Design and Numerical Investigation of Advanced Radial Inlet for a Centrifugal Compressor Stage

Paper Number: IMECE2004-60538

Yunbae Kim
Dresser-Rand Company
Olean, NY 14760

Jay Koch
Dresser-Rand Company
Olean, NY 14760

Key words : centrifugal compressor, radial inlet, numerical simulation, inlet distortion, secondary flow, incidence.

ABSTRACT

The performance of a centrifugal compressor stage can be seriously affected by inlet flow distortions due to an unsatisfactory inlet configuration and the resulting flow structure. In this study, two radial inlets were designed for a centrifugal compressor stage and investigated numerically using a commercially available 3D viscous Navier-Stokes code. The intent of the design was to minimize the total pressure loss across the inlet while distributing the flow as equally and uniformly as possible to the impeller inlet.

For each inlet model, the aerodynamic performance was calculated from the simulation results and then the results from both models were evaluated and compared. The second radial inlet design outperformed the initial design in terms of total pressure loss, flow distortion and uniformity at the impeller inlet. Furthermore, the aerodynamic performance of the second radial inlet was insensitive to wide range of mass flow rates compared to the initial design due to the distinctive geometric features implemented for the second inlet design.

INTRODUCTION

The use of radial inlets for centrifugal industrial compressors is very common. Piping and installation constraints, as well as other compressor mechanical constraints often govern the design of the inlet. These constraints often result in complicated geometry that is non-axisymmetric and beyond the scope of traditional one-dimensional design tools. In recent years Computational Fluid Dynamics (CFD) has been used successfully to model radial inlets and the results compared well with available test data (Flathers et. al., 1994, Koch et. al. 1995). While these studies have been very important for the inlet design itself, they did not focus on the impact of the inlet profile on impeller performance. One of the related studies (Hohlweg and Amineni, 2000) demonstrated that the changes in the inlet geometry can have noticeable impact on impeller performance. More experimental and numerical investigations were performed for the study of inlet distortion effect on the centrifugal impeller (Ariga et. al., 1982) and on the compressor stage (Kim et. al., 2001 and Engeda et. al., 2003). According to these studies, the flow distortion upstream of impeller can cause significant efficiency drop and reduce the surge margin for the compressor stage while the performance penalty with incidence due to the flow distortion is relatively more on the overload side.

One area that is not addressed in these studies is the radial inlet design where the installation constraints have significantly changed the original design constraints. One example occurs when an existing unit is retrofitted with fewer stages. This results in a large increase in the available space for the inlet. Another example is the replacement of an existing compressor with a new compressor. The client will often request that the new compressor flanges be within the same envelope as the existing compressor to eliminate/minimize process piping changes. This can result in a nozzle with multiple bends that can lead to a distorted velocity profile inside the inlet. The inlet design presented in this paper requires both an inlet nozzle that is not optimally placed above the impeller and a long axial extension of the inlet to the impeller inlet.

NOMENCLATURE

Abs	Absolute quantity
C	Velocity
mdot	Mass flow rate
P	Static pressure
Pt	Total pressure
LC	Total pressure loss coefficient, $(Pt_1 - Pt_2)/(Pt_1 - P_1)$

Greek

α	Flow angle
β	Relative flow angle
σ	Standard deviation

Subscript

1	Inlet flange of radial inlet
2	Exit flange of radial inlet
Ave	Averaged quantity
θ	Tangential component

INLET DESIGN METHODOLOGY

The objective of a radial inlet for a compressor is to minimize the total pressure loss across the inlet while distributing the flow as uniformly as possible with the minimum distortion to the eye of the impeller. The inlet must also provide the prescribed level of inlet swirl. If the inlet flow is highly distorted does not provide the prescribed level of inlet swirl, the compressor efficiency can be degraded significantly, first at impeller and next in the following downstream components due to the propagation of the undesired flow characteristics. By matching the design parameters properly with the minimum flow distortion between an inlet and impeller, it is possible to bring the best efficiency and operating range for a compressor stage. Therefore, the downstream flow properties of an inlet can have a strong influence on the performance of the entire compressor stage. The three main aspects of compressor radial inlet performance are incidence, total pressure loss, and flow uniformity/distortion.

Incidence for vanes (blades) in stationary (rotating) components is defined as the difference between absolute (relative) flow angle and actual vane (blade) angle. The incidence loss occurs due to the angle of attack of the flow at vanes (blades). In case of impeller, if the relative flow angle does not coincide with the blade angle, the tangential component of relative velocity will be wasted and appears as a head loss. If the incidence loss is excessive, additional losses due to the boundary layer separation occurs. Positive incidence occurs due to the decreased meridional velocity and negative incidence occurs due to the increased meridional velocity. The former and the latter correspond to the incidences that cause the flow to impinge on the pressure and the suction side of the blade, respectively. Radial inlet design concerns the incidences at scoop vanes and IGV vanes as well as at impeller blade leading edge due to the non-uniform or non-prescribed swirling feature of flow from the inlet component.

Total pressure loss coefficient is defined as the total pressure difference between upstream and downstream stations divided by the upstream dynamic pressure. It is one of the important measures quantifying aerodynamic performance for a stationary component. Total pressure loss coefficient is an indication of what is happening in the flow passage as a result of surface friction, incidence, boundary layer growth, flow separation, stall, etc. When this parameter is used with the same reference dynamic pressure, the incremental total pressure loss towards the stations downstream can identify the performance characteristics at each component as well as make the comparison of different designs clear. Unless drastically significant, the magnitude of total pressure loss itself at inlet component in general does not have much contribution to the compressor stage efficiency degradation. It is the intensity of distortion or non-prescribed swirling feature of the flow that can have more influence on the performance of compressor stage. For this reason, relatively higher total pressure loss could, but not necessarily, mean more distorted flow feature. With the insertion of vanes if the flow can be guided properly, the inlet design can enhance the compressor performance even with a sacrifice of increased total pressure loss due to the increased surface friction.

Uniformity without any distortion would be the most desired flow condition for turbomachinery components. Due to the curved geometric nature of the components and spatial constraints in the design, certain degree of flow distortion is not often avoidable. The control of distortion for inlet components is particularly important due to the influence and the propagation of the inlet flow on the entire stage. For the present study, as the measures of the flow distortion/uniformity, various aerodynamic parameters including mass flow, meridional velocity, tangential velocity, static pressure, total pressure, flow angle and its standard deviation are evaluated at the end of IGV passages.

Therefore, in brief, the focus of inlet design work should be to minimize the total pressure loss and incidence as well as to make the flow as uniform as possible with the minimum distortion across the flow passage. Although the effects of the three aspects described previously are coupled to one another, individual evaluation of these aspects as the performance measure made the comparison of radial inlet models clear for the present study.

GEOMETRY AND GRID GENERATION

The geometry for each of the inlet designs was created with the Unigraphics CAD package and imported to ICEMCFD, grid generation suite from ANSYS. Figures 1 and 2 show the geometry of two radial inlet designs that are the subject of this study. The two inlet designs are distinctively different in two areas, the plenum distribution and the vanes used to direct the flow from the plenum into the downstream inlet guide vanes.

The first radial inlet design has three rows of vanes between the compressor flange and the impeller inlet. Two “splitter” vanes are located in the plenum region along the symmetric plane at the top and bottom of the inlet. The second row of vanes is located downstream of the plenum. These vanes and are relatively thin constant thickness vanes that are equally spaced circumferentially at the trailing edge, starting from 12 o’clock position. The third row of vanes is located in the axial duct downstream of the 90-degree bend.

The second radial inlet design has the same three rows of vanes between the compressor flange and the impeller inlet. The main distinction is the design of the vanes in the second row and the leading edge setting angles of these vanes. The profiles and the setting angles of scoop vanes were modified based on the simulation results of the first inlet design. The area distribution in the plenum of the second design was also modified to vary circumferentially based on the results of the first inlet design.

Due to the symmetric geometry for both of the radial inlet models, the grids were created for the half of the geometry only (180-degree sector model). This allowed a significant reduction of modeling and computation time. Prior published radial inlet studies have been done primarily with hexahedral element grids.

Hexahedral elements provide an efficient grid, but are time consuming to create for radial inlets due to the complicated geometric characteristics. This is often very limiting for design studies as the grid generation time reduces the number of geometric variations that can be reviewed. An alternative approach is to create a grid with tetrahedral elements that takes much less time to generate. As the overall grid size increases with a tetrahedral mesh, the resolution of the flow field and the overall accuracy of the solution approaches the results with a hexahedral mesh (Hutchinson et al). Therefore, reducing the grid generation time, at the expense of extending the computational time, can minimize the total engineering time required for a solution. Based on this reasoning, a tetrahedral mesh was created for this problem.

Figure 3 shows the tetrahedral mesh created for the geometry of radial inlet design #1. As shown in Figure 4, finer tetrahedral elements are implemented near the leading and trailing edge of vanes to capture the flow details around the vanes. After the initial tetrahedral mesh is created, 5 layers of prism mesh with an expansion ratio of 1.5 is implemented on all boundary walls for better resolution of boundary layer. The first grid point near the wall is determined based on the estimation of y+. The CFD results indicated an average y+ value around 200. Figure 5 illustrates the imbedded prism layers on the wall boundary. Effort was been made to have an equivalent mesh size and quality for both inlet designs. The summary of grid size for each of the inlet designs is shown below.

Inlet model	Geometry	Mesh type	Element count	Node count
Radial inlet 1	Half	Tetra / Prism	2223382	683227
Radial inlet 2	Half	Tetra / Prism	2218979	697084

BOUNDARY CONDITIONS AND MODEL SETUP

The total pressure, flow direction, and total temperature were imposed as the inlet boundary conditions. Mass flow rate was applied as the outflow boundary condition. Mass flow rate varied on the basis of Mach number ranging from 0.05 to 0.10 at the impeller inlet, which is roughly equivalent to 87.6% to 176% of the impeller design mass flow. Methane (CH_4) was used as the fluid in the domain.

CFX5.6, a commercially available 3D viscous code from ANSYS, was selected for the CFD analysis. The second order discretization scheme was used for all solutions. The imbalance of mass, momentum and energy between the inlet and the outlet were closely monitored and the residual of those were converged to a maximum residual level of 1.0e-04. All solutions were solved in parallel using a LINUX cluster.

SIMULATION RESULTS AND DISCUSSIONS

Figures 6 though 15 show the postprocessed results from the numerical simulation of both radial inlet designs. The simulation cases are classified based on the inflow Mach number 0.05, 0.075, and 0.10. As has been stated previously, the existence of plenum area where flow is distributed before being accelerated into the narrow annular passage makes the performance of the radial inlet design #2 less sensitive to the wide range of mass flow rate. Therefore, the streamline and vector plots are very similar for each of the flow rates specified. For this reason, qualitative plots corresponding to inflow Mach number 0.05 only are included here for both radial inlet design #1 and #2. For the purpose of quantitative comparison, various aerodynamic parameters including mass flow, meridional velocity, tangential velocity, static pressure, total pressure, flow angle and its standard deviation are evaluated at the impeller inlet.

Comparisons of the flow field for the two inlet designs are shown in Figures 6-8. The behavior of the fluid particles is different for the two designs. Based on the simulation results of radial inlet 1, the second vane row leading edge angles and profiles were adjusted to have lower levels of incidence and to improve the exit flow field.

Another major difference appears at the bottom of the inlet. In case of the radial inlet #1, the vane at the bottom of the plenum is not properly guiding the flow to the second vane row as indicated in Figure 6. The flow is stagnated and vortices are formed near the plenum vane at the bottom of the inlet before finding a path to the second vane row. This suggested that a reduction area, such as the implementation shown with radial inlet design #2. It can be noticed that this effect also influences the flow in the other passages much farther away from the bottom of the inlet. The swirling feature of the flow in radial inlet design #1 results in increased losses and a distorted flow profile in the passages downstream of the second vane row (as compared to the radial inlet design #2). It should be noted that the reduction of losses in inlet design #2 is quantified but it is not significant due to the low momentum flow in this regime.

Figures 9-15 show the quantitative performance comparison of the two radial inlet designs at various inflow Mach numbers. Note that the results are based on the numerical simulation of half of the geometry. The vane passages are numbered from top to bottom on the half of the geometry for each model. Also note that the number vanes is different between the first design and the second design. Figure 9 compares the designs in terms of the total pressure loss coefficient calculated between the inlet flange and impeller inlet. It indicates that less loss occurs with radial inlet design #2 over the range of inflow Mach numbers. Note that the total pressure loss is less influenced with the increased Mach number over the range in case of radial inlet design #2.

Figure 10 shows the normalized mass flow distribution. The mass flow distribution is equally distributed with $\pm 1\%$ among vane passages regardless of the inflow Mach number. Likewise, normalized static and total pressure plots shown in Figures 11 and 12 indicate a variation of $\pm 0.2\%$ with reference to the averaged quantities. Note that the inflow Mach number has less impact on the distribution of mass flow, pressure and total pressure among the passages for radial inlet design #2.

The evaluation of mass averaged tangential velocity or flow angle relative to meridional direction can be misleading due to the cancellation of positive and negative quantities coexisting in the flow regime. Both of the quantities are responsible for incidence at the impeller leading edge and thus the reduction of head. For this reason, tangential velocity, flow angle and its standard deviation are evaluated based on the absolute values, which are essentially the measure of flow distortion and uniformity in the region.

Figure 13 shows the comparison of tangential velocity distribution at IGV passage exit for the various inflow Mach numbers. The results for radial inlet design #1 indicate relatively higher distortion at vane passages 4 and 5. As previously stated, this resulted partially from the lack of flow control at the bottom of the inlet.

For inlet design #2 the magnitude of the tangential velocities is reduced and is less sensitive to the different inflow Mach numbers compared to the radial inlet design #1. Accordingly, as shown in Figure 14, the flow angle variation has been improved, indicating less distortion of the flow at the impeller inlet. The relatively higher tangential velocity and flow angle at the 4th vane passage for inlet design #2 are attributed to the high flow turning at the 2nd row of vanes upstream of this passage. Although the flow quality could be enhanced further by modification of the vane, it was considered a minor impact on the performance and not pursued. Figure 15 shows the comparison of the standard deviation of flow angle over the range of outflow Mach numbers. The standard deviation of flow angle is the measure of uniformity in the direction of flow. The uniformity of inlet design #2 was far less sensitive to the wide flow range and improved noticeably from inlet design #1.

SUMMARY AND CONCLUSIONS

Two radial inlet models have been designed and numerically simulated for a centrifugal compressor stage. Based on the CFD analysis of the first inlet design, the geometry was modified for the second inlet design. Tetrahedral/prism mesh provided fast consistent modeling for the complicated geometry allowing additional design variations to be evaluated. The comparison of aerodynamic performance indicated that the second inlet design outperformed the first inlet design by reducing total pressure loss across the inlet and having more uniformity and less distortion at the end of the inlet vanes for the following centrifugal impeller. Furthermore, the relatively insensitive aerodynamic profile for the second inlet design over the wide flow range makes the inlet design adaptable to a wide range of operating conditions.

ACKNOWLEDGMENTS

The authors wish to thank Dresser-Rand for the continuous support on this project and the permission to publish this paper.

REFERENCE

J. Koch, P. Chow, B. Hutchinson, S. Elias, 1995, “Experimental and Computational Study of a Radial Compressor Inlet”, ASME 95-GT-82.

M. Flathers, G. Bache, 1994, “An Experimental and Computatinal Investigation of Flow in a Radial Inlet of an Industrial Pipeline Centrifugal Compressor”, ASME 94-GT-134.

W. Hohlweg, N. Amineni, 2000, “Effect of Reduced Inlet Space on A Medium Flow Coefficient Centrifugal Compressor Stage”, ASME IMECE 2000, PID-Vol.5, pp. 99-108.

B. Hutchinson, F. Shi, J. Sorokes, J. Koch, “Investigation of Advanced CFD Methods and their Application to Centrifugal Compressors”.

I. Ariga, N. Kasai, S. Masuda, Y. Watanabe, I. Watanabe, 1982, “The Effect of Inlet Distortion on the Performance Characteristics of a Centrifugal Compressor”, ASME Paper 82-GT-92.

Y. Kim, A. Engeda, R. Aungier, G. Direnzi, 2001, “The Influence of Inlet Flow Distortion on the Performance of a Centrifugal Compressor and the Development of Improved Inlet Using Numerical Simulations”, Institution of Mechanical Engineers (IMechE), Journal of Power and Energy, 2001, Vol. 215, Part A, pp. 323-338.

A. Engeda, Y. Kim, R. Aungier, G. Direnzi, 2003, “The Inlet Flow Structure of a Centrifugal Compressor Stage and Its Influence on the Compressor Performance”, ASME Journal of Fluids Engineering, 2003, Vol. 125, 779-785.

FIGURES AND TABLES

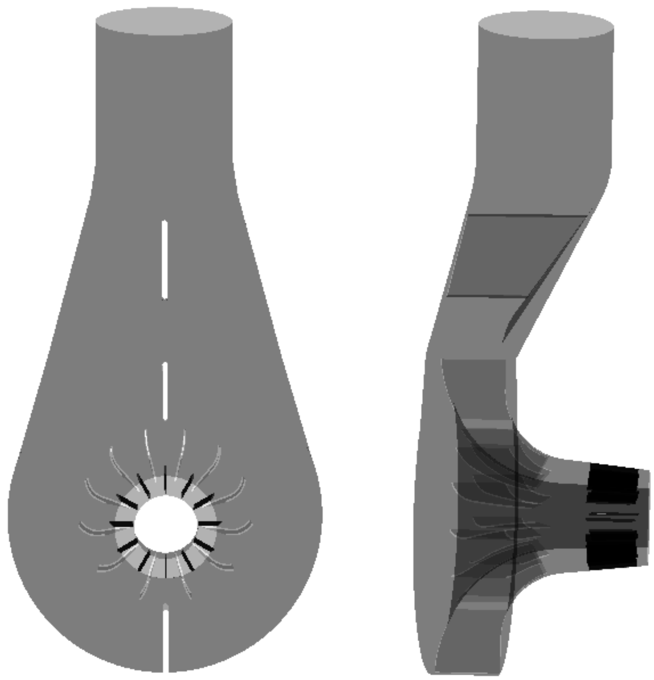

Figure 1. The geometry of radial inlet 1 (front and side view)

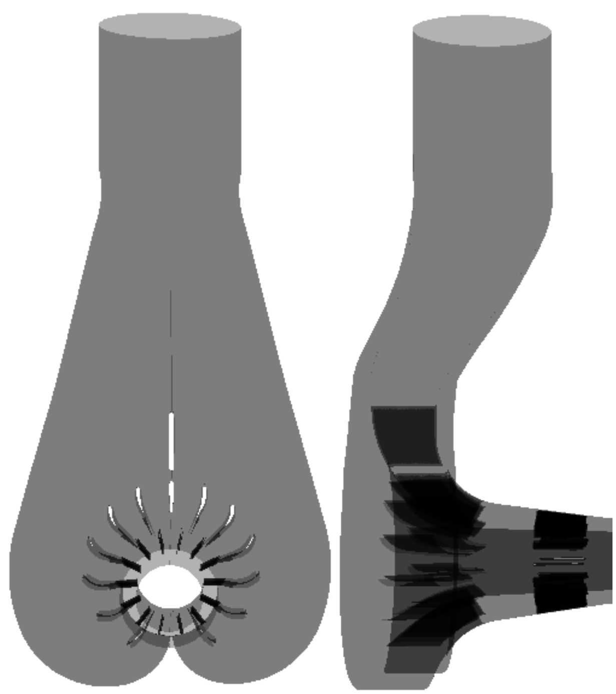

Figure 2. The geometry of radial inlet 2 (front and side view)

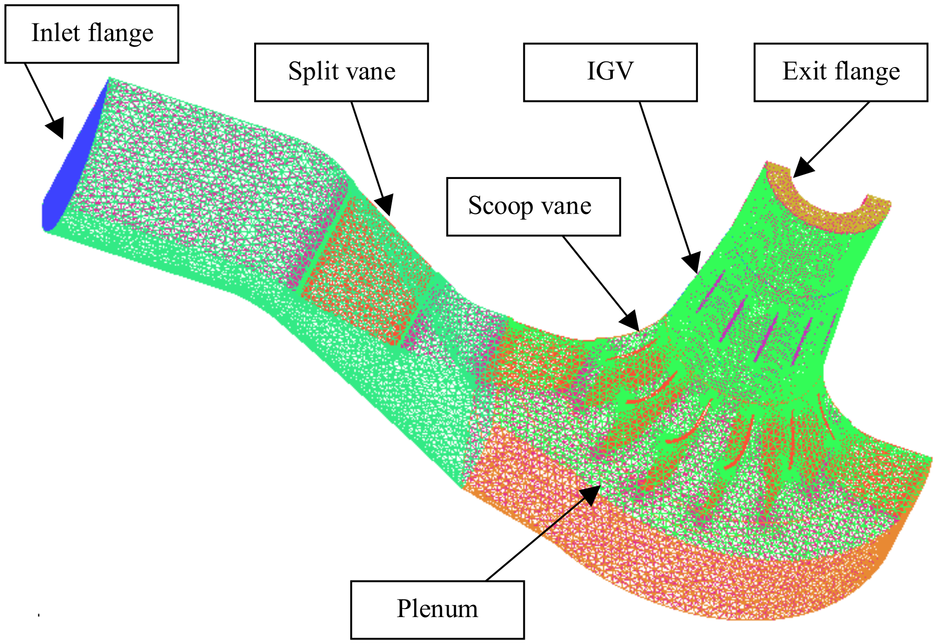

Figure 3. Tetrahedral mesh on half of geometry of radial inlet 1

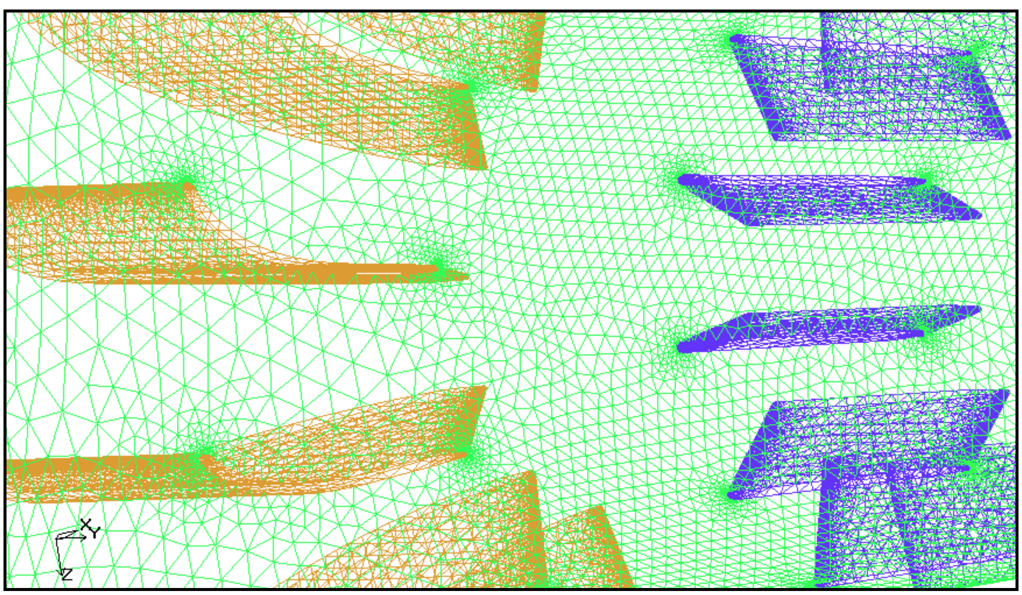

Figure 4. Leading and trailing edge of vanes with small tetrahedral elements

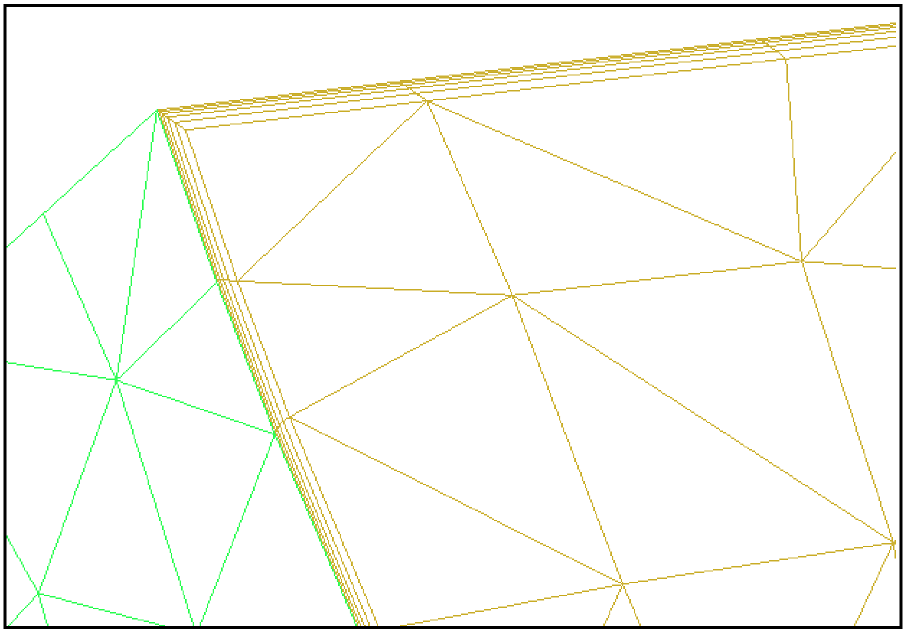

Figure 5. Imbedded prism layers on the wall of inlet models

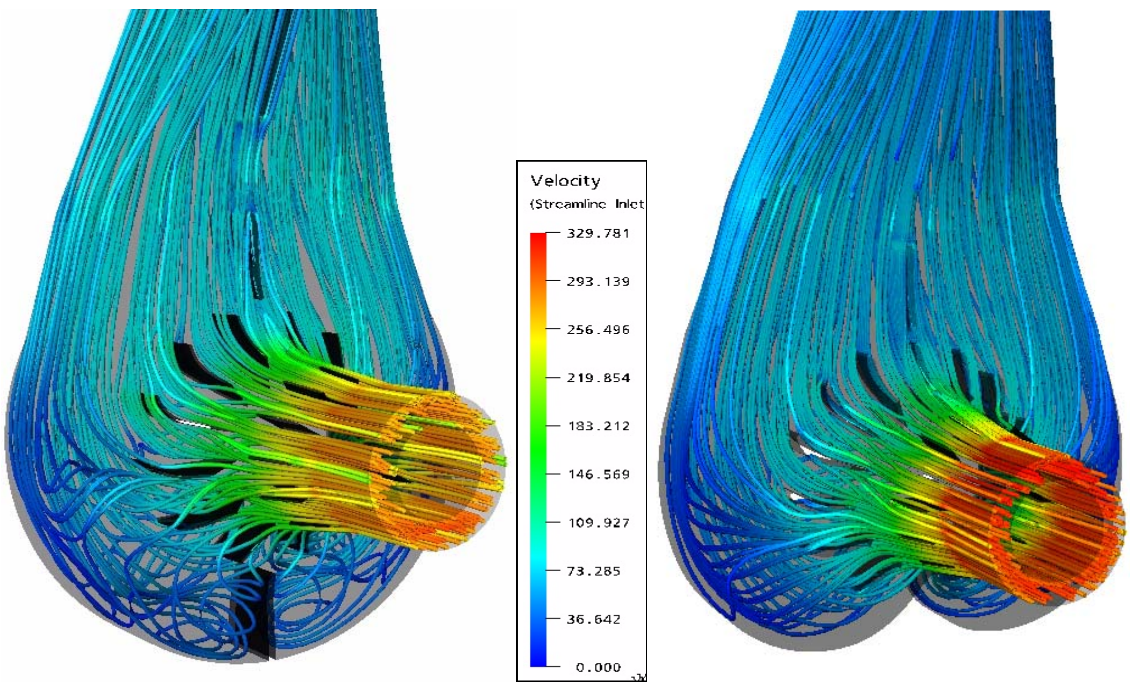

Figure 6. Streamline of radial inlet 1(left) and 2(right) @ inlet Mach number 0.05

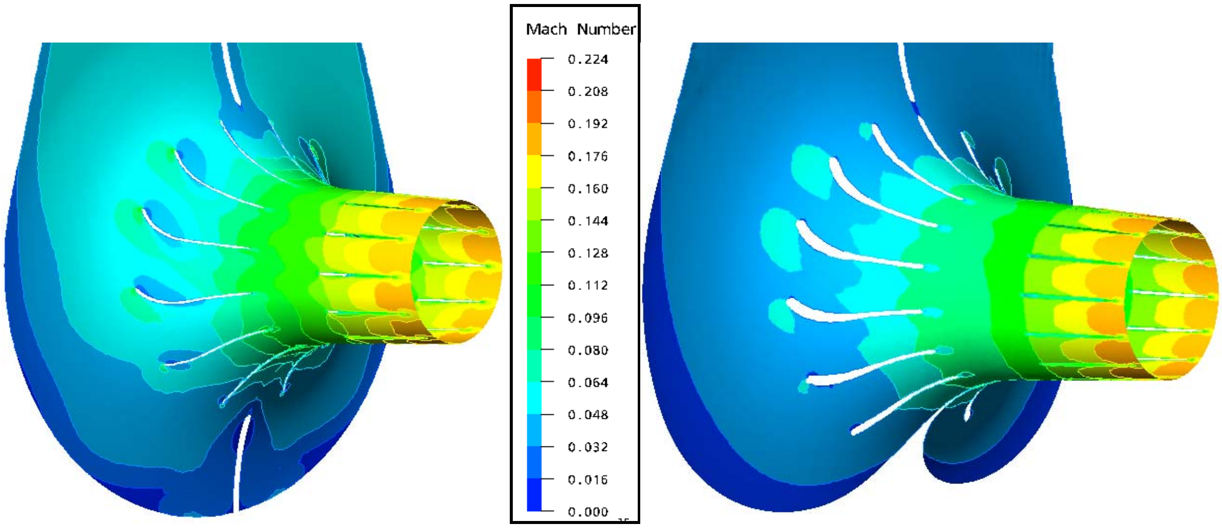

Figure 7. Mach number of radial inlet 1(left) and 2(right) @ inlet Mach number 0.05

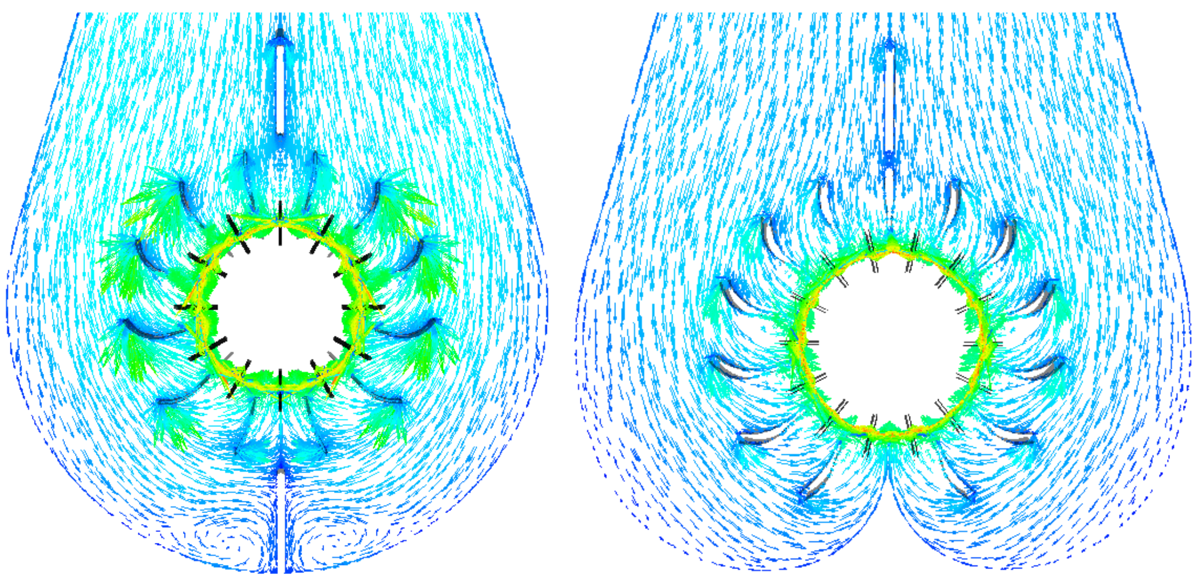

Figure 8. Vector plot of radial inlet 1(left) and 2(right) @ inlet Mach number 0.05

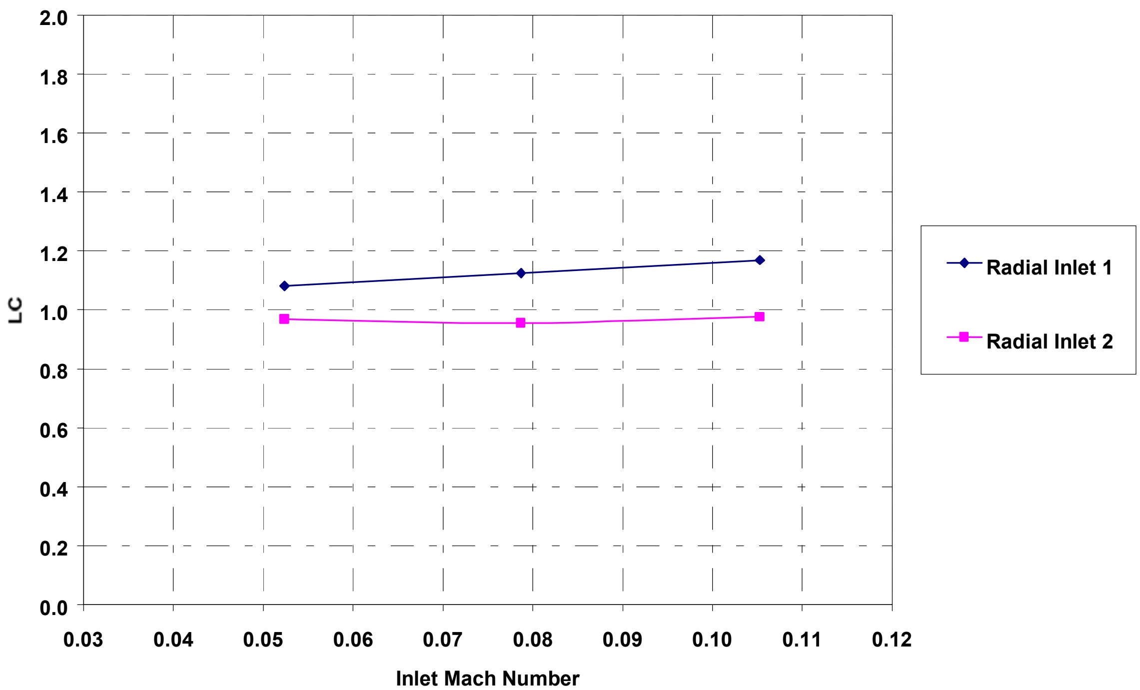

Figure 9. Total pressure loss coefficient across the flow passage

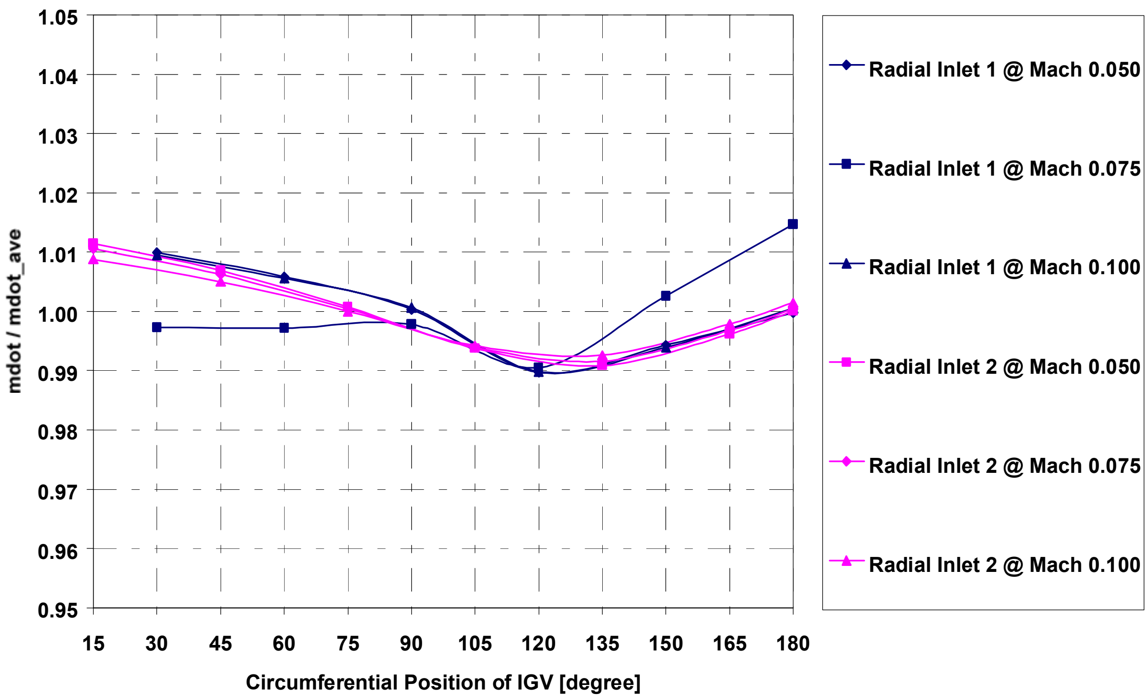

Figure 10. Normalized mass flow distribution @ various inlet Mach no.

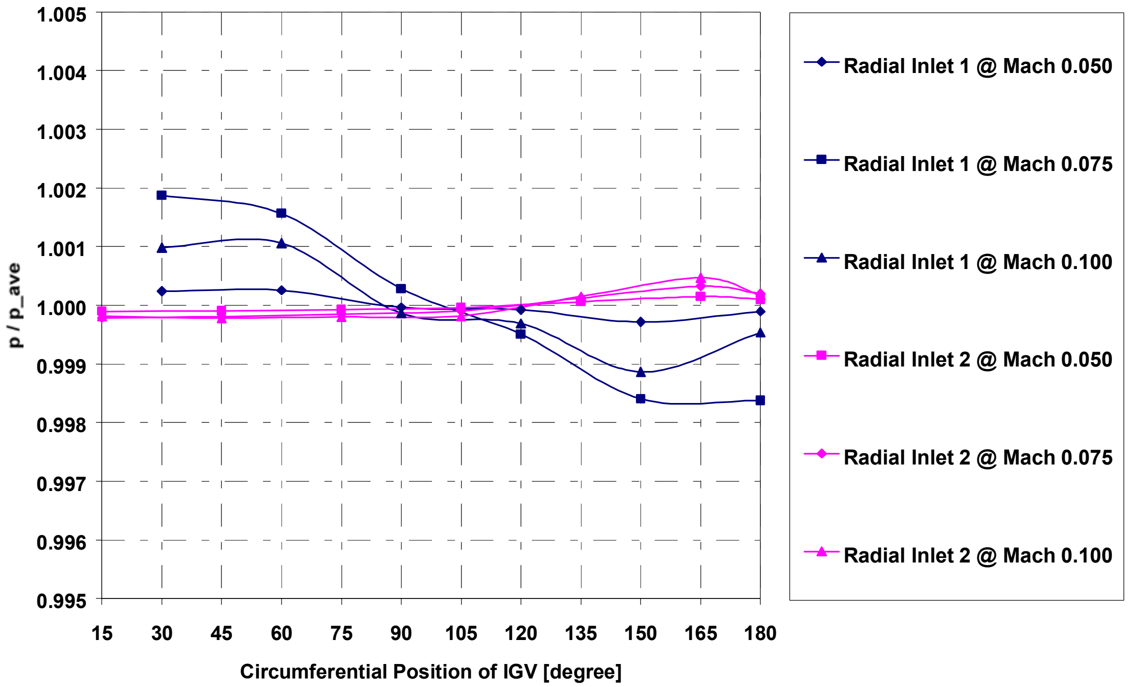

Figure 11. Normalized pressure distribution @ various inlet Mach no.

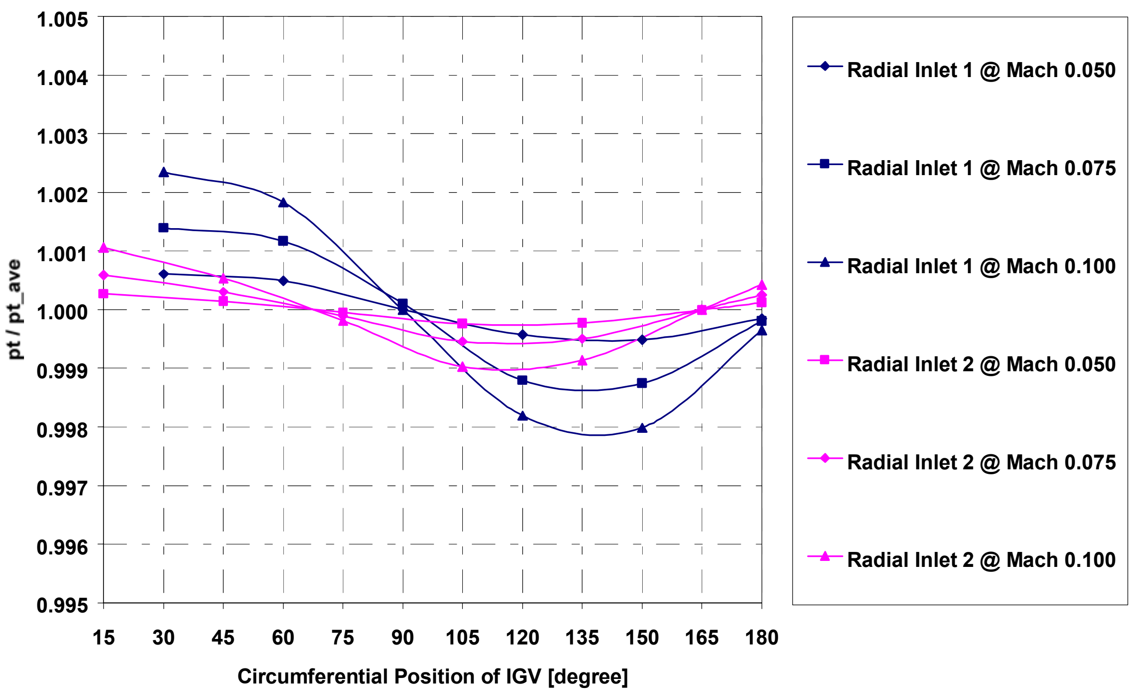

Figure 12. Normalized total pressure distribution @ various inlet Mach no.

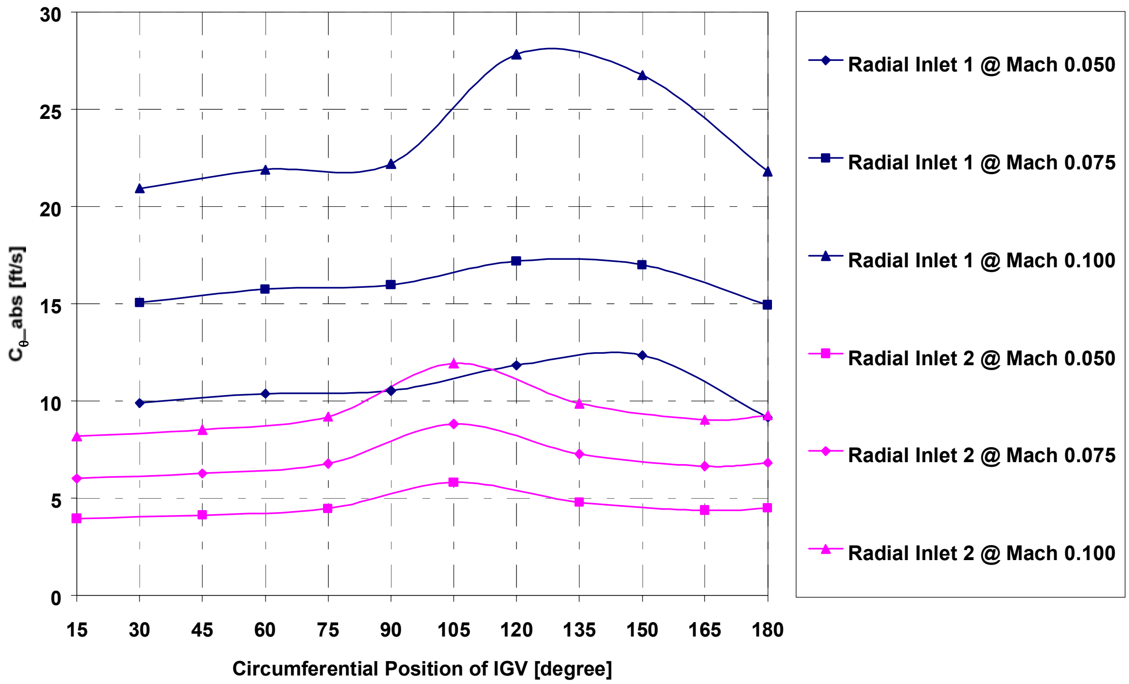

Figure 13. Absolute tangential velocity distribution @ various inlet Mach no.

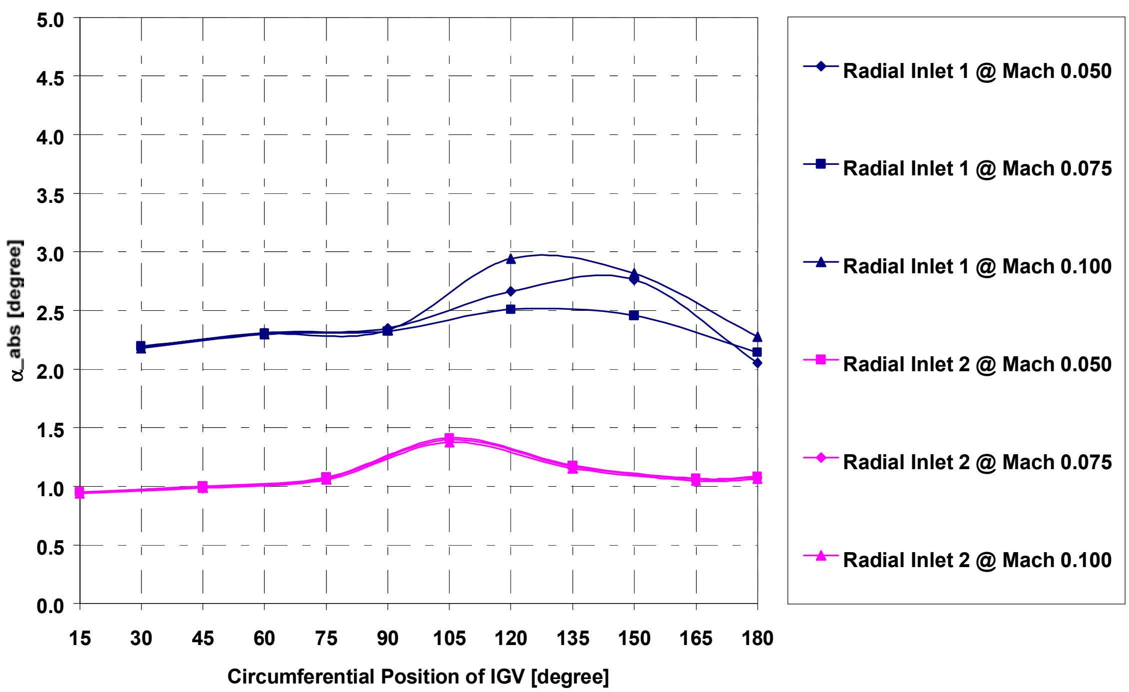

Figure 14. Absolute flow angle distribution @ various inlet Mach no.

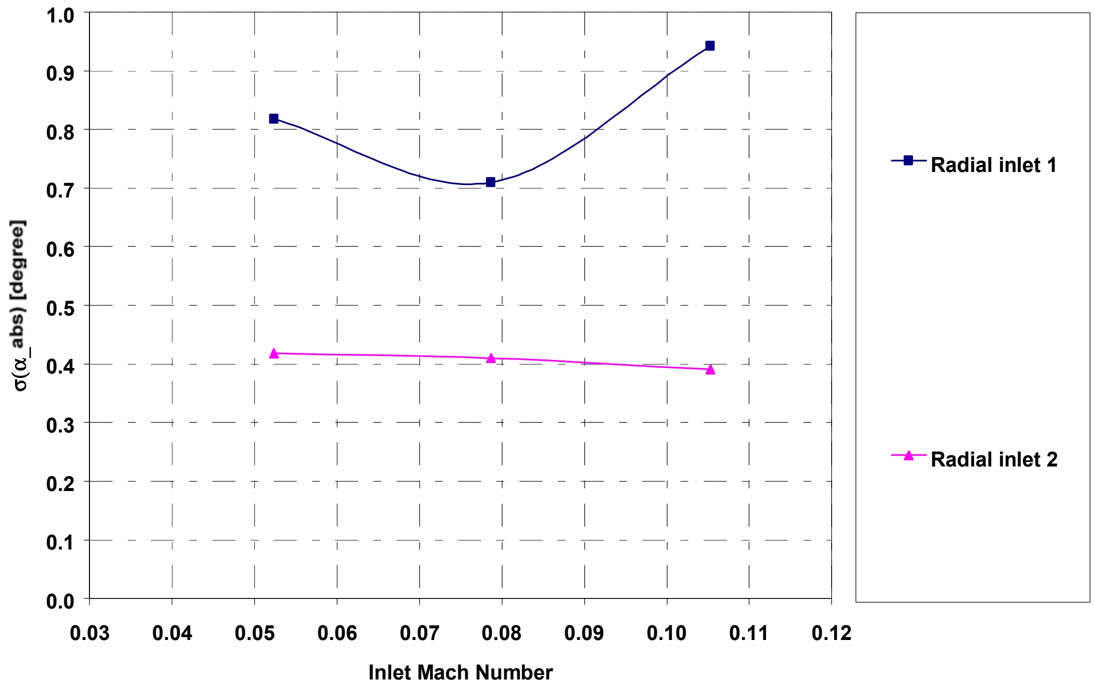

Figure 15. Standard deviation of absolute flow angle @ various inlet Mach no.