CFD Analysis of Inlet Flow Distortions on an Axial Fan

Tiziano Ghisu
Politecnico di Torino
Via Taranto 12a
09170 Oristano, Italy
Tel. 0039 0783 357071
Mobile 0039 347 9697259
e-mail: tizianoghisu@hotmail.com

ABSTRACT

The usual approach to compressor design considers uniform inlet flow characteristics. Especially in aircraft applications, the inlet flow is quite often non uniform, and this can result in severe performance degradation. The magnitude of this phenomenon is amplified in military engines due to the complexity of inlet duct configurations and the extreme flight conditions. CFD simulation is an innovative and powerful tool for studying inlet distortions and can bring this inside the very early phases of the design process.

This project attempts to study the effects of inlet flow distortions in an axial flow compressor trying to minimize the use of computer resources and computational time. The first stage of a low bypass ratio compressor has been analyzed and its clean and distorted performance compared outlining the principal changes due to uneven flow distribution: drop in mass flow, increase in pressure and temperature ratios, decrease in surge margin. The attention has then been focused on the effect of the level of the distortion on the stage characteristics.

NOMENCLATURE

P	pressure
PR	pressure ratio
Q	non-dimensional mass flow $\left[kg \cdot \sqrt{K} \cdot kPa^{-1} m^{-3}\right]$
T	temperature
Theta	angle of rotation
TR	temperature ratio
W	mass flow

INTRODUCTION

One of the main assumptions made in the design of a compressor or a fan is the inlet flow uniformity. Even if one of the principal aims and design rules of the engine intake is the generation of the best possible conditions at the fan inlet, in real engine operations the inlet flow is quite often distorted, and the extent of this non-uniformity is strictly related to the particular application of the engine and to its geometry. Bigger distortions are usually expected in military applications and in supersonic flight.

The main effect of inlet distortion is felt by the compressor (particularly the low pressure compressor or fan). The main problems on this component are loss in mass flow, efficiency, surge margin and the origin of stability problems due to the variation of flow properties with time and space [1].

The most common type of distortion is the total pressure distortion and quite a lot of effort has been dedicated through the years to the study of its causes and effects. This is the principal reason why this is also the type of distortion chosen to deal with in this study.

The use of CFD for studying distortion can give an important save in terms of costs and time, bringing this study into the very early phases of the design process. CFD offers the distinct advantage that it can be performed at any level of the engine development process.

Since the first three dimensional simulation of an isolated rotor (NASA rotor 67) the use of CFD for complex turbomachinery flows studies has increased exponentially and nowadays examples of successful simulations with undistorted condition are quite common also for complex geometries such as transonic fans ([2] and [3]).

An excellent example is given by [4] who managed to perform a complete simulation of the whole rotor under distorted conditions by linking together 162 processors for a total main memory of 41 GB. The strategy adopted by other studies has been to reduce the number of blade passages in the simulation, in order to make the study acceptable in relation to the computer capabilities. A further example is given by [5] who studied the effects of steady and unsteady inlet flow distortion obtaining a good match with the experimental data.

This study approaches the problem of inlet flow distortion with particular attention to the use of computer resources and time required. The aim is to try to develop a CFD simulation of the first stage of a transonic fan under clean and distorted conditions using a personal computer with a quite common RAM memory of 1 GB within reasonable amount of time (usually less than 10 hours for a distorted flow simulation).

The compressor used was especially developed for this study and it is a three stages transonic fan with a design sea level mass flow of 70 kg/s and a pressure ratio of approximately 4.2. The geometry and the grid have been created inside CFX-TurboGrid and the simulations have been performed in CFX-TASCflow.

The performance of the first stage was analyzed on a constant rotational speed line and the effects of different patterns of distortion (namely radial and stratified) were compared near design point, stall and choke, aiming to explain the displacement of the stage characteristic. The attention was then focused on the effect of the level of the distortion on the stage characteristic.

METHODOLOGY

The presence of non-uniform flows at the inlet of a gas-turbine immediately affects the performance of the low pressure compressor. For this reason the study focuses only on the first of the 3-stages of a military engine fan, designed particularly for this project. The design of the compressor was accomplished with the use of physical relations (such as the continuity equation and the Euler's turbomachinery equation) and some empirical rules for the optimization of the blade geometry. Double circular arc blades were found to be a very good choice in terms of performance under various conditions, simplifying also notably the design process. The final achievements of this design were a sea level design mass flow of 70 kg/s with a pressure ratio of 4.2, at the rotational speed of 12,000 rpm. Further details on all the design process and on the final compressor characteristics are reported in [6].

All the simulations were developed for a flight altitude of 11,000 meters with a Mach number of 1.2, for a reduced inlet undistorted total pressure of 55 kPa and a total temperature of 279.3 K. The salient features of the stage performance are the following:

Pressure ratio	1.72
Isentropic efficiency	88.2 percent
Mass flow	39 kg/s
Rotor tip relative Mach number	1.63
Rotor blade aspect ratio	2
Stator blade aspect ratio	1.8
Rotor space/chord ratio	0.72 (mid-span)
Stator space/chord ratio	0.80 (mid-span)
Rotor blade number	19
Stator blade number	23

The stage was analyzed with a CFD commercial code (CFX-TASCflow developed by AEA Technology) and the performance under clean conditions were taken as the base to evaluate the effects of inlet flow distortions.

Grid Generation

The grid was created within CFX-Turbogrid, that provides an efficient tool for generating structured hexahedral mesh.

The importance of a good grid for a CFD analysis is clearly very high. The nature of the study itself, which is a numerical simulation of the flow path inside the compressor, gives a fundamental role to the location and the number of the grid points. However, being one of the main aims of this project to try to approach the problem with a fairly limited use of computer resources, big attention was devoted to the creation of a grid that could be a good compromise between accuracy and computational time.

In particular, as the optimum grid size is a fundamental issue for the optimisation of the objective mentioned above and because of the conflicting opinions available in the literature ([7] used 200,000 grid nodes per flow passage for a similar simulation, [4] 370,000 while [5] obtained satisfying results with a much coarser grid of 23,000 points per flow passage), a study was completed on the effect of the grid size on the solution for the stage operating at design point. This led to the final choice of a grid composed by 89 points in the meridional direction, 32 in the tangential one and 24 spanwise (in total 68,000 points per passage for each blade row – Figures 1 and 2) which allowed to keep the relative errors of the principal parameters used to assess compressor performance below 0.2% (calculated relatively to a 200,000 points grid). This led to a reasonable computing time (less than 24 hours for the most complex time-dependent runs). A complete explanation of the whole process is available in [6].

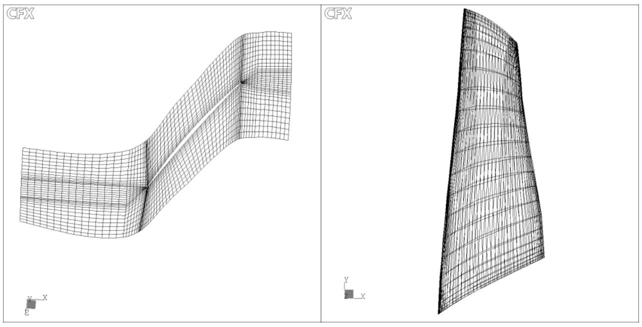


Figure 1 – Rotor grid

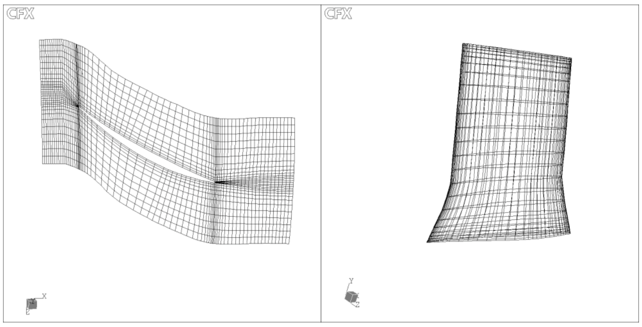


Figure 2 – Stator grid

Numerical Method

The two grids (rotor and stator) created separately were attached inside CFX-TASCflow. The use of a "stage" attachment in the mixing plane between the two blade rows had the effect of averaging circumferentially all the flow properties at the exit of the rotor. Clearly this introduced some approximation and is usually recommended for simulations such as rotor/stator interaction near the design point (where no big wake is generated). The presence of the mixing plane between the two blade rows would probably have the effect of partially damping the distortion itself as this enters the stator: several studies have been carried out in recent years at Cranfield University (UK) to understand the level of inaccuracy introduced by this assumption, finding the change in results to be always limited to some percent. Anyway, this is at the moment the only feasible approach for this type of simulations on a complete stage, because the use of a time varying attachment would introduce a double level of time dependency, which would raise exponentially the computational resources required Besides, it is surely a good approximation as first attempt to study the effect of inlet flow distortion on a compressor stage.

The simulations were performed with the standard k-ε model provided inside the software, with the aid of a wall-function to solve the flow near the walls. The discretisation method used to solve the Navier-Stokes equations is the "Modified Linear Profile Scheme", which is the one that gives the best trade off between accuracy and robustness. It is an implicit method first order accurate in time and second order accurate in space.

Clean inlet performance

The compressor performance under clean inlet conditions was analyzed, not only for stage performance and methodology validation, but to determine the design constant non-dimensional rotational speed line as well, which was then used as the baseline for evaluating the distorted performance.

The constant rotational speed chosen for obtaining the compressor line is obviously the design rotational speed (12000 rpm) and the simulations were done by imposing the inlet total pressure, the total temperature and the direction of the flow as the design ones and the average static pressure in the outlet. The various points of the map have been obtained by changing the outlet static pressure. This is equivalent to allow more or less mass flow to pass through the passage, changing therefore the operating point of the fan.

The method used for the evaluation of the compressor performance was expected to work properly at design point and in general when the incidence of the flow on the blade was low enough not to give big separation. More problems were on the contrary expected in near stall conditions, for the problems related to the use of a steady state simulation to analyze a stalled flow, which is turbulent and hence fluctuating by definition.

The performance of the compressor was however analyzed until the last point where the method showed convergence, assuming the stall to be located where the convergence stopped due to the presence of unsteady phenomena which could not be captured by the solver.

The shape of the constant speed line is typical of a stage characteristic. The maximum pressure ratio of the whole stage is just below 1.85 and the peak efficiency is in the order of 88.3% (Figures 3 and 4).

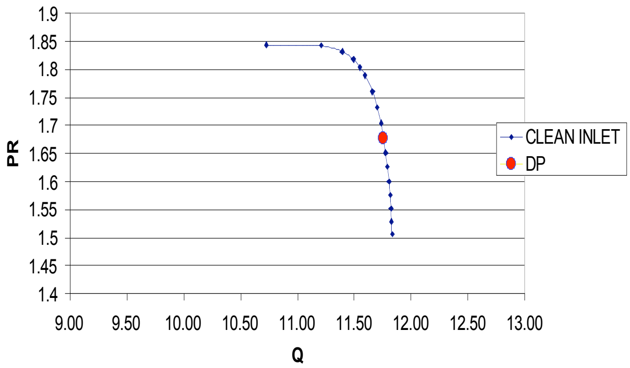


Figure 3 – Stage undistorted characteristic

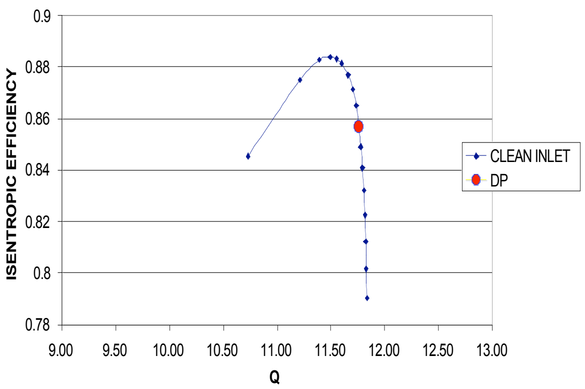


Figure 4 – Stage undistorted efficiency curve

Distorted performance

Both the grid and the numerical methods used for the analysis of the distorted performance of the compressor are the same as in the clean inlet case. The main difference is related to the boundary conditions given at the inlet of the stage, where the value of total pressure is no more uniform.

The stage performance was simulated once again by using one blade passage. Clearly this is a reasonable approximation when the extent of the blade passage itself is small compared with the distorted sector. To increase the capability of the method, the inlet of the blade passage was divided in several parts (both radially and circumferentially), allowing a better approximation of the distorted pattern.

Radial and mixed distortion (named "stratified"), were studied and the effect of the level of the distortion on the stage performance was analysed on a stratified distortion pattern. The approach used is quite similar, but while the first type can be solved with a steady simulation, the second one involves time periodic simulations, with a considerable increase of time and computer resources required.

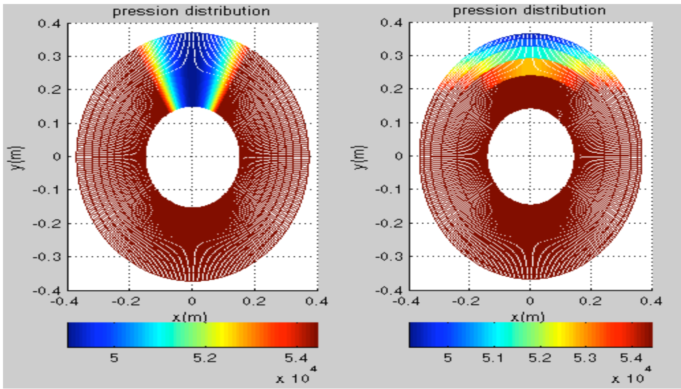


Figure 5 – Radial and stratified distortion patterns

Once again the simulations were performed by imposing the inlet total pressure, the total temperature and the direction of the flow, plus the exit static pressure. Considering the low extent of the distortion (usually in the order of 20% of the whole annulus) and the not too big level of distortion (however the biggest distortion levels usually encountered in aircraft applications are in the order of 20%), the approximation of a constant outlet static pressure can be considered acceptable. The different points on the map were obtained again by changing the outlet static pressure.

Clearly now the inlet total pressure is no longer constant (and varies linearly until the point of maximum distortion – see Figure 5), in both time and space. CFX-TASCflow allows time periodic boundary conditions to be insert as a function only of the time. The problem of the dependence of the inlet flow characteristics from the position along the span of the blade was solved by subdividing the inlet of the flow passage in several "inflows", each one being linked to a different function of the time (which is proportional to the angular position of the blade). An example of the inlet used for the study of the stratified distortion is shown in Figure 6.

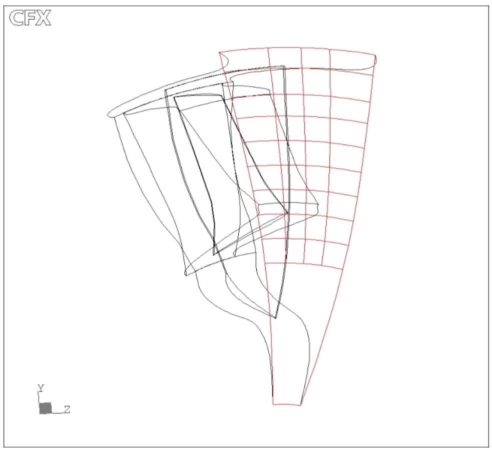


Figure 6 – Inlet configuration

The total pressure distortion was been approximated with a polynomial in the distorted region, as is shown in Figure 7, again in the case of the stratified distortion affecting the 20% of the annulus with a level of 15%.

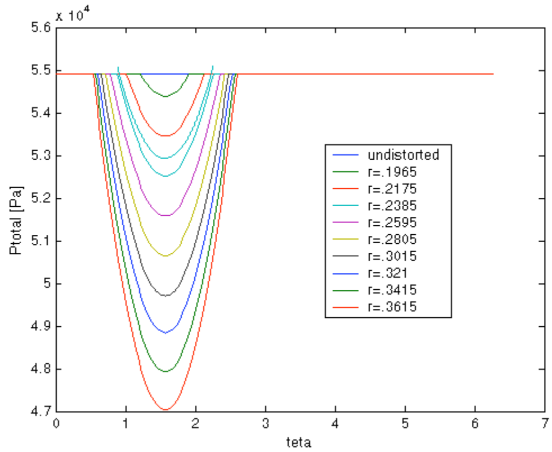


Figure 7 – Numerical functions used to simulate the inlet distortion

A choice was made to keep the outlet static pressure constant during the simulation. This is not exactly true especially when one stage is studied in isolation, but is more and more justified if the distortion studied is quite small. Clearly this is just one approximation, but it seems to be the most reasonable in absence of experiments which could give an idea of the trend. The same approach has been used in some others CFD studies on distortions [4].

To make this approach more acceptable, another solution was used, which is an adaptation of the "parallel diffuser" technique already used in other studies [8]. By putting a long enough diffuser at the exit of the stator, the problem of the specification of a fluctuating exit static pressure could be avoided (Figure 8). The portion of the duct with lower total pressure and then lower inlet velocity produces a lower static pressure drop than the portion with high inlet velocity and total pressure. Simplifying, the presence of the diffuser allows fixing the constant static pressure condition far enough from the exit of the stage, where the static pressure changes accordingly with the distortion imposed.

Furthermore, this solution facilitated the convergence especially in the near stall simulations, where a bigger wake was expected to be generated, allowing the study of transient simulation quite close to the compressor surge.

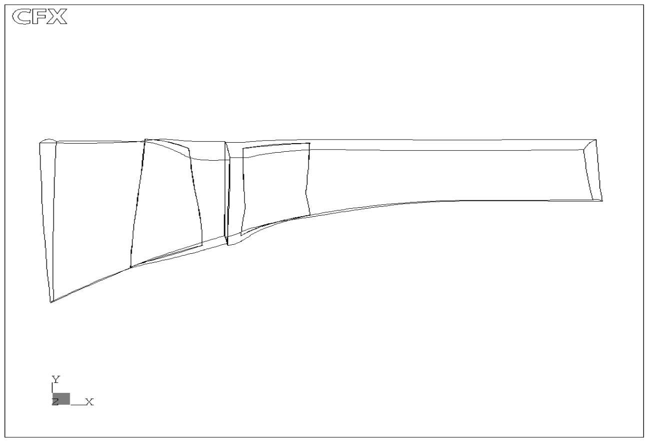


Figure 8 – Exit diffuser

RESULTS – GENERAL CONSIDERATIONS

As stated earlier, two different types of inlet total pressure distortions were simulated. The main difference between them is the computational method used: while the radial distortions can be simulated with a steady state approach, stratified distortions need a time periodic approach, which requires a bigger computational time and particular attention to some parameters such as the time step of the simulation which is responsible for the convergence of the solution.

Because of the relative simplicity of the problem, radial distortions allow an easier understanding of the phenomena involved and could be a good first step in the study of the effect of uneven inlet flow distributions. On the other side, mixed distortions are without any doubts more interesting, because they constitute the most common types of distortion encountered by the engine in real operations and the most dangerous ones for its stability.

Furthermore, since it was impossible to set up a comparison with experimental data (not available for this just designed fan), a simpler distortion gives the possibility of validating the method against several distortion studies developed in the past, which are mainly on the subject of radial and circumferential distortions.

Radial distortions (steady simulations)

The principal effect of the presence of the low total pressure region is the change in the spanwise distribution of the flow and in the related parameters.

The first important parameter analyzed is the axial velocity of the flow at the inlet of the compressor. The simulations with uneven pressure distribution show a lower axial flow velocity in the distorted region and a bigger one in the undistorted region, compared with the clean inlet results. This difference in the axial flow distribution was expected as it is one of the most recognised effect of radial distortions. An example is shown for tip radial distortions of different levels in Figure 9.

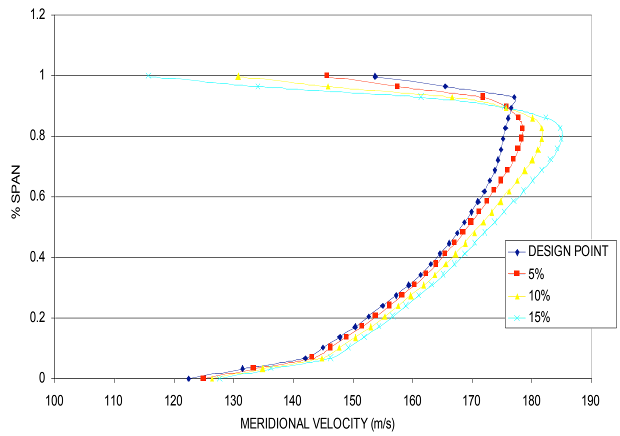


Figure 9 – Axial flow velocity distribution change for tip radial distortion

The presence of a different absolute velocity clearly affects the distribution of the flow around the blade and particularly the incidence of the flow itself on the blade. A bigger incidence causes the load on the blade to rise, because of the bigger deflection that the blade row is trying to impose to the flow. This can represent a problem for the compressor stability especially in regions where the blades are already highly loaded, such as the hub and the tip.

The temperature rise across the blade row (being proportional to the change in whirl velocity imposed to the air) is a significative measure of the load carried by the blade. Figure 10 shows how the decrease in inlet axial velocity is directly linked to an increase in temperature rise (the inlet total temperature of the flow is kept constant between undistorted and distorted simulations).

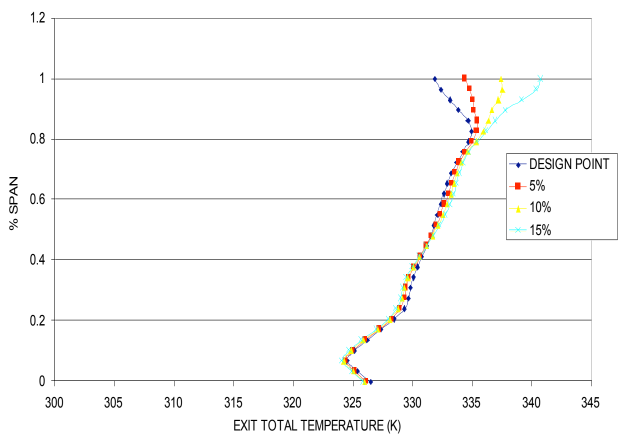


Figure 10 – Exit temperature distribution change for tip radial distortion

The increase in temperature rise was however usually found to be not enough to overcome the lower inlet pressure: the second stage will therefore have to deal again with a distortion in pressure (less significant than the inlet one) and with a new distortion in temperature, that will make the compression of the flow more difficult.

The efficiency deserves particular attention: clearly each profile has an incidence of maximum efficiency. Relative to this incidence the efficiency decreases both if the angle of the flow relative to the blade rises (due to a bigger separation) and if this angle is reduced (due to the bigger frictions of the flow on the blade). Therefore if the distortion happens near design point or even more near choke, where the incidence of the flow on the blade is low, the reduced axial velocity could give an improvement in the compressor efficiency, while the same condition in the last part of the compressor constant speed line would surely worsen the performance, or even stall the portion of the blade affected by the distortion if this is already highly loaded and the distortion itself is big enough. The distortion will however have the effect of reducing the stall margin of the compressor. If the region of the compressor affected by the stall is too large, dangerous phenomena such as surge or rotating stall can affect the engine stability.

Another very important loss from the engine point of view is the decrease in mass flow created by the reduced axial velocity of the flow, that reduces the thrust extracted from the engine.

Clearly it is impossible to compare this results with experimental ones, because they strictly depend upon the particular compressor geometry. However the comparison of these with the analysis of past studies shows a good and encouraging matching. It is worth to mention the results shown in [9], [10] and [11], which found the principal effect of radial tip distortion to be the increase in loading in a region that was already heavily loaded, with the related effects of change in performance and in efficiency, that could face slight improvements or reductions depending on the particular operating point.

Stratified distortions (unsteady simulations)

As stated earlier, the principal difference in these simulations is given by the fact that, as the generic blade passes through different conditions as it moves past the annulus, a time periodic simulation is needed. The number of time steps required to have a convergent solution is strictly related to the particular distortion considered. The results can be analyzed in terms of averaged parameters, which basically express the mean of the flow properties across the whole annulus, but it is also interesting to see how they vary while the blade is moving across the annulus through the distorted region.

The basic change (in function of which all the others can be explained) introduced by the presence of the low pressure region is again a lower axial velocity of the flow. This causes an increased incidence of the flow on the blade, with a consequent rise of the load and a reduction of the margin that the profile has relative to the stall.

Figure 11 shows the variation in inlet axial velocity for a stratified distortion (10% level) when the blade moves from the undistorted region through the low pressure one.

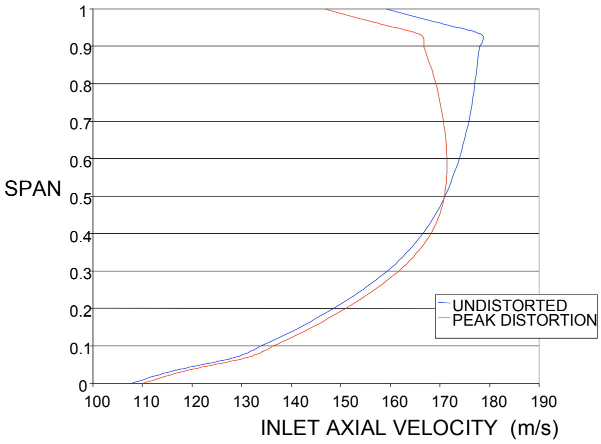


Figure 11 – Inlet axial velocity change for a stratified distortion

An extensive comparison of the results with experimental ones is now more difficult, mainly because of the lack of experimental or numerical studies of stratified distortions. However the general trends found are similar to the ones shown in several previous studies such as [12], [13] and [14].

Clearly the first parameter affected from this variation in inlet flow velocity in the distorted sector is the mass flow. Figure 12 shows the decrease in mass flow (non-dimensionalised with the undistorted value).

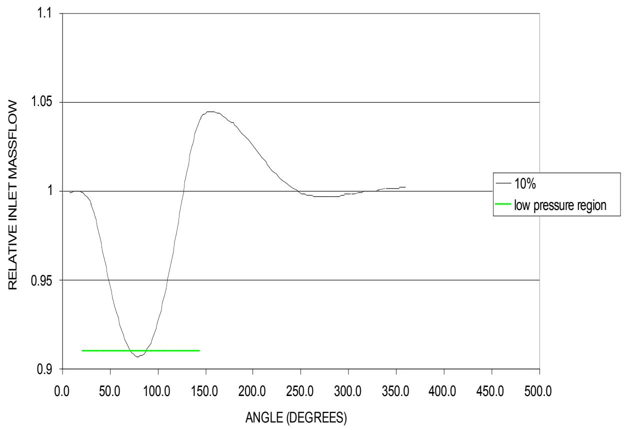


Figure 12 – Mass flow reduction in the spoiled region

As the simulation is now transient, there is no longer equilibrium between the mass flow entering and exiting the control volume in the generic time step. There is a delay in the transfer of the information between the inlet and the outlet of the stage, evident from the exit mass flow (Figure 13), where the effect of the distorted region has moved in the same direction as the rotation of the spool and has been largely attenuated. This clearly affects the second stage in a favorable manner, because it will have to face lower operating problems than the first one.

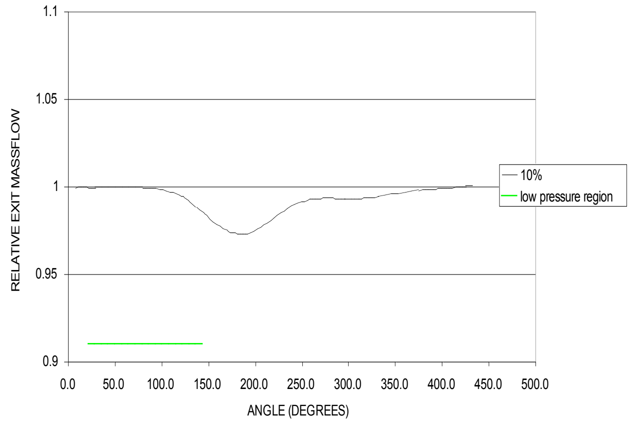


Figure 13 – Exit relative mass flow

Due to the bigger incidence angle, the temperature ratio increases in the distorted region. The work done by the compressor on the fluid is bigger because bigger is the lift exerted on the blade and the turning imparted to the air also increases. This will have again (as for the radial distortion) the positive effect of reducing the level of pressure distortion faced by the following stages, but at the same time will create a temperature distortion with its related problems. The reduction in pressure distortion across the stage and the creation of a temperature distortion is evident in Figures 14 and 15, which again refer to the stratified distortion case named above. This is clearly true only if the distortion is not big enough to separate the flow.

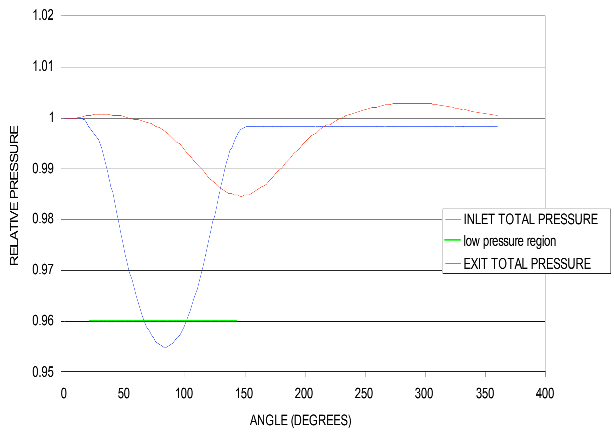


Figure 14 – Comparison of inlet and outlet pressure variation

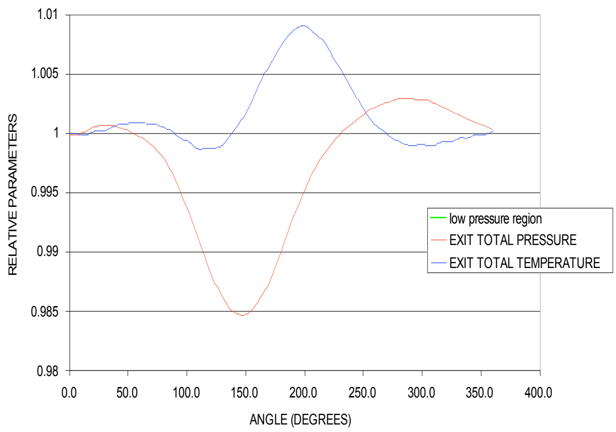


Figure 15 – Pressure and temperature ratios relative to undistorted conditions

From Figure 15 pressure and temperature distortions between first and second stage of the compressor do not seem to be perfectly in phase. This phenomenon is due to the fact that all the data extracted are mass averaged, and the increase in temperature in the spoiled sector (20% of the annulus in this case) is damped by the decrease in mass flow in the same sector, while the decrease in pressure (much bigger) is only partially attenuated by the lower massflow.

The efficiency deserves a particular comment: while the ones outlined until now are general issues that have been noticed in all the simulations runs during the project, there is not really a general trend for the isentropic efficiency. However it can be asserted that the effect on the efficiency is the result of principally two conditions: the working point of the compressor and the distortion pattern: more specifically, if the distortion happens far enough from the compressor surge line, the performance of the compressor could also be improved by the presence of a lower inlet pressure which tend to move the incidence angle towards the optimum one. On the contrary, if the margin from the surge line is already limited, a further rise of the incidence of the flow on the blade would probably worsen the performance by increasing the deviation of the flow leaving the profile. This phenomenon can be exacerbated in cases of big levels of distortion in the last part of the constant speed line. The general effect is therefore a reduction of the surge margin of the compressor, that will appear more evident from the study presented below.

RESULTS – PARTICULAR

Different levels of distortion and stage characteristics

Starting from the stage undistorted performance outlined in page 3 (Figures 3 and 4), three different levels of distortion (5%,10% and 15%) were simulated in three different working conditions of the compressor, one near design point, one closer to choke and one closer to stall, to understand the behavior of the constant speed line under distorted conditions. The shape of the distorted region was always kept unchanged (stratified distortion affecting 20% of the annulus area). A picture of the inlet pressure contours in the case with 15% distortion is shown in Figure 16.

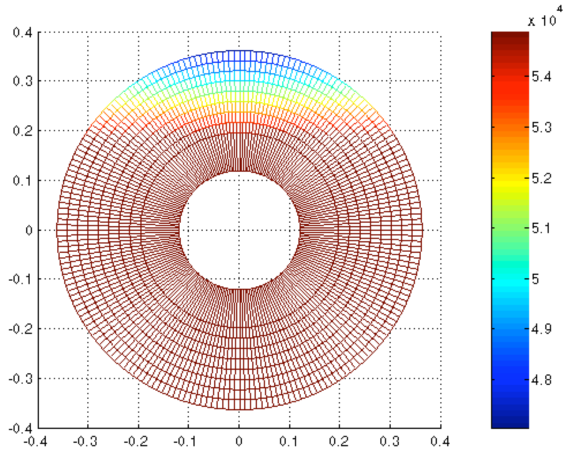


Figure 16 – Stratified distortion pattern

In the cases studied, the principal effect of a bigger distortion level was found to be a bigger reduction in the axial velocity of the flow, that is linked to reduction in mass flow and an increase in pressure ratio and temperature ratio, as largely explained before. The three different levels of distortion produce in the three points studied a very similar time history, due to the fact that none of them was strong enough to produce an excessive change in incidence of the flow and then to stall the blade. For example, the time-histories of inlet mass flow and of exit total pressure are shown for the distortion affecting the stage at the original design point of the stage itself (Figures 17 and 18).

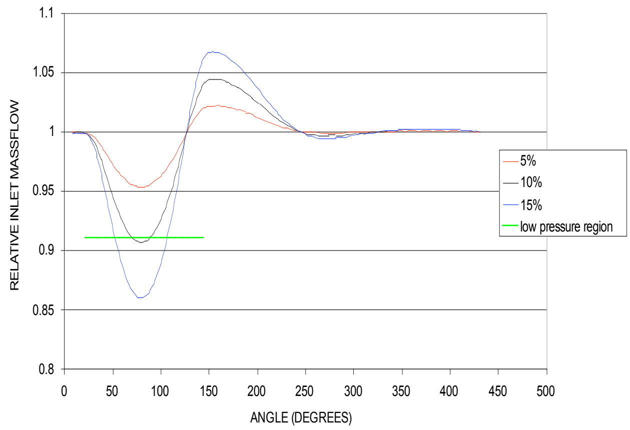


Figure 17 – Inlet mass flow for different levels of distortion

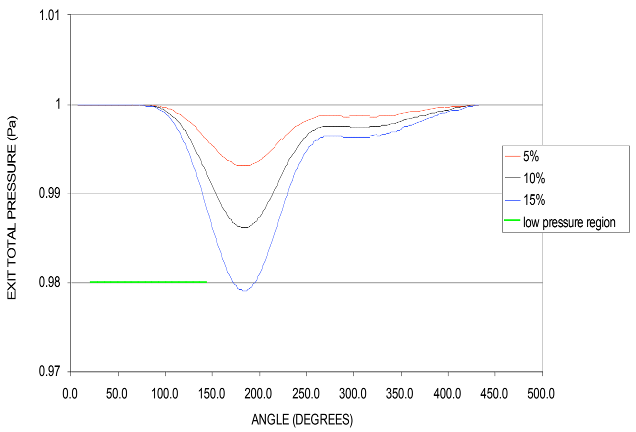
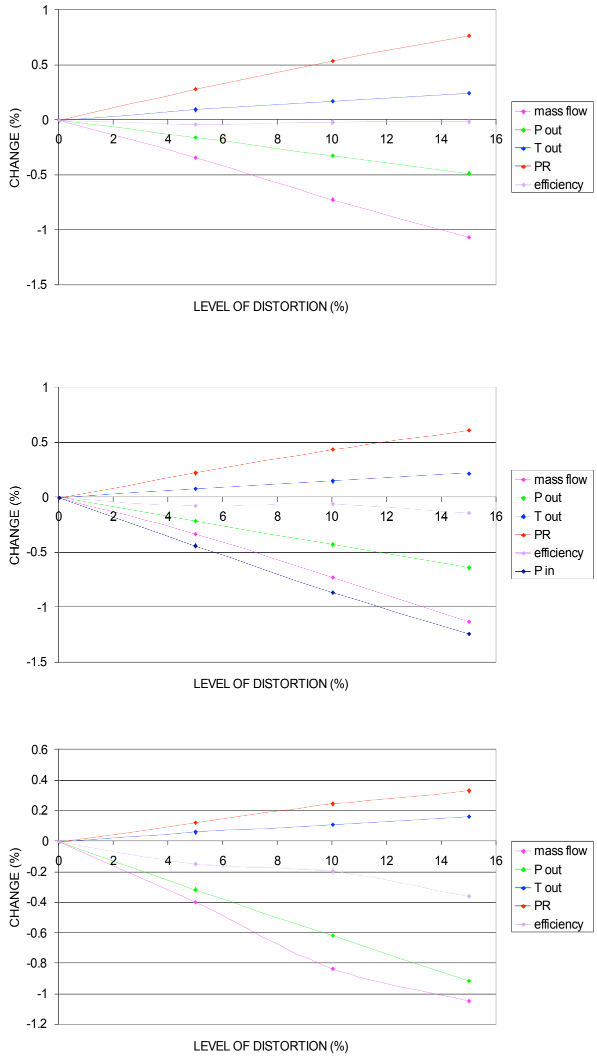


Figure 18 – Exit pressure for different levels of distortion

The global effect of the presence of a distorted inflow was summarized by averaging all the different parameters (weighted averaging). The three points where these effects were studied do not show big differences. The variations relatively to the undistorted conditions are always the same previously outlined (drop in mass flow, rise in pressure and temperature ratios). It is however interesting to notice that, while the efficiency keeps almost constant for the three levels of distortion near choke and near design point, the simulations performed near stall show a decrease in efficiency as the level of the distortion rises. This is surely due to the fact that in these operating conditions the incidence of the flow on the blade is already close to its limit and the presence of a lower pressure region has the effect of raising it further, causing a big separation and then a drop in efficiency. All these results are on line with the principal foundings in the field and especially with [10], who found a strict dependence of the change in performance (and particularly of the efficiency) from the analyzed operational point. This is evident in Figure 19.



(a) (b) (c)

Figure 19 – Change in performance near choke (a), design point (b) and stall (c)

The changes in the flow parameters appear to be proportional to the level of distortion

The same features are noticeable in the stage maps, which for the distorted conditions are presented together with the undistorted ones in Figures 20 and 21.

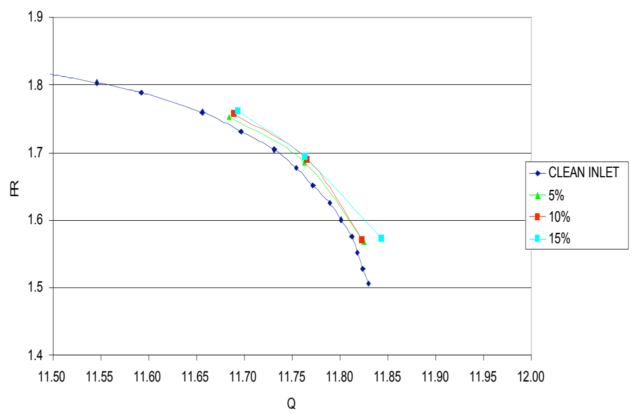


Figure 20 – Stage distorted characteristics (stratified distortion)

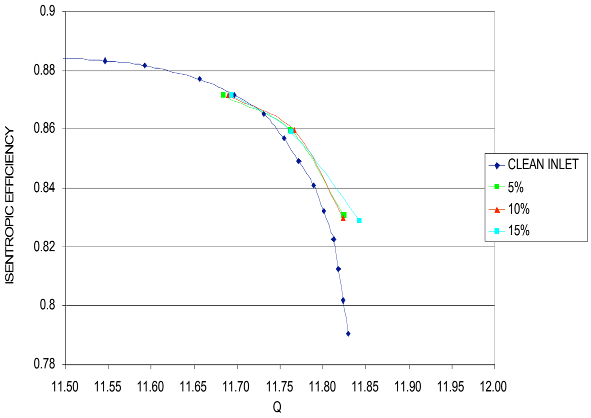


Figure 21 – Stage distorted efficiency map (stratified distortion)

While the drop in mass flow is not evident because of the inlet total pressure drop that makes the non-dimensional mass flow to rise (with a general displacement of the constant speed line toward higher mass flows, already evident in [10]), two other effects are clear: firstly, the pressure ratio has increased (Figure 20), secondly, in the last point (near surge) the efficiency is starting to drop (Figure 21).

In order to better understand this phenomenon, more simulations have been performed for three higher pressure ratios for the 15% distortion case. In the two last cases a big separation was found while the blade was passing through the distorted region, in contrast to an undistorted non separated flow. The load on the blade rises due to the increased incidence of the flow and reaches a value that the blade is not able to carry anymore. The flow separates (Figure 22).

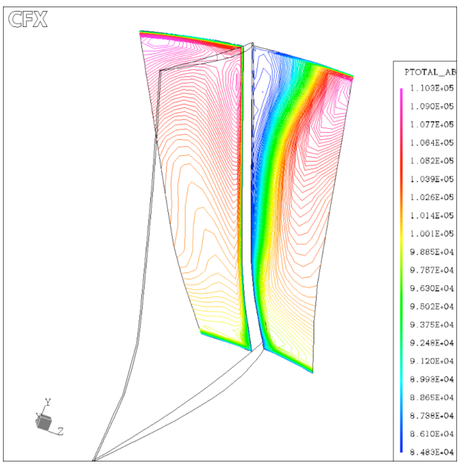


Figure 22 – Separation in the tip region due to the presence of the distorted region

The effect of this phenomenon on the pressure ratio is really minimal because the remainder of the blade is still working properly. A reduction of the efficiency is however present. This is shown more clearly by comparing the stage map and the efficiency map with 15% distortion with the ones with undistorted inlet conditions (Figures 23 and 24). Furthermore, this could represent a very important problem for the compressor stability, because the stall at a the tip of a transonic compressor blade can often evolve and produce a surge event or a rotating stall, very difficult to analyze in a CFD simulation.

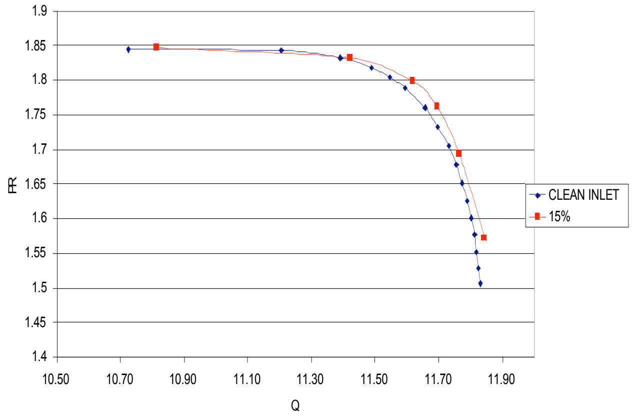


Figure 23 – Stage map comparison: undistorted conditions with 15% stratified distortion

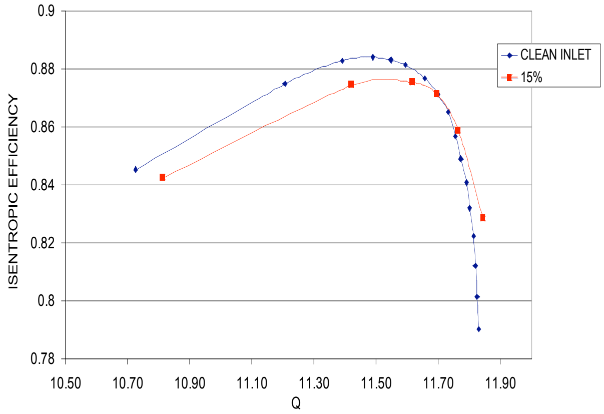


Figure 24 - Efficiency map comparison: undistorted conditions with 15% stratified distortion

Some final considerations can be summarized with the help of these last two maps. At low pressure ratios the principal effect of the presence of the distorted region is the reduction in actual mass flow; pressure ratio and efficiency tend to rise due to the incidence approaching zero from negative values. At high pressure ratios the mass flow is still reduced but pressure ratio and efficiency do not rise anymore because the blade is very close to its limits (in terms of load). For the compressor analyzed the maximum pressure ratio is roughly the same as the undistorted case, while a significant reduction in the peak efficiency is present due to the above described separation of the flow

More complex is to analyze the variation in surge margin: the tip region of the stator blade has been found to be stalled in the last simulations. Due to the relatively small extent of the separated region, the efficiency drop is not dramatic, but still remarkable. However, the simulations performed allow the general trend to be understood: the presence of a lower pressure region moves the operating point towards the surge (bigger pressure ratios and lower massflows), therefore reducing the surge margin of the compressor, as expected from several studies ([1], [10] and [11] for example).

CONCLUDING REMARKS

This project tries to approach the problem of CFD analysis of inlet flow distortions with particular care for the computer resources and the computational time required. Different types of distortion have been tested on an expressly designed compressor, finding the main effect to be the drop of the axial velocity of the flow in the distorted region. This drop leads to the change of some of the global parameters, which usually describe the performance of a compressor, such as mass flow, pressure ratio and efficiency. All the different effects are inter-related: the decrease in axial velocity changes the direction of the flow relative to the rotating blade, increasing the load on the blade and moving the operating point towards the surge line. Clearly the complete simulation of the flow behavior close to surge is very difficult because of the complex nature of a flow with big incidence and big separation. However, the changes in the stage speed line have been outlined, together with the principal trends in presence of distorted flow.

The results collected during this study were compared with some important experimental foundings in this field, showing an interesting similarity that surely suggests further analysis in the capabilities of CFD in analyzing distorted flows in turbomachinery. It would be surely interesting to carry out a more detailed comparison with experimental results, not possible in this case because of the peculiarity of the compressor used.

A part from the general analysis, a more detailed study on the effect of the level of distortion on the compressor performance was completed. The results were very satisfying and demonstrate all the possibilities of this tool in analyzing complex turbomachinery phenomena.

Furthermore, the same approach could allow many types of distortions to be studied (not only pressure ones but really any type of distortion) with different patterns. This could lead to a more precise and focused use of CFD in turbomachinery design, which could help in saving time and resources by bringing complex analysis into the very early phases of the design process.

AKNOWLEDGEMENTS

This work has been performed in the framework of a double degree agreement between Politecnico di Torino, Italy and Cranfield University, UK and led to the simultaneous acquisition of a British MSc and of an Italian Laurea. As such, a dominant part of the related activity was conducted in the Cranfield laboratories. The Author is deeply indebted to his British supervisor, Professor Pericles Pilidis, for his continuous and essential advise.

REFERENCES

[1] Reid, C., 1969, *The Response of Axial Flow Compressor to Intake Flow Distortion*, ASME Paper 69-DT-29

[2] Walker, G.J., 1993, *The Role of Laminar-Turbulent Transition in Gas Turbine Engines: a Discussion*, Journal of Turbomachinery volume 115, pp 207-217

[3] Jennions, I.K., Turner, M.G., 1993, *Three Dimensional Computations of a Transonic Fan Using an Explicit Flow Solver and an Implicit k-ε Solver,* Journal of Turbomachinery volume 115, pp 261-272

[4] Hirai, K., Kodama, H., Nozaki, Kikuchi, Tamura, Matsuo, 1997, *Un steady Three- Dimensional Analysis of Inlet Distortion in Turbomachinery,* AIAA-97/2735

[5] Davis, M., Hale, A., Beale, D., 2002, *An argument for Enhancement of the current Inlet Distortion Ground Test Practice for Aircraft Gas Turbine Engines*, ASME 2001-GT-507

[6] Ghisu, T., 2003, *Axial Compressor Response to Inlet Flow Distortion by a CFD Analysis,* joint MSc Thesis and Tesi di Laurea, respectively at Cranfield University and Politecnico di Torino

[7] Copenhaver, W.W., Hah, C., Puterbaugh, S.L., 1993, *Three Dimensional Flow Phenomena in a Transonic High-throughflow, Axial-Flow Compressor Stage,* Journal of Turbomachinery, Vol. 115, No. 2, ASME

[8] Mikolajczak, A.A., and Pfeffer, A.M., 1974, *"Methods to Increase Engine Stability and Tolerance to Distortion",* AGARD Lecture Series No. 72, Sect. 7.

[9] Schmidt, J.F., Ruggeri, R.S., 1978, *Performance With and Without Inlet Radial Distortion of a Transonic Fan Stage Designed for Reduced Loading in the Tip Region*, NASA Technical Paper 1294

[10] Conrad, W., Sobolewski, A.E., 1950, *Investigation of Effects of Inlet-Air Velocity Distortion on Performance of Turbojet Engine,* NACA Research Memorandum, RM E50G11

[11] Harry, D.P., Lubick, R.J., 1955, *Inlet-Air Distortion Effect on Stall, Surge, and Acceleration Margin of a Turbojet Engine Equipped with Variable Compressor Inlet Guide Vanes,* NACA Research Memorandum, RM E54K26

[12] Smith, I.D., Braithwaite, W.M., Clavert, H.F., 1956, Effect of Inlet Air Flow Distortions on Steady State Performance of J65-B-3 Turbojet Engine, NACA Research Memorandum, RM E55I09

[13] Huntley, S.C:, Sivo, J.N., Walker, C.L., 1955, *Effect of Circumferential Total Pressure Gradients Typical of Single Duct Installation on Performance of an Axial Flow Turbojet Engine,* NACA Research Memorandum, RM E54K26a

[14] Conrad, E.W., Hanson, M.P.J.E., 1955, *Effects of Inlet Air Flow Distortions on Steady State Altitude Performance of an Axial Flow Turbojet Engine,* NACA Research Memorandum, RM E55A04