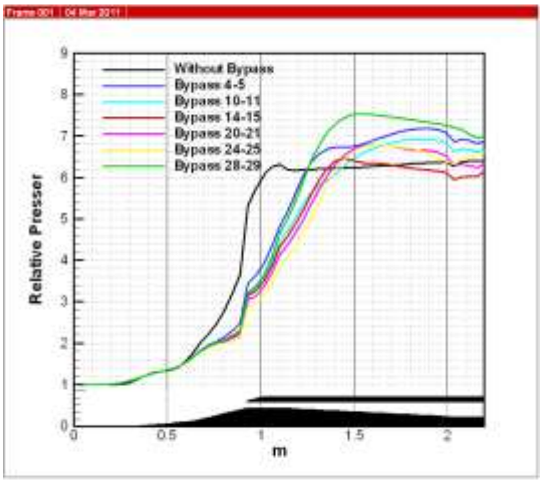


Figure (12) Relative Pressure Variation along intake for all cases.