Int. Symp. on Heat Transfer in Gas Turbine Systems
9 – 14 August, 2009, Antalya, Turkey

FRONT BULKHEAD UPSTREAM EFFECT ON INLET IGV COMPRESSOR FLOW OF MS5002B GAS TURBINE

Rabah Dizene[1*] and Djamel Chekerker[1]
[1] Mechanical Engineering Department
Advanced Mechanics Laboratory
University of USTHB, Bab Ezzouar, 16110, Algiers ALGERIA
([*] Corresponding author: rb_dizene@yahoo.fr)

ABSTRACT. The IGV compressors at stationary applications are subject to inlet total pressure distortion as well, for example, in industrial installations with poorly designed upstream bends. The consequences of a compressor include performance degradation, unsteady blade forces, vibration and very importantly, a reduction in stall margin when compared to a compressor with no distortion. Impact of exhaust air flow front bulkhead on the inlet Guide Vane of MS5002B gas turbine compressor is covered. In order to judge the degradation of aerodynamic components, we will evaluate the effect of air flow coming from the front bulkhead. Flow disturbances created at the bulkhead installed upstream the row of IGV cause cracking and change the IGV blade surface with increasing local dynamic stresses. Bulkhead caisson air flow needs to be analyzed for determining the optimal forms of its sidewall that influences the behavior of flow through the IGV. Complex phenomena of turbulent air circulating within the bulkhead chamber were found most often on gas turbines types of MS 5002B with a suction side of a right side. Dead zones and eddies moves appear and constitute obstacles to the flow.
.

INTRODUCTION

Gas turbines are widely used in industrial applications. Proper maintenance and operating practices can significantly affect the level of performance degradation and thus time between repair, and overhauls of a gas turbine. As in the case of aircraft propulsion, the inlet to power plant system must supply flow to the compressor with minimal pressure loss, distortion or unsteadiness because a significant pressure loss reduces the overall system performance, while distortion or unsteadiness effects can result in stall or surge of the compressor. High stage counts, flexible rotors, high speeds and pressures make the application prone to self-excited vibrations.
Our aim is to cover the impact of exhaust air flow front bulkhead on the inlet Guide Vane of MS5002B gas turbine compressor (see Figure 1). The IGV compressors at stationary applications are subject to inlet total pressure distortion as well, for example, in industrial installations with poorly designed upstream bends. The consequences of a compressor include performance degradation, unsteady blade forces, vibration and very importantly, a reduction in stall margin when compared to a compressor with no distortion.
In order to judge the degradation of aerodynamic components, we will evaluate the effect of air flow coming from the front bulkhead. Flow disturbances created at the bulkhead installed upstream the row of IGV cause cracking and change the IGV blade surface with increasing local dynamic stresses. Consequently, the bulkhead caisson air flow need to be analyzed for determining the

optimal forms of its sidewall that influences the behavior of flow through the IGV. This may cause changes in airfoil geometry and in airfoil surface quality and resistance. When fluid flow was deflected, fluid forces from these excitation sources have a cross coupling component which can excite swirling. Any tendency to swirl at a natural frequency generates fluid damping forces, which oppose the swirl. Instability occurs when net cross-coupling forces exceed net damping forces at the natural frequency, and high cycle fatigue (HCF) is observed in vibrations phenomena.

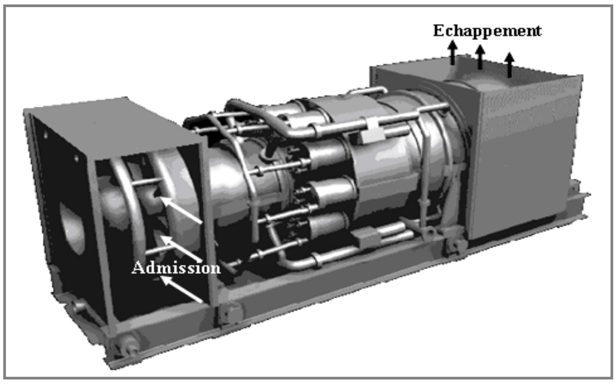


Figure1. MS5002B Gas Turbine with Lateral fluid flow suction

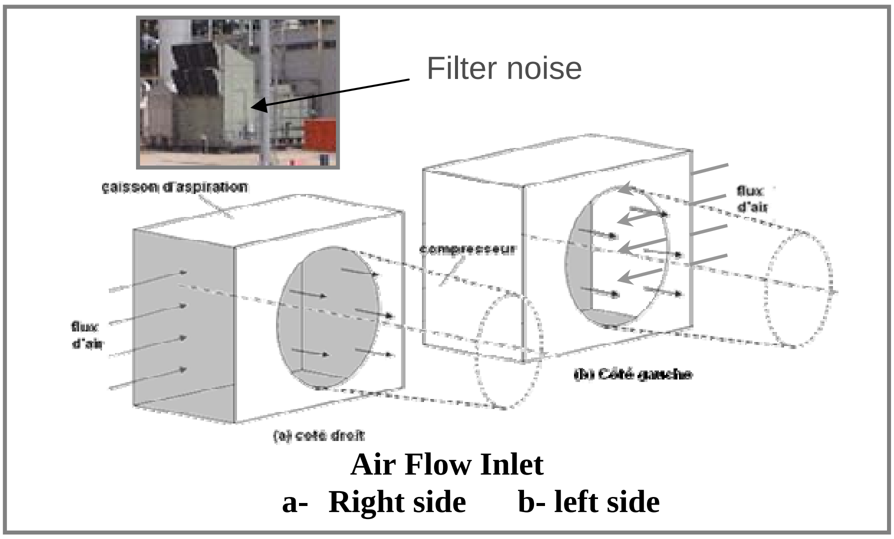


Figure2. Bulkhead geometry

The risk that can provide a bad operating IGV blades are the extensive damage caused by the detachment of one or more blades on the gas turbine in general and on the blades and fixed axial compressor. This detachment is the result of a High Cycle Fatigue (High Cycle Fatigue), which itself is caused by complex phenomena of turbulent air circulating within the bulkhead chamber. These phenomena were found most often on gas turbines types of MS 5002B with a suction side of a right side, as shown in Figure 2, which reflects the rectangular shape of the chamber and the presence of the gas which is a barrier to air flow, thus causing turbulence. As a result, dead zones and eddies moves appear and constitute obstacles to the flow.

INTAKE PARAMETER

An understanding of the process through which the energy is essential before the inlet ducts which supply compressor inlet could be designed. Intake is not different from other existing engineering system where a fraction of the supplied energy goes waste. Current aircraft and power plant designs typically consider the compressor and the inlet as separate sub-systems, where the connection between them is given as a pressure recovery and distortion specification (SAE Aerospace Committee, 1983) that must be met at the aerodynamic interface plane (AIP).
Many parameters may be considered to produce inlet distortions: high revolution speed of compressor which results in non uniform flow, bulkhead geometry and duct configuration which can develop strong separation flows. So, the compressor behavior must be considered together with the inlet.
.

Inlet Flow Deviation The inlet distortion represents a physical deformation of the air flow and can results in non axisymetric flow of the compressor and which may be translated in fluctuation of aerodynamic forces per one revolution. Inside a curved part of bulkhead, the swirl is caused by the shape of the duct itself. Swirl represents a form of energy loss, as the energy is used in accelerating this flow in the angular direction and does not contribute to IGV compressor process. Along with various distortions, the latter effects can results on strong separation flow at the first stage of rotor compressor and generate a High Cycle Fatigue

Blade Interactions In turbomachinery, there are aerodynamic interactions between blade rows of rotors and stators. In the axial compressor case, the first row used is the so-called Inlet Guide Vanes (IGV) as stationary row of blades. Many types of IGV can be used. In the MS5002B gas turbine, the IGV system achieves variable flow by means of a simple pitch angle variation. Interaction effects between rotor/stator blades can result in wake and secondary flows development. These wake regions will be minced and deflected by the adjoining blade row. This will generate locally strong velocity diminution and high swirl level, whose will be convected downstream.

PROBLEM AND PHYSICS

An incident occurred recently after several years of operation and caused a failure of some Inlet Guide Vane. This phenomenon most often has been found on the MS 5002B type gas turbines (Gas Turbines, practical manual of GE Power Systems (2002, 2004)) with a suction side on the right side. The damage caused by the separated IGV on the rotor and stator blades of the axial gas turbine compressor is considerable. Analysis of IGV row shows the existence of critical areas as shown in Figure 3. A summary observation of IGV surfaces has allowed to highlight the defects that have been recorded (figure 4), and present themselves as cracks in the head and the leading edge of the IGV.

Because of depression created by the compressor, air flows from bulkhead intake to the IGV row at compressor inlet. The airflow streamlines are divided into two parts: the first one is deflected to angle of 90 degrees by the intake bulkhead and occupies their entire volume (see Figure 5a) The second result on recirculation flow areas located at bulkhead right angles (see figure 5b). Then they come together at bulkhead exit and head to compressor IGV grid.

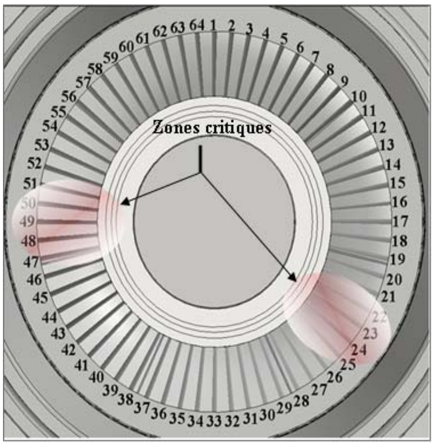


Figure3. Critical areas on IGV blade row

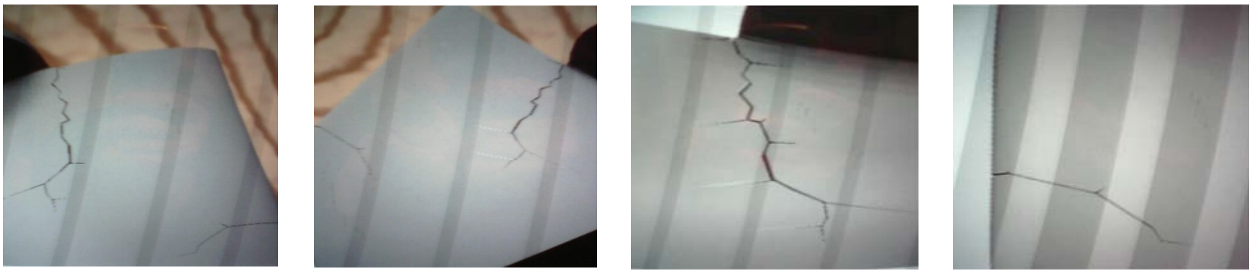


Figure4. Crack type of MS5002B IGV Compressor

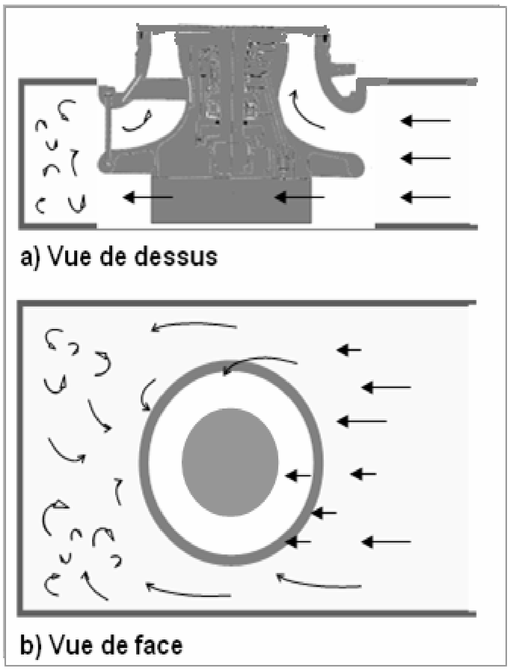


Figure5. Schematic shape of Model system (bulkhead intake and duct)
5a- up view 5b- front view

These observations are typical of a fatigue failure that occurs when the IGV metal structure has been subject to fatigue stresses or to a high fatigue cycle (HFC).The number of cycles may depend on:

The reference of material.
Material temperature.
Atmosphere (corrosive environment, for example).
Unstable flow.
The aerodynamic amplitude variations.

Liamis et al (1995) analyzed the interactions between an IGV row and transonic compressor rotor with a CFD calculation based on complete 3-D analysis. Their main result is summarized by the appearance of pressure waves caused by the rotor shocks. The spreading of waves will interact with boundary layers in the internal passage of IGV during their reflections on the IGV walls. This will cause a significant change in terms of output and speed of flow during one cycle of interaction. Currently, designers evaluate the high cycle fatigue in turbomachinery by CFD analysis with a single blades row, using high unsteady exciting due to fixed upstream / downstream flow conditions and also due to the blade rotational motion. The unsteady resulting force is then used to determine the unsteady stresses with a finite element modeling. The values of these stresses are compared to those of practical use for strain materials to the HCF. Hence, we can easy imagine that much damage failures and breaks of these machines are mainly due to poorly simulated and poorly predicted coupled rotor / stator interactions. Robert T. Johnston and Sanford Fleeter (2002) have done experimental work to study and quantify the unsteady variations flow characteristics of the IGV wakes, interacting with the flow field at the exit of a compressor rotor running at high speed. Pressure and velocity fields of IGV wakes are measured for two blade IGV passages and recorded simultaneously during a rotor transition period. The results show a very strong interaction between the IGV and the rotor blades wakes. Indeed, when the two wakes are in phase, the results show the maximum pressure losses and the high velocity deficiencies.

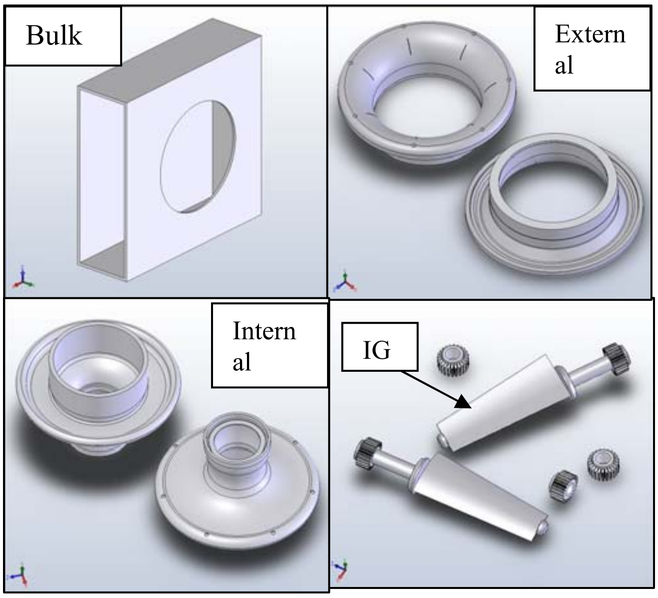
A better understanding of physical phenomena is essential for improving performance and design of modern turbomachineries. Most technical designs today are based on aerodynamic stationary analysis, while the interaction between the rows of blades of a turbomachinery generates a complex, unsteady and periodic 3-D flow. It is therefore important to assess as accurately as possible the influence of unsteady periodic phenomena, including loss generation and extend the design techniques so as to take into account these unsteadies. Therefore, unsteady numerical simulations take place as a promising tool for the design of turbomachinery efficiency, under condition, however, that low calculation costs may be possible.
The current trend of the design is to reduce the number of stages and spacing between rows of compressors and turbines blades in order to reduce weight and increasing their performance. As a result, the bringing rows will increase the aerodynamics and interactions coupling between different blade rows. To assess the problems of high cycle fatigue (HCF), analytical and numerical tools are needed to simulate the unsteady environment flow inside a row of blades for each stage. These tools must be able to model directly the coupling and interaction effects between these rows, and also be validated with appropriate experimental data before the results being introduced into the design systems. Several CFD analyses of aerodynamic turbomachinery instability were developed by using computer codes like TAM-ALE3D and Q3D Navier-Stokes equations. The use of such high speed turbomachinery analyses is beginning to yield satisfactory results.

Many difficulties are encountered in our problem modeling. We can notice, for example:
- Complex physical phenomena that contribute to the onset of cracking.

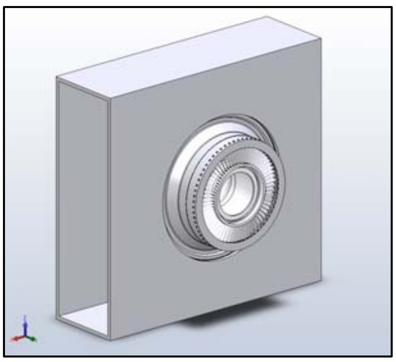
- Lot of mechanisms that interact between them (IGV / Rotor, rotor / stator and stage / stage interactions). In accordance with initial specifications, we are interested in this study to the airflow through the gas turbine bulkhead including the admission and duct bodies and IGV row. This is our model system to study in this paper.

NUMERICAL CALCULATION AND MODELLING

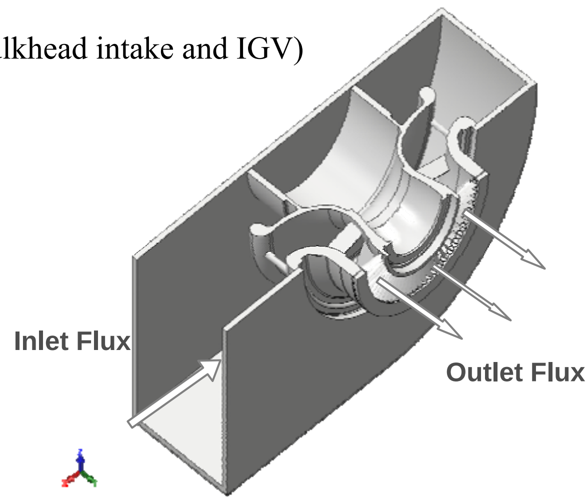
The simplifying assumptions have been adopted to alleviate the cost calculations. It begins with the generation of the geometry of the whole box, the body of admission and wheel blade IGV. The characteristics (size and scale) of the system model are identified through the diagrams and documents provided by the manufacturer. The 3-D geometric shapes reproduction and various parts assembly of the bulkhead were conducted with Solids Works software. The main elements are collected and shown in Figure 6.



6a. Main components (bulkhead intake and IGV)



6b. Model system



6c. Transverse Section of the Model system

Figure 6. MS5002B Gas Turbine Intake Model system

The characteristics of the MS5002B bulkhead have been reproduced and the model obtained has been analyzed. This model shown in figure 2 has been proven quite accurate and effective. The system model includes the Cashion plenum inlet and exit duct. SolidWorks is a 3-D mechanical design and parametric software using the Microsoft Windows graphical interface. Figure 6 shows together main elements which constitute our model system shown in Figure 6.

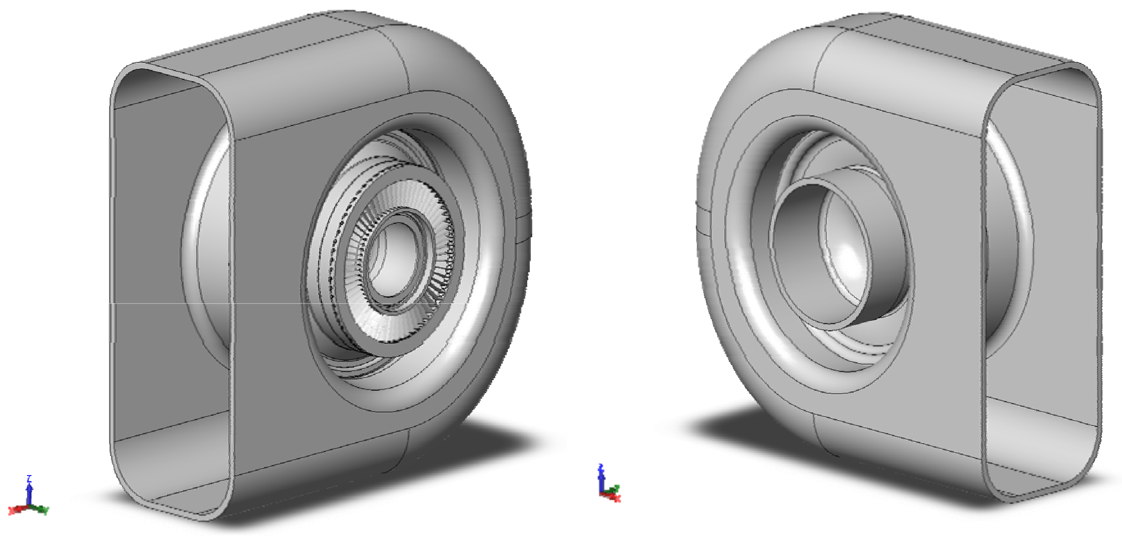


Figure 7. MS5002B Gas Turbine Intake Model system.
Configuration of solution.

GOVERNING EQUATIONS, MESHING AND BOUNDARY CONDITIONS

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_k}(\rho u_k)=0 \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t}+\frac{\partial}{\partial x_k}(\rho u_i u_k-\tau_{ik})+\frac{\partial P}{\partial x_i}=S_i \tag{2}$$

$$\frac{\partial(\rho E)}{\partial t}+\frac{\partial}{\partial x_k}((\rho E+P)u_k+q_k-\tau_{ik}u_i)=S_k u_k+Q_H \tag{3}$$

This The turbulent modeling is done using the k- ε model of two transport equations; turbulent kinetic energy and dissipation rate, written as below:

$$\frac{\partial \rho k}{\partial t}+\frac{\partial}{\partial x_k}(\rho u_k k)=\frac{\partial}{\partial x_k}\left(\left(\mu_l+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_k}\right)+S_k \tag{4}$$

$$\frac{\partial \rho \varepsilon}{\partial t}+\frac{\partial}{\partial x_k}(\rho u_k \varepsilon)=\frac{\partial}{\partial x_k}\left(\left(\mu_l+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_k}\right)+S_\varepsilon \tag{5}$$

where the source terms are:

$$S_k = \tau_{ij}^R \frac{\partial u_i}{\partial x_j} - \rho\varepsilon + \mu_t P_B \tag{6}$$

$$S_\varepsilon = C_{\varepsilon 1} \frac{\varepsilon}{k}\left(f_1 \tau_{ij}^R \frac{\partial u_i}{\partial x_j} + \mu_t C_B P_B \right) - C_{\varepsilon 2} f_2 \frac{\rho \varepsilon^2}{k} \tag{7}$$

and:

$$P_B = -\frac{g_i}{\sigma_B} \frac{1}{\rho} \frac{\partial \rho}{\partial x_i} \tag{8}$$

The constants C_μ, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k and σ_ε are defined as empirical constants. The typical values used by FloWorks are: $C_\mu = 0.09$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 1.3$, $\sigma_\varepsilon = 1$.

Calculation domain and meshing The Navier-Stokes equations are written in an unsteady state using Cartesian coordinate axes. Figure8 show the solution meshing.

The first stage of any numerical simulation problem is the generation and adaptation meshing with the geometric structure. A network of good quality is essential for the calculation procedure so that the results are acceptable.

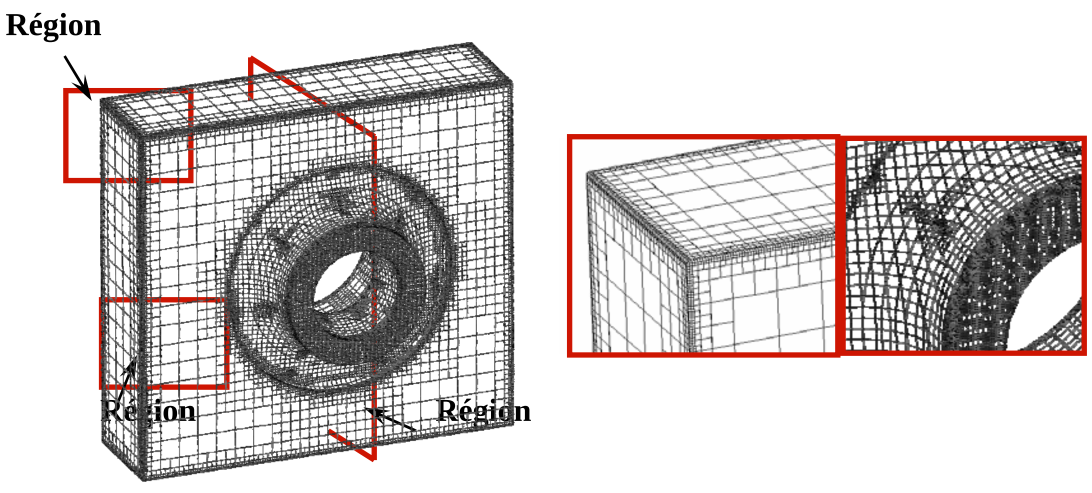


Figure8. 3-D Calculation Domain Meshing

FloWorks has a powerful automatic mesh generation. it is controlled only by three parameters:

- Level of resolution results.
- The minimum size of the fluid passage
- The minimum thickness of the solid wall.

The minimum size of the fluid passage and the minimum thickness of the solid wall allow to FloWorks solver the recognition of small cavities that has the structure with solid walls. Adjustment of these parameters affects the size characteristics of a cell. The resolution level of results governs the total number of cells produced in the mesh to get more accurate results.

The optimum values of the minimum fluid passage size and the minimum wall thickness is taken equal to 40mm and 100mm respectively. With an average level required for the resolution and obtain the required results, the total number of cells produced in a mesh is of 131,107 cells

Boundary conditions The Boundary conditions of the numerical simulation of flows in turbomachinery are complex and very difficult to define. In the grid blade calculation, the boundary conditions at upstream and downstream are specified for typical problem. At the upstream side, in addition to stagnation pressure and temperature conditions, a other condition on the incidence angle, the Mach number or the tangential absolute velocity component are specified. In this study, the boundary conditions upstream and downstream are specified in openings model system sides because of the geometric structure and objectives.

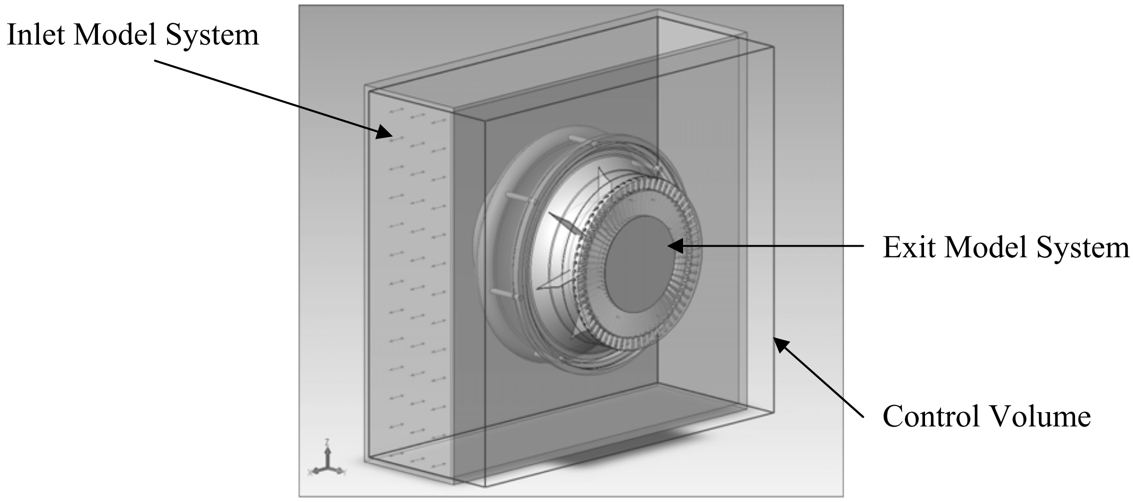


Figure 9. Inlet/Outlet boundary Conditions

Upstream boundary conditions At the Cashion plenum entrance, the following conditions are specified:

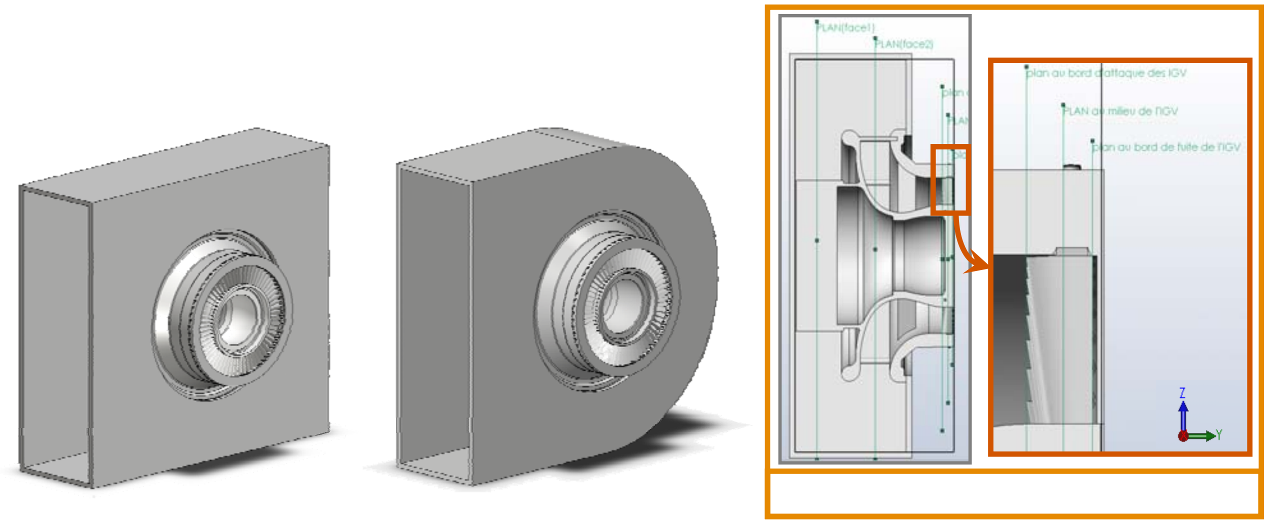
Static pressure: Ps = 1.01325 bar.
Temperature: T = 293.2 K.
Turbulence rate: 5%.
Mixture length: 14.3934 mm.

Downstream and wall boundary conditions The output and the walls boundary conditions are respectively the flow rate and the non-slip condition.

RESULTS AND DISCUSSIONS

Two bulkhead geometric configurations have been tested in order to know the effects of geometric changes made on the airflow behavior through the IGV channels. The first case (figure10.a) relates to the basic structure defined in Figure 5. The second case presents variations suggested by the manufacturer (figure 10.b). A round plate is welded instead of the right angles corner to prevent the reverse flow formation.

Result discussions are mainly focused on the comparison of the two tested configurations. Perpendicular planes to the axial direction of the bulkhead admission present visualization flow surfaces arranged in parallel showing pressure and velocity fields. This allows following the flow behavior from the fluid flow admission to blades IGV exit.



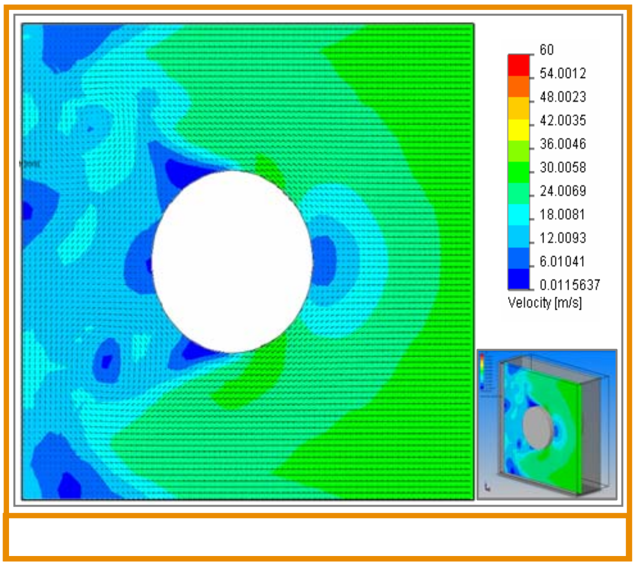
10a. Basic case **10b. Modified case** **10c. Section view of basic case**

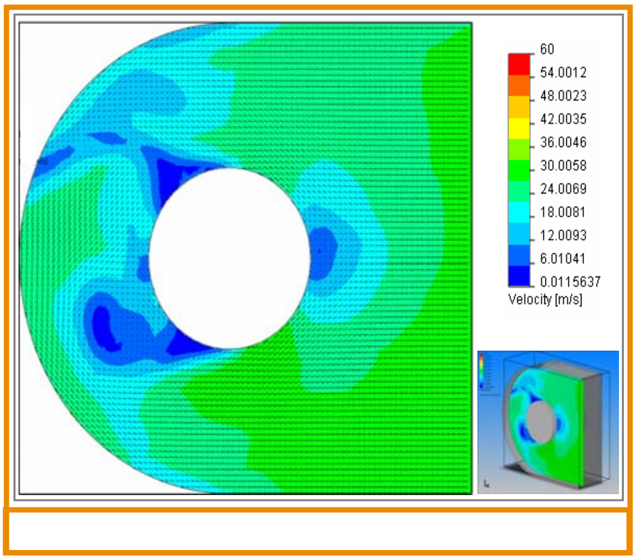
Figure 10. Model System of Bulkhead Configurations.

Figure 11 show comparisons of the velocity distributions of the two cases studied. A slight difference is seen between the two distributions. Indeed, as we approach the cylinder part, there is a different evolution. The airflow is slightly accelerated (V = 30m / s to V = 24m / s) highly towards the lower side than up the cylinder. Low velocity areas appear in the aftermath and remain attached to the cylinder, as shown in Figure 66. Flowing towards the wall downstream of the modified intake bulkhead, fluid flow undergoes an acceleration from V = 18m / s to V = 24m / s on the low side. Velocity vectors distribution seems influenced by the geometric shape. This results on the formation of two vortex zones of different sizes.

The resulting aerodynamic force exerted on the IGV is discussed. The calculation is made for each individual IGV. Thus, all the IGV are numbered to identify the every resulting force on each IGV. It has been noticed that the maximum value occurs on the IGV No. 1 and the lowest values on that No. 32 to 35 in the opposite area. This is may be due to the axial direction inlet in IGV channels. In addition to this value peak particularly on the IGV No. 1, there are two regions containing high values of the aerodynamic forces: region including the blades numbered No. 13 to 20 and 44 to 52.

These two zones could be considered as critical areas corresponding to a maximum value of 110N. The compared aerodynamic force results between the two configurations are shown on Figure 12. The major observation is the tendency of uniform distribution of efforts on the IGV blade row of the modified bulkhead. Indeed, the variation takes place around a mean value estimated at around 50N.





11a. Basic case **11b. Modified case**

Figure11. Inlet Bulkhead Velocity Distribution

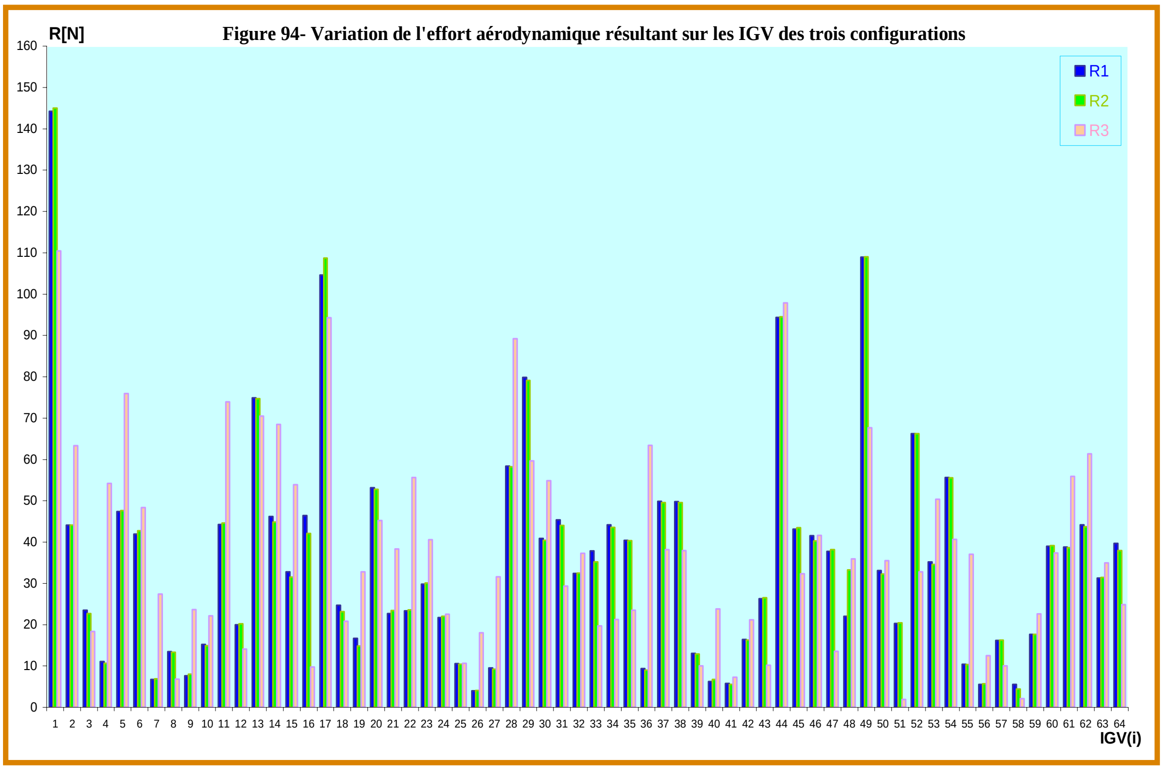
CONCLUSION

Numerical study of air flow through a bulkhead model system of axial gas turbine compressor has been made. Two configurations were analyzed. It has been found that complex mechanisms govern the flow through the bulkhead intake to IGV trailing edge. Qualitative views results were represented here by velocity fields and the resulting aerodynamic force on the IGV. The basic configuration results designated by R1 show the following:
Inside the box:

- A non-symmetrical flow structure governs around and behind the cylinder.
- Vortex structures due to the sudden change of direction form on all intake sides.

Through the IGV blades:

- The static pressure remains too general constant.
- Vortex Areas caused by the velocity field distortions tend to move towards the axial direction.



R_1. Basic case **R_2 , R_3. Modified case**

Figure12. Resulting aerodynamic force exerted on the IGV

ACKNOWLEDGEMENTS

We would like to present our thanks to Engineers and people authority of hydrocarbon and oil society, SONATRACH Company of Hassi Messaoud, Algeria. Our thanks to GE Engineers.

REFERENCES

Gas Turbines, practical manual of GE Power Systems (2002), COMK/MARK 834/II. Studio Tre Fasi –9-2002.

Gas Turbines, practical manual of GE Power Systems (2004), COMK/MARK 834/II. Studio Tre Fasi –1-2004.

Liamis, N., Bach, J. L., and Burgaud, F., (1995) "Numerical simulations of Stator-Rotor Interactions on Compressor Blade Rows," *Propulsion and energetic Panel, 85th Symposium*, CP-571, Paper 36 AGARD, 1995.

SAE Aerospace Council Division, Technical Committee S-16 (1983), "Aerospace Information Repot: inlet Total-pressure-distortion considerations for gas-Turbine engines", Tech. Rep., Society of Automotive Engineers, May 1983. AR-1419.