Proceedings of ASME Turbo Expo 2011
GT2011
June 6-10, 2011, Vancouver, British Columbia, Canada

GT2011-45589

Effects of using Hydrogen-rich syngas in Industrial gas turbines while maintaining fuel flexibility on compressor design

Pascal Nucara and Abdulnaser Sayma

Thermo-Fluid Mechanics Research Centre
University of Sussex, Brighton, BN1 9QT, U.K.

ABSTRACT
Most of the current industrial gas turbine systems are designed to operate with conventional fossil fuels. Recently, the use of Low Calorific Value (LCV) fuels gained interest, particularly, Hydrogen rich Syngas resulting from coal and solid waste gasification. When LCV fuels are used the performance and behavior of the engines could significantly change and modifications may be needed. For instance, due to the relatively low heating value the fuel mass flow rate will be much higher than natural gas, increasing substantially the mass flow through the turbine. This leads to a decrease of demand for air from the compressor, which results in increased back pressure, reduction of stall margin and possible compressor instability.
This paper presents a preliminary study to pave the way to the design of a 300 MW industrial gas turbine's compressor with the objective of operating efficiently with Hydrogen rich syngas, while maintaining the flexibility for a quick switch to natural gas in the event of gasifier failure or breakdown of feedstock supply.
NASA Rotor 37 is used as the test vehicle to provide design concepts because of its simplicity in being a single stage compressor and the availability of experimental data for the CFD model validation. Geometric modifications were performed on the rotor to shift the working line towards an estimated lower air mass flow rate working line. Further modifications were investigated in order to maintain the design point compressor efficiency primarily based on sweep and lean of the blade. Once the new working line geometry was obtained, inlet variable guide vane (IGV) effects were explored to allow the compressor to shift to the original working line without further changes to the blade shape.

NOMENCLATURE

Symbols

Air mass flow rate [kg/s]	$\dot{m}_a$
Fuel mass flow rate [kg/s]	$\dot{m}_f$
Total mass flow rate [kg/s]	$\dot{m}_{tot}$
Fuel/air ratio [-]	f
Specific heat at constant pressure [J/(kg·K)]	C_p
Combustion chamber inlet temperature [K]	T_{in}
Combustion chamber outlet temperature [K]	T_{out}
Combustion chamber inlet specific enthalpy [J/kg]	h_{in}
Combustion chamber inlet specific enthalpy [J/kg]	h_{out}
Combustion efficiency [-]	η_b
Fuel calorific value [J/kg]	Q_f
Compressor work done $[m^2/s^2]$	W
Blade rotational speed [m/s]	U
Rotor inlet absolute velocity tangential component [m/s]	$u_{t,in}$
Rotor outlet absolute velocity tangential component [m/s]	$u_{t,out}$

Abbreviations

Inlet guide vane	IGV
Integrated Gasification Combined Cycle	IGCC

Low Calorific value LCV
Variable stator vane VSV
Axial sweep AXS
Tip chord sweep TCS

INTRODUCTION

Current gas turbine technology for power and heat generation is generally optimised for natural gas. However the dramatic fluctuations in natural gas prices and concerns about security of supply, combined with requirements to reduce emissions, led to the requirement for development of reliable gas turbine technologies for alternative fuel combustion. Among others, the Integrated Gasification Combined Cycle (IGCC) is currently one of the most attractive technologies. The main incentives are low fuel costs, significant reduction in emissions and high efficiency. In future plants, the possibility of implementing pre-capture technologies for carbon dioxide is an attractive advantage.

The use of syngas in commercial gas turbines necessitates the design and development of modified or alternative reliable and stable combustion systems. For instance, due to the significantly lower heating value of the syngas compared with that one of natural gas, much more fuel is needed to reach the required turbine entry temperature. Thus the compressor has to decrease air mass flow in order to keep the turbine mass flow constant [1]. For high and constant rotational speed, this results in a strong increase of the back pressure and a significant reduction of thermal efficiency of the compressor in addition to reduction of the surge margin. If the percentage of hydrogen in the syngas is high, the compressor may be driven to complete instability and possible surge. In any case, the reduction in the surge margin would not be tolerated because of the engine operation stability requirements.

Recently, the investigation of compressor off-design performance due to the use of alternative fuels has attracted great attention. It is known that solutions to this problem include inter-stage bleed control and modified variable geometry stator schedule, which allow controlling the mass flow rate and rotor air flow incidence respectively.

Furthermore, in recent work it has been suggested that an appropriate design choice to improve the thermal efficiency when using syngas is a two-shaft engine configuration [2-3]. However, the latter solution is not considered in this paper.

The main objective of this work is to present a preliminary investigation to pave the way for the design of a multistage axial compressor for an IGCC power plant. The industrial gas turbine is being designed to operate with hydrogen rich syngas fuel which could include up to 80% Hydrogen; however it will be also required to switch occasionally to natural gas combustion.

A computational study is performed using the well known NASA Rotor 37 as test case. The CFD code was first validated using the baseline geometry and good agreement was found between experimental data from the literature and numerical predictions. The original geometry was then modified in order to shift the working line towards a lower air mass flow range related to hypothetical syngas combustion, while maintaining similar performance. The aim of the design features then would be to allow the rotor to operate with a satisfactory efficiency if the working point has to be moved back towards the original mass flow range without changing the rotor geometry. With these intents the effects of varying the stagger angle, using sweep and lean and introducing a variable inlet guide vane are here investigated and discussed. It is believed that combinations of some of these techniques could result interesting concepts for the intended multistage compressor design and hence some possible combinations are investigated.

COMPUTATIONAL MODEL

The CFD code used is the present work is based on the methodology developed by Sayma et al. [4]. The code is an edge-based solver for the RANS compressible Navier-Stokes equations and both Spallart-Allmaras and the standard k-ε turbulence models are implemented. In this study the former has been used for the turbulence closure.

The grid tool allows the generation of semi-structured meshes using a combination of structured and unstructured meshes, the former in the radial direction and the latter in the axial and tangential directions; a structured two dimensional O-grid is used to resolve the boundary layer around the blade (see Fig. 1).

The central differencing scheme used which is stabilized using a mixture of second and fourth order matrix artificial dissipation. In addition, a pressure switch, which guarantees that the scheme is total variation diminishing (TVD) and reverts back to a first-order Roe scheme in the vicinity of discontinuities, is used for numerical robustness. The resulting semi-discrete system of equations is advanced in time using a point-implicit scheme with Jacobi iterations. All the numerical simulations are started at choked conditions and then marched toward the near stall point with gradual increase of back pressure. This has been obtained by adding a variable nozzle area downstream the rotor, which allow the drawing of a whole characteristic curve by changing the throat size, hence without changing the static back pressure boundary conditions [4].

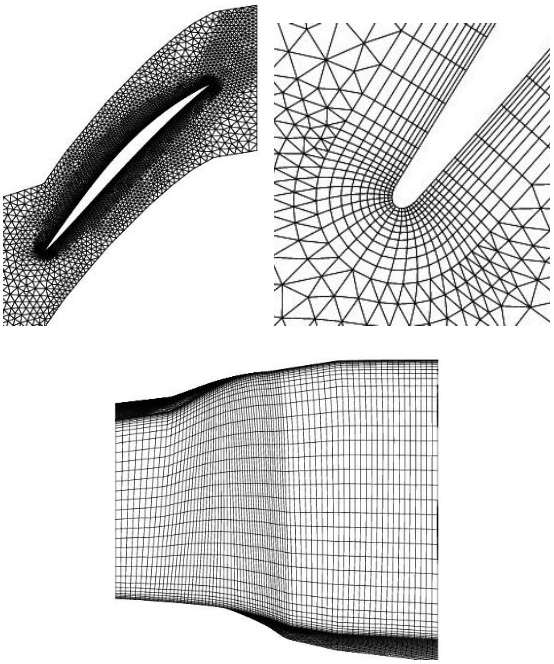


Fig.1 Computational grid. Midspan (and leading edge) view of Rotor 37(up) and radial distribution.

Between the rotor and the nozzle an outlet guide vane (OGV) is placed with the objective of removing the swirl from the flow as it enters the nozzle. The three elements are joined together in a single mesh with mixing planes at the interface boundary and steady state simulations are performed over the assembled grid.

Other boundary conditions are specified as follows: at the inlet, constant (ambient) values for both total temperature and total pressure are specified. Periodic conditions are applied in the blade-to-blade direction. Finally, near the solid walls the standard wall function is applied with slip conditions.

The convergence criterion is based on the residuals from the flow equations. Choked flow points converged in about 3000 time steps. Starting from the choked condition, once the solution is converged, the nozzle area is reduced. Calculations at lower flows were restarted from converged solutions at higher flow rates. As the stall boundary is approached the fan back pressure is increased in very small increments (very small changes in nozzle area). The number of iterations for a fully converged solution increases near stall. Cases are considered to be stalled if the residuals criterion is not satisfied after a pre-specified number of iterations (typically a maximum of 10000).

MODEL VALIDATION

In this work NASA Rotor 37 is used as test case. Rotor 37 is a low aspect ratio inlet rotor for a core compressor. It has 36 multiple circular-arc (MCA) blades and a design pressure ratio of 2.106 at a mass flow of 20.19 kg/s.

The rotor was originally designed and studied experimentally at the National Aeronautics and Space Administration Lewis Research Center (now NASA Glenn Research Center) in the late 1970's by Reid and Moore [5-6]. It was then retested in the 1990's by Suder, et al. [7]. Their data have been largely used for numerical methods validation.

Figure 2 shows the Rotor 37 meridional view; the design details are reported in table 1.

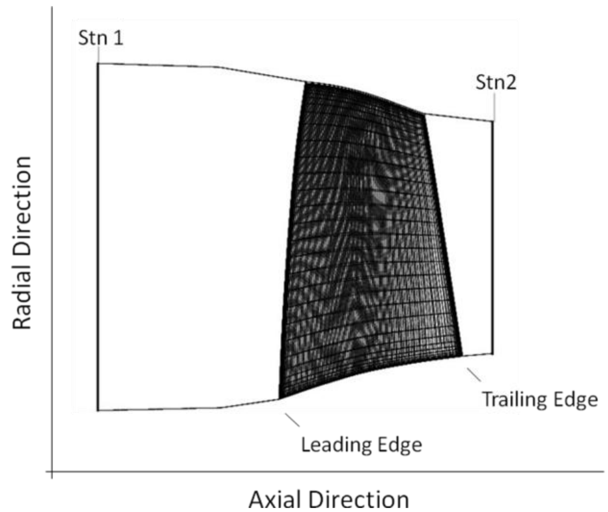


Fig.2 Rotor 37 cross section.

Table 1 Design values for NASA Rotor 37

Number of rotor blades	36
Tip solidity	1.288
Rotor inlet hub-to-tip diameter ratio	0.7
Rotor blade aspect ratio	1.19
Rotor tip relative inlet Mach number	1.48
Rotor hub relative inlet Mach number	1.13
Design tip clearance (mm)	0.356
Choking mass flow rate (Kg/s)	20.93
Design wheel speed (rad/s)	1800
Tip speed (m/s)	454.136
Reference temperature (k)	288.15
Reference pressure (KPa)	101.33
Rotor total pressure ratio	2.106
Rotor polytropic efficiency	0.889

The flow field at the design speed is mainly examined in detail at two different operating conditions: near peak efficiency and near stall, corresponding at 98% and 93% of chock mass flow respectively. Although improvements have been made to numerical modeling techniques over the past years, some of the important aspects of the flow field have not been fully explained with numerical studies based on RANS. For instance, with regards to the near peak efficiency condition it has been observed that some details of the measurements proved to be especially difficult to predict [8]:

- The total pressure ratio is usually overestimated while adiabatic efficiency is under predicted.
- The total pressure distribution downstream of the rotor differs from the experimental data below the 40 percent span. In this region the experiments show a *pressure deficit* (see Figure 4) that has not been generally well reproduced by numerical codes. The low-pressure region has prompted much discussion in literature. Even if this phenomenon is not yet fully understood, it has been mostly related to the hub corner stall, which can be explained as follows: due to high aerodynamic loading of Rotor 37, the passage shock extends along all the blade; in the hub region the interaction of the passage shock with the boundary layer causes the three-dimensional corner separation, which leads to the pressure deficit.
- Most codes calculated a higher total temperature ratio downstream of the rotor in the region close to the tip.

Comparisons between experimental data and numerical predictions obtained in this work are reported below.
The compressor performance characteristics are displayed in Figure 3.

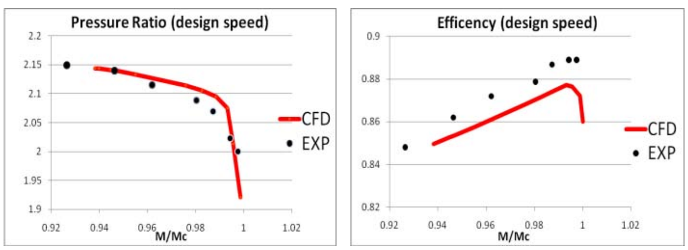


Fig.3 Pressure ratio and efficiency maps.

The computed and experimental mass flow rates are normalized using the corresponding choked mass flow. Compared with the measured value of 20.93 kg/s, the numerical simulation predicted a lower value of the choked mass flow, of about 20.80 kg/s. It can be noticed how the numerical prediction slightly overestimates the pressure ratio and under predicts the efficiency. As discussed, these discrepancies can be observed in other reports [8-11].
The spanwise distributions of total pressure ratio and total temperature ratio at the passage exit (station 2 in Figure 2) are shown in Figure 4 for the near peak efficiency condition.
With regards to pressure ratio, the numerical results agree fairly well with the data measurements, except near the tip. Most important is that the code predicts a reliable pressure deficit near the hub.
Furthermore an accurate CFD prediction of the total temperature radial distribution can be observed. The numerical results differ from the data only above of 90% span, as found out by most of codes. However it is believed that this discrepancy could be also due to some experimental uncertainties in this tip gap region, since one would expect a grater temperature gradient to the wall.

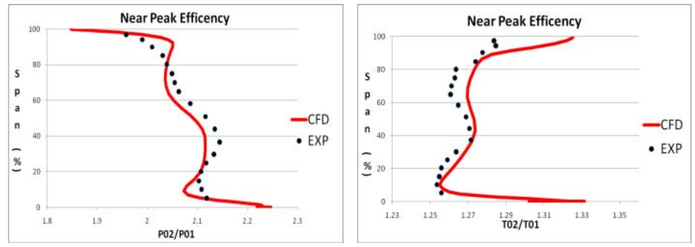


Fig.4 Total pressure ratio and total temperature ratio distributions downstream of the rotor, at near efficiency operating condition.

The spanwise distributions of total pressure ratio and total temperature ratio downstream the rotor at near stall conditions are shown in Figure 5.

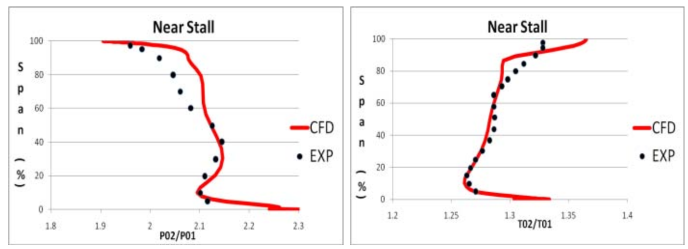


Fig.5 Total pressure ratio and total temperature ratio distributions downstream of the rotor, at near stall operating condition.

Between the hub and the midspan the CFD prediction of the total pressure ratio agrees well with the measurements. Above the midspan the computed pressure ratio is higher than the data but the shape remains reliable.
As for the near peak efficiency condition, the numerical results relative to the temperature ratio at near stall condition match well with the experimental data. A small discrepancy can be found in the tip region.
Finally the blade to blade relative Mach number at 95% span at both near peak efficiency (high flow) and near stall (low flow) conditions is compared with measured contours.
It can be observed (Figure 6) that downstream of the interaction between the tip clearance vortex and the passage shock a region of low relative Mach number exists as a result of the effective reduction in the flow area (blockage). When increasing the back pressure the midpitch flow field becomes more distorted due to the strengthening of the interaction. Further, it is clear that at low flow condition the passage shock (and therefore the blockage) becomes more severe and moves upstream. As stall boundary is approached, the shock in the front of the blade is expelled. Downstream the shock the flow is slightly subsonic, thus it rapidly accelerates on the suction

surface until it encounters the shock from the adjacent blade [11].

The computed relative Mach number contours are shown in Figure 6. A fairly good qualitative agreement with the results reported in [11] was observed.

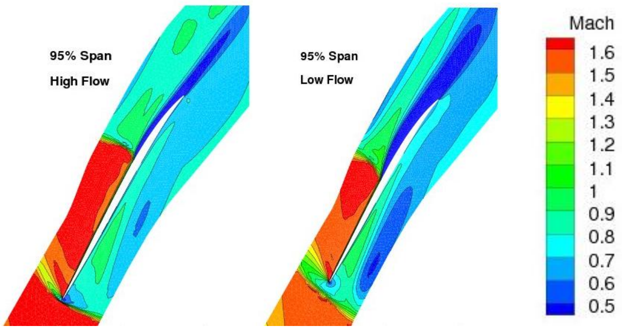


Fig.6 Contour of calculated relative Mach number along the 95% span stream surface for the rotor operating at design speed for high flow and low flow conditions**.**

GEOMETRY MODIFICATIONS

In this section, rotor geometry modifications which affect the mass flow range and performance of the original rotor are investigated. The analysis focuses on the effects produced by blade modifications of stagger, sweep and lean and introduction of IGV. Obviously the influence of bleeding can't be investigated on a single stage, but it will be fundamental for controlling the off-design performance of the multistage industrial gas turbine's axial compressor.

The baseline rotor is redesigned with the aim of operating at a certain lower mass flow range as a result of using a hydrogen rich syngas fuel. The objective is then to show the possibility of providing acceptable efficiency if the compressor has to work at the original conditions.

The syngas combustion is estimated as follows:

The fuel/air ratio for a generic natural gas is calculated by applying the enthalpy balance of the combustion process:

$$\dot{m}_a h_{in} + \dot{m}_f \cdot (h_f + \eta_b \cdot Q_f) = (\dot{m}_a + \dot{m}_f) \cdot h_{out} \qquad (1)$$

The following values are assumed:

$$Q_f = 36\ MJ/kg$$
$$\eta_b = 0.95$$
$$T_{out} = 1500\ K$$
$$C_p = \cos t = 1004.5\ (J/K)/kg$$

Considering the near peak efficiency condition ($\dot{m}_a \cong 20.5\ kg/s,\ T_{in} = 350\ K$) from equation (1) the fuel/air ratio and the fuel mass flow and total mass flow rates for a natural gas combustion can be estimated:

$$f = \frac{\dot{m}_f}{\dot{m}_a} \cong 0.034$$
$$\dot{m}_f \cong 0.7\ kg/s$$
$$\dot{m}_{tot} \cong 21.2\ kg/s$$

Assuming then a syngas calorific value of $Q_f = 12\ MJ/kg$, the new fuel/air ratio can be found from equation (1). Thus, keeping the same total mass flow rate of the natural gas case, the near peak efficiency air and fuel mass flow rates relative to the syngas combustion are evaluated:

$$f \cong 0.1$$
$$\dot{m}_a \cong 19.4\ kg/s$$
$$\dot{m}_f \cong 1.8\ kg/s$$

Thus, a reduction of the air mass flow range of about 5% has been estimated.

Stagger Angle Effects

Ramakrishna et al. [12] reported that when an axial rotor operating at a certain conditions is restaggered, the change in passage area results in changing the mass flow rate, at which the flow incidence is the same. When closing the rotor passage (positive stagger) the characteristic curves shift to the left. Moreover it was observed that the effect on rotor performance was to reduce the losses while maintaining similar (or slightly higher) pressure ratio. These effects are mainly related to the reduction of flow velocity relative to the blade.

In order to shift the characteristic curves towards a lower mass flow range, in this study the original Rotor 37 blade was positively staggered. The increments of stagger angle were approximated considering the change in inflow angles in circumferential direction, due to the axial velocity reduction when the estimated syngas combustion occurs.

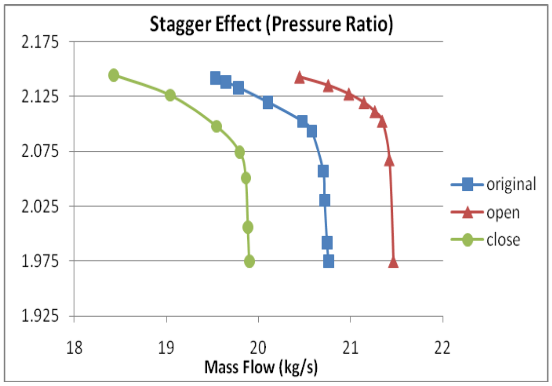
For a more complete analysis the negative stagger angle effects were also investigated. The variation of stagger angles linearly varies from hub to hip (see table 2).

Table 2 Radial stagger variation

Span (%)	Positive increment (deg)	Negative increment (deg)
0 (hub)	1.972	- 1.972
20	1.878	- 1.878
40	1.789	- 1.789

60	1.705	- 1.705
80	1.6267	- 1.6267
100 (tip)	1.5495	- 1.5495

From the results (see Figure 7), it can be observed that the positive stagger approximation is nearly suitable for the required syngas conditions. In fact the near peak efficiency operating point relative to the closed blade corresponds to a mass flow rate of about 19.5 (kg/s).
As expected, with regards to performance, the results show that the main effect of increasing the stagger angle on rotor performance is to increase the efficiency (reduction of losses) without significantly affecting the pressure ratio. In addition, due to the flow velocity reduction when closing the blade, a small positive influence on the stall margin can be noticed.



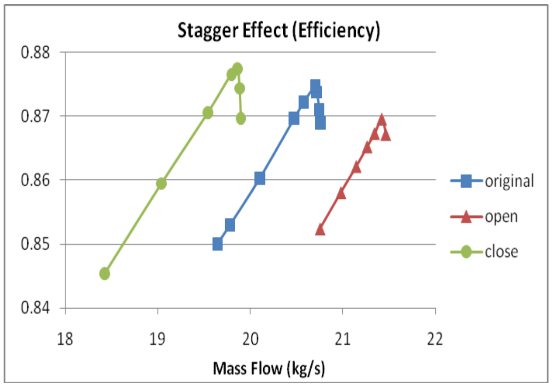


Fig.7 Stagger angle effects on Rotor 37 performance.

Sweep and Lean Effects

The introduction of transonic axial compressors has led to the development of new blade design techniques, such as sweep and lean, with the aim of improving the aerodynamic behavior of the three dimensional flow. In literature two main definitions of sweep and lean can be found. The common choice is to define sweep and lean as the movements of blade sections respectively in the axial (axial sweep, AXS) and tangential direction. In the other case, sweep and lean are defined to be moving the blade sections along the chord line (tip chord line 'true' sweep, TCS) and perpendicular to the chord line respectively. In this paper the former definition has been used.
The sweep is considered to be forward if the airfoil sections close to endwall result curved upstream. The lean is taken to be positive if the airfoil sections close to the endwall are tangentially skewed towards the direction of rotation.
Recently the sweep and lean techniques have been largely investigated and have gained wide use. Despite that, there is still some controversy about their effects in axial compressor design.
However, it seems to be generally agreed that the use of forward sweep has a remarkably positive influence on the stall margin. One explanation of the improvement has been attributed to the reduction of blade loading in the front area at the tip region. This results in a decreased tip leakage flow which is believed to be one of the main causes for stall inception [13-14]. Another important reason was attributed to a more uniform radial distribution of aerodynamic loading and the consequent less accumulation of the blade surface boundary layer at the tip [15-16]. Controversy is found about the effects of forward sweep on compressor performance.
On the other hand it seems that the use of positive lean could significantly improve the efficiency as a consequence of curving downstream the passage shock front, thus reducing the shock strength [13][18].
In a recent work, Benini et al. [18] investigated the effect of axial sweep and tangential lean on Rotor 37. Their results show that both forward and backward swept rotors slightly improve the original efficiency and substantial higher choking mass flow was obtained with a specific backwards blade. High increment of the overall efficiency was instead observed by applying a positive tangential lean, which drastically modified the shock structure within the blade passage. On the other hand, detrimental results were obtained with a negative lean.
Jang et al. [19] described an optimization procedure of blade sweep of Rotor 37. The adiabatic efficiency was selected as the objective function. Using the tip chord line sweep definition the optimization was performed considering backward swept blades. It was found that the optimum shape improves efficiency mainly in the middle of the span by moving downstream the shock on the suction surface.
In this paper two generic axially swept and two generic tangentially leaned blades were analysed. In doing this no optimization was performed and no attempt was made to improve the design for the new flow behavior. Furthermore mechanical constraints were not considered. The main purpose was to examine the general trend of sweep and lean aerodynamic effects.
Following the work done by Benini et al. [18] it has been decided not to change the original relative position between the tip section and the endwall.
Figure 8 shows the modification of the original stacking line, obtained by moving six control points located at 20%, 40%,

60%, 80% and 100% of the span. Figure 9 shows the related new rotor geometries obtained.

The results obtained with swept blades are reported in Figure 10. It can be noticed that compared with the original geometry, the forward swept blade improved the stall margin and resulted in a lower choking mass flow rate. No significant effects on compressor performance have been observed.

On the other hand the backward swept rotor negatively affected the surge margin but provided a higher chocking mass flow rate and a small increase of the efficiency was obtained.

The characteristic curves obtained with tangentially leaned rotors are compared in Figure 11. Both rotors produced a choking mass flow rate similar to the original one. The positive leaned rotor achieves an improvement in the efficiency without affecting the pressure ratio. Furthermore a slight gain in stall margin has been observed. On the other hand the negative leaned rotor globally deteriorated the baseline performance.

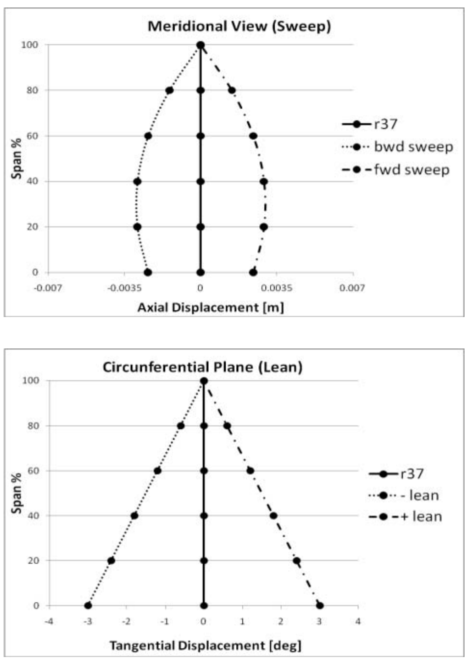


Fig.8 Stacking lines modifications.

Fig.9 Meridional (up) and front views of modified rotors.

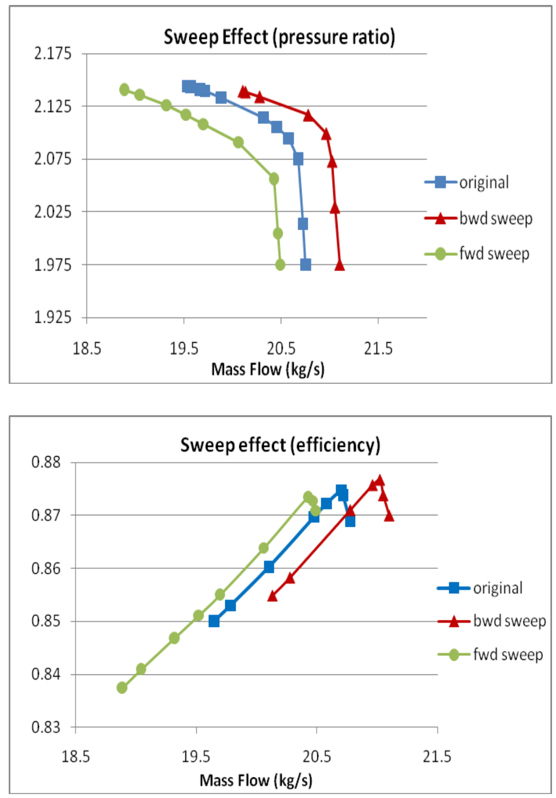
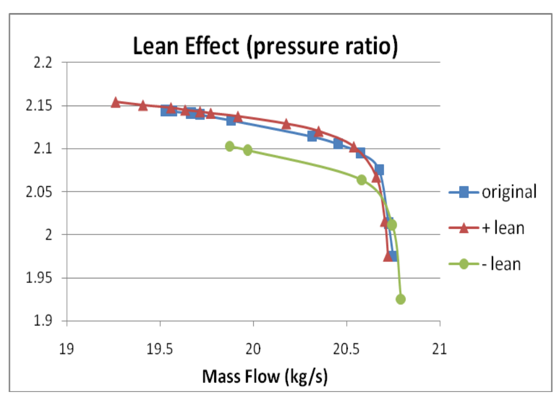


Fig.10 Sweep effects on Rotor 37 performance.



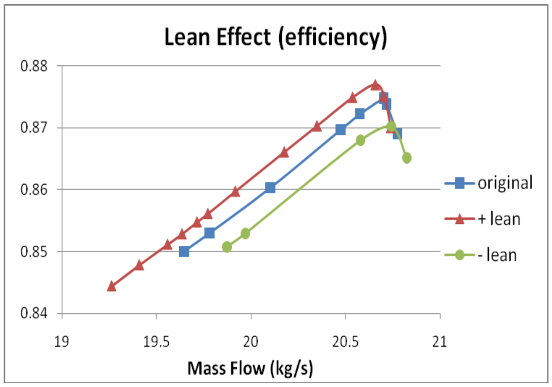


Fig.11 Lean effects on Rotor 37 performance

INLET GUIDE VANE

It is known that at off design condition, axial compressor's inlet guide vane and variable stator vanes are mainly used to redirect the air flow towards the relative downstream rotors in order to keep the incidence angle within an acceptable range. Thus, Variable Stator Vans (VSV's) play the main role for reducing the losses which always occur at off design conditions.

In transonic compressor's early stages IGV's and VSV's can be also used to modify the shock structure and position along the rotor blade passage by reducing the inlet relative velocity, so that an improvement in efficiency can be obtained.

However this method reduces the work done by the compressor $(W = U(u_{t,out} - u_{t,in}))$ since the incoming flow acquires (or increases) a tangential velocity component $u_{t,in}$ (see Figure 12).

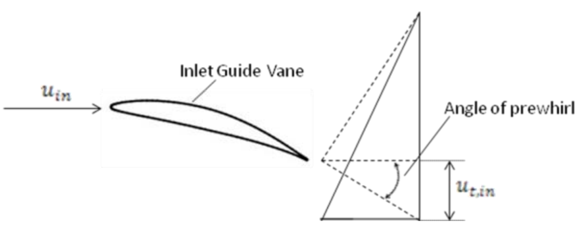
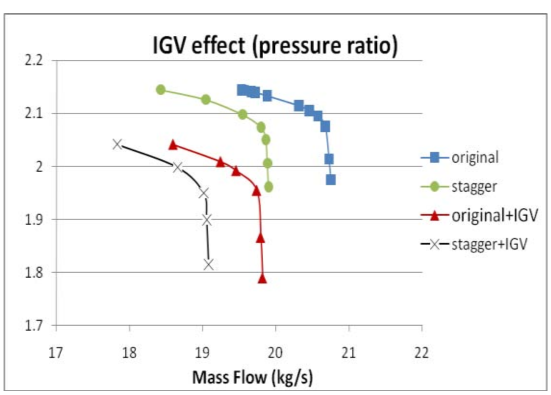


Fig.12 Velocity triangle modification at inlet



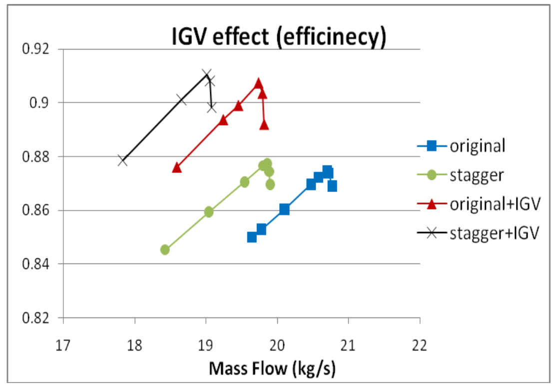


Fig.13 IGV effects on Rotor 37 performance

Figure 13 shows the compressor maps obtained by introducing an IGV to Rotor 37. The IGV blade angle was chosen in order to deflect the incoming axial flow by about 10 degrees. This allowed reducing the mass flow range towards the value estimated as a result of using syngas. The same IGV was then applied to the positively staggered rotor.

As expected the results show a reduction of pressure ratio (about 5%) and a substantial increase of efficiency (about 4%).

With regards to the purposes of this work, from Figure 13 a basic possible design would be to accept a reduction of work done in favor of a high efficiency, for the hypothetical syngas combustion. Then opening the IGV would allow shifting the working line towards the original mass flow rate and performance.

POSSIBLE CONFIGUARTIONS

It is believed that by combining some of the rotor geometry modifications discussed it would be possible to improve the operating conditions of a compressor required to work with syngas while maintain fuel flexibility.

In this work a simple solution has been previously proposed in Figure 13. However a further possible configuration was investigated. Firstly, the combination of positive lean and positive stagger was considered, recalling that both modifications showed a small improvement in efficiency and stall margin. The previously investigated leaned blade was used. The stagger angle was chosen instead in order to obtain a lower choking mass flow rate which would be close to the original one using a variable inlet guide vane during operation. The results are reported in Figure 14. As expected, compared with the original geometry, the modified rotor produced slightly higher efficiency and better stall margin.

The IGV has been then introduced in order to shift the working line toward the mass flow rate estimated as a result of using syngas. However this time, compared with Figure 13, a lower IGV stagger angle is needed, which implies a lower reduction of pressure ratio (Figure 15). It is important to note that by opening the IGV is then possible to operate near the original hypothetical design point (mass flow rate around 20.5 kg/s) with an acceptable efficiency reduction, of about 1.5%. The loss in pressure rise can be compensated by the addition of further stages if required.

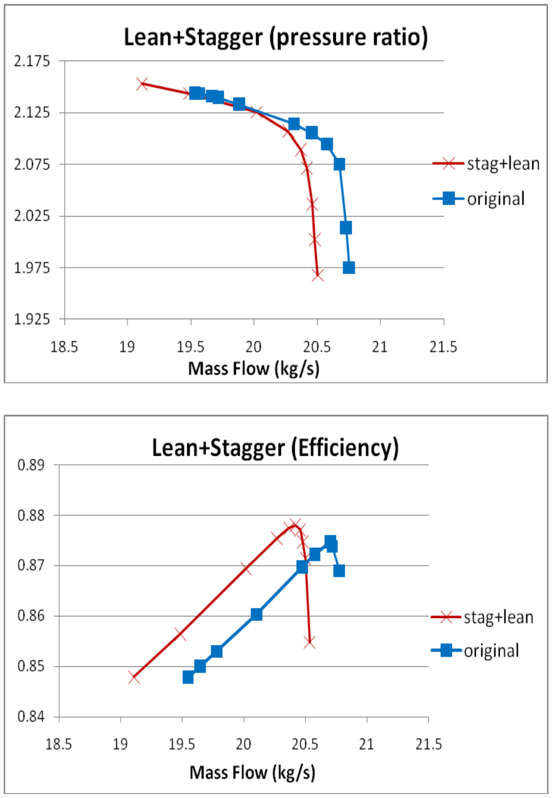


Fig.14 Combination of positive lean and positive stagger effects on Rotor 37 performance.

Thus, it has been demonstrated that a trade off exists between the design pressure ratio for the hypothetical syngas combustion and the off-design efficiency when the natural gas combustion is required. Obviously the discussed configuration can be slightly varied depending on the design requirements.

Another attempt could be to consider the backward swept blade and positive stagger. The idea is to combine the higher mass flow rate obtained with the backward blade and the positive effect on the stall margin produced by staggering the rotor, again recalling that both modifications showed an improvement in efficiency. It is believed that by varying the contribution of sweep and stagger and then introducing an IGV, characteristic curves similar to those of Figure 15 can be produced.

However, with an optimization process (which would largely alleviate the traditional trial and error CFD approach) it would be very interesting to investigate in depth all the possible configurations, by modifying the quality of lean and sweep and their combination with the IGV and stagger angle variations. However, in this study, the main objective was exploring trends that can be used in a multistage compressor design rather than producing optimum design for Rotor 37.

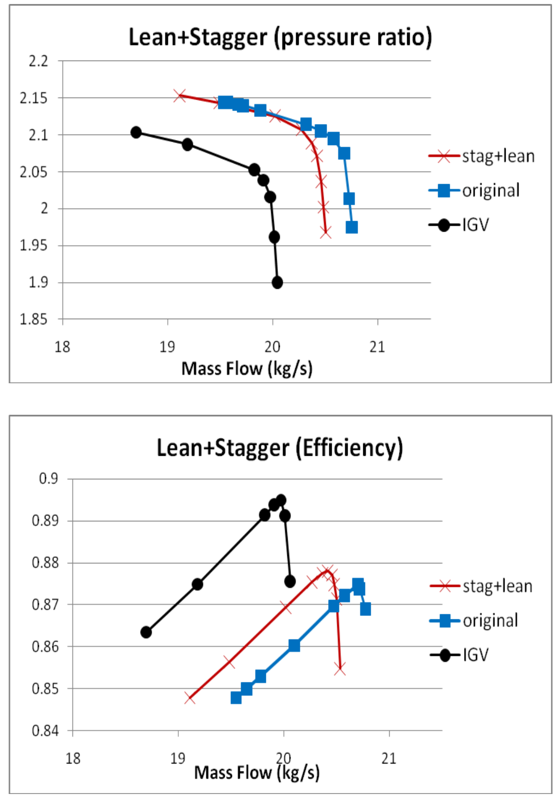


Fig.15 Introduction of IGV to the lean and stagger combination.

CONCLUSIONS

This work is a preparatory stage in the process of designing a 300 MW industrial gas turbine's compressor, the main fuel in the gas turbine is Hydrogen rich syngas. However, a main requirement is to maintain fuel flexibility allowing the gas turbine to switch back to natural gas whenever needed. The research aimed at exploring combinations of design modifications that can be used in the design of the multi-stage compressor which were tested using CFD. The CFD model was firstly validated using Rotor 37 as test case. The same compressor was then used as a vehicle for the purpose of this work. The effects of changing the blade shape by introducing sweep and lean were investigated. Other modifies, such as variation of blade stagger angle and introduction of IGV were also discussed. Inter-stage bleed control is a further tool that could not be investigated in this a single stage compressor, but it has been mentioned that such technique will be fundamental for the specific multistage axial compressor design.

Ad-hoc modifications were performed with each modification individually investigated with the main objectives of

identifying trends rather than producing optimum design. This was followed by investigating combinations of these modifications.
Staggering the blade by closing can lead to the required reduction in mass flow rate with a small increase in efficiency, and almost no change in pressure rise.
Forward swept blade improved the stall margin and resulted in a lower choking mass flow rate with no significant effect on compressor efficiency. Meanwhile, backward swept rotor negatively affected the surge margin but provided a higher chocking mass flow rate and a small increase of the efficiency.
The positive leaned rotor achieved an improvement in the efficiency without affecting the pressure ratio and a slight gain in stall margin. On the other hand the negative leaned rotor globally deteriorated the baseline performance.
The introduction of an IGV reduces the mass flow rate and pressure ratio and as expected, increases the efficiency.
While most of these effects are well documented in the literature, the interesting results of this study is the introduction of combinations what will serve the purpose of syngas operation while maintain fuel flexibility. A particular configuration, which considered the combination of positive lean and stagger with the introduction of an IGV, was then studied. For the test case considered the results showed a reliable rotor design solution for the achievement of fuel flexibility. However, substantial improvements could be obtained by applying optimization procedures.

ACKNOWLEDGMENTS

This work was funded by the European Framwork 7 as part of the project H2-IGCC which is coordinated by the European Turbine Network.

REFERENCES

[1] Christer Björkqvist, European Turbine Network (October 2008), “Low emission gas turbine technology for hydrogen-rich syngas”, Collaborative Project, FP7-ENERGY-2008-TREN-1, ENERGY 2008.6.1.4: Advanced Gas Turbines for Solid Fuel Gasification Processes.

[2] Eriksson P., Genrup M., Jonshagen K., Klingmann J., (2009), “Off-design performance investigation of a low calorific gas fired two-shaft gas turbine”, ASME Turbo Expo 2009, Paper GT2009-59067.

[3] Ferreira, S. B. and R. do Nascimento M. A., (2007), “Gas turbine engine off-design performance simulation using syngas from biomass derived gas as fuel” , ASME Turbo Expo 2007, Paper GT2007-27804.

[4] Sayma, A. I., (2009), “Towards virtual testing of compression systems in gas turbine engines”.

[5] Reid, L. and Moore, R. D., “Design and overall performance of four highly-loaded, high speed inlet stages for an advanced, high pressure ratio core compressor,” NASA TP-1337, 1978.

[6] Reid, L. and Moore, R. D., “Experimental study of low aspect ratio compressor blading,” ASME Paper 80-GT-6, 1980.

[7] Suder, K. L., and Celestina, M. L., “Experimental and computational investigation of the tip clearance flow in a transonic axial compressor rotor,” NASA TM-106711, 1994.

[8] Hah C.,, “Large eddy simulation of transonic flow field in NASA Rotor 37”. NASA/TM-215627, 2009.

[9] Chima R.V., “SWIFT code assessment for two similar transonic compressors”, NASA/TM-215520, 2009.

[10] Teixeira, J. A., Ivey, P. C., Behesthi B.H. and Ghorbanian K., “Parametric study of tip clearance casing treatment on performance and stability of a transonic axial compressor”, Journal of Turbomachinery, October 2004, Vol. 126, Issue 4, pp 527-536.

[11] Suder, K. L., and Suder, K. L. (1997),”Blockage development in a transonic, axial compressor rotor”, NASA Technical Memorandum 113115.

[12] Ramakrishna, P. V., and Govardhan, M. (2010), “Numerical study of the stagger angle effects in forward swept axial compressor rotor passages”, ASME Turbo Expo 2010, Paper GT2010-23160.

[13] Denton, J. D., & Xu, L. (2002), ”The effects of lean and sweep on transonic fan performance”, ASME Turbo Expo 2002, Paper GT2002-30327.

[14] Bergner, J., Kablitz, S., Hennecke, D. K., Passrucker, H., and Steinhardt, E. (2005), “Influence of Sweep on the 3D Shock Structure in an Axial Transonic Compressor”, ASME Turbo Expo 2005, Paper GT2005-68835.

[15] Wadia,A.R. Szucs,P.N., and Crall,D.W., 1998, “Inner workings of aerodynamic sweep”, ASME Journal of Turbomachinery, Vol.120, pp.671-682.

[16] Yamaguchi N, Tominaga T, Hattori S and Mitsubishi T., “Secondary-loss reduction by forward-skewing of axial compressor rotor blading”, In: Proceedings of 1991 Yokohama international gas turbine congress, vol. 2, 1991.

[17] Ji, L., Chen, J., and Lin, F. (2005), "Review and understanding on sweep in axial compressor design", ASME Turbo Expo 2005, Paper GT2005-68473.

[18] Biollo, R., and Benini, E. (2008), "Aerodynamics behaviour of a novel three-dimensional shaped transonic compressor rotor blade", ASME Turbo Expo 2008, Paper GT2008-51397.

[19] Jang, C., Li, P., and Kim, K. (2005), "Optimization of blade sweep in a transonic axial compressor rotor", JSME International Journal Series *B*, Vol. *48* (No. 4), pp 793-801.