Issues Surrounding Multiple Frames of Reference Models for Turbo Compressor Applications

Z. Liu
Dresser-Rand

D. L. Hill
Dresser-Rand

Follow this and additional works at: https://docs.lib.purdue.edu/icec


This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information. Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at https://engineering.purdue.edu/Herrick/Events/orderlit.html
ISSUES SURROUNDING MULTIPLE FRAMES OF REFERENCE MODELS FOR TURBO COMPRESSOR APPLICATIONS

Zheji Liu and D. Lee Hill
Advanced Aero Development
Dresser-Rand
Paul Clark Drive
Olean, NY 14760
E-Mail: zheji_liu@dresser-rand.com

ABSTRACT

A centrifugal compressor system is often modeled by assigning different frames of reference to individual rotating and stationary components. For a relative frame of reference, additional terms accounting for the coriolis and centrepredal forces are required in the momentum equations. Also needed is a suitable interface model between the impeller and surrounding stationary components. In this study, three different techniques are used to model various configurations of a turbo compressor. They are the Frozen Rotor model, Circumferential Average model, and the Transient Sliding Mesh model. The first two models allow for a steady-state approximation. All three approaches give different results when the models include non-axisymmetric components and when strong interaction occurs between the rotating and stationary parts. A thorough investigation is presented and reasons offered for the distinctly different results. The motivation for this effort is derived from inability to consistently correlate predicted performance values obtained from using the steady state models with test data. This is especially true for off-design operation and for models that have varying blade periodicity.

INTRODUCTION

As industrial manufacturers strive to reduce design cycle time and their dependence on prototype testing, more efforts are being spent on developing numerical models that physically describe the behavior of their hardware. This is especially true for manufacturers producing large machinery because of the excessive high construction costs for one-of-a-kind hardware and because of the long lead times required for materials. These limitations are forcing the OEM to only test to validate not to investigate. Under these circumstances, design engineers are no longer able to rely on just past experience while using traditional 1-Dimensional empirical based tools. They have to get the design correct the first time. This requirement also applies for diagnosing aerodynamic problems occurring in machines already installed in the field. If there is a problem, the machine down time is normally very limited, in order to minimize the customer’s production loss. Consequently, there is often little opportunity for excessive testing.

The use of Computational Fluid Dynamics (CFD) offers a cost-effective way to design and investigate fluid flow hardware. The success of this type of calculation depends on both the user and the models in the code. In addition to the traditional topics such as turbulence model or numerical schemes, turbomachinery problems have an additional problem of handling the rotating and stationary components. Historically, researchers have either analyzed the component separately [1-2] or patched the components together achieving a one way coupling [3].

A centrifugal compressor stage has both rotating and stationary components, which are coupled together in a CFD model by one or two interfaces. The multiple frames of reference (MFR) technique is one such interface model [4-5]. The rotating component, such as the impeller in the centrifugal compressor, is modeled in a rotating frame of reference, while the stationary components are assigned to a stationary frame of reference. An interface is applied at each junction where the change of the frame of reference takes place. There are two types of interface techniques to exchange the information between the different frames of reference. The first type of interface is called Circumferential Averaging, where the upstream flow velocity profile is first averaged circumferentially before transferring the information to the downstream region or frame of reference. This method assumes the flow going to the downstream inter region is steady and axisymmetric. Since it circumferentially averages the values at the interface before imposing them on the neighboring reference frame, any upstream flow non-uniformity or distortion in the circumferential direction will not be preserved in the next inter region. Note that a hub to shroud velocity distribution is allowed for some models.

The second type of interface implemented in the MFR analysis of the steady state CFD simulation is called Frozen Rotor. The flow profile variation in the circumferential direction is now “preserved” across the interface. However, the relative position between the two components mode' d in the inter frames of reference is fixed in...
time and space, so this interface transfers the non axisymmetric flow distribution developed only at the given relative position between the rotor and the stationary components to the neighboring region. Any circumferential flow distribution change due to the variation of the relative position between the two involved components is not considered in this interface.

The Circumferential Averaging and Frozen Rotor interface techniques have been used to couple portions of the flow path. The work of [6-7] are good examples. Most recently, Liu et al. [8] used the Frozen Rotor interface to model a pipeline compressor with Low Solidity Vanes (LSV). They used 360 degree flange to flange CFD model of 1.5 millions nodes was built to study the compressor performance at different operating conditions. Good agreement in physical trends was obtained between their CFD results and the experimental data. Based upon what they learned from the CFD analysis and test data of the existing design of the pipeline compressor, Biba et al. [9] redesigned the compressor with better performance and lower noise level. Biba et al. [10] also applied the Frozen Rotor interface in a series of CFD analysis to design and improve a low volume flow and high pressure compressor stage. They successfully designed a new compressor stage with improved efficiency. Although their CFD results tend to over-predict the performance, qualitative agreement between the CFD results and test data was satisfactory. Moore and Hill [11] also applied a similar approach in the design improvement of a novel swirl brake. Recently, a field compressor performance problem was successfully solved by Liu et al. [12] mainly through CFD analysis. This work was successful because the design process took into account of the interface model biasing of the solutions.

The operation of a centrifugal compressor is inherently an unsteady process. The aerodynamic interaction between the rotating part and the stationary parts is an important contributor to the unsteadiness of the flow present in the centrifugal compressor. Neither of the two interfaces implemented in the steady state CFD analysis is capable of predicting the unsteady effects resulted from the rotor-stator interaction due to their relative position change. A third type of interface, the transient sliding mesh interface, is available to simulate the fluid motion caused by the relative movement between a rotor and stationary components in turbomachinery. In this approach, a sliding interface is used between the moving mesh of the rotor and the non-moving mesh of the stationary parts. During such a transient solution process, the moving mesh is made to slide past the stationary one by a certain degree during each time step according to the defined rotational speed and the time step size, and the information exchange continuously across the sliding interface. The flow field variation in both time and space, specifically in the circumferential direction, due to the unsteady aerodynamic interaction and the coupling effects between the rotating component and the stationary ones, is fully taken into account in the transient sliding mesh methodology.

As parallel processing matures and the computing costs continue to drop, the computationally intensive transient CFD has become possible outside of government laboratories. Rai [13] solved the unsteady Navier-Stokes equations to study the turbine rotor-stator interaction. A similar study of the same problem with grid refinement was performed by Madavan and Rai [14] to study the rotor-stator interaction in an axial turbine stage. The turbine stage geometry was modified to a single-stator and a single-rotor airfoil combination for the sake of reducing mesh size and computing time. The results from both studies are in good agreement with experimental data. Cizmas et al. [15] numerically investigated the effects of the inter-stage gap size on the turbine efficiency by solving the unsteady Euler/Navier-Stokes equations. Richman and Fleeter [16] studied the unsteady aerodynamics in an axial transonic compressor. They predicted the compressor performance map with very close match to the measured data but over-predicted the IGV steady surface pressures and under-predicted the magnitude of the unsteady component. All these unsteady CFD studies were on axial turbomachines. The geometry of the CFD models used in these studies are only a slice segment in the axisymmetric direction. This certainly reduces the computing time and cost but does not have the full extent of the interaction considered.

None of the above studies offers a comparison between all three models. It is beneficial to know the advantages and disadvantages of the approaches in modeling the real world turbomachinery problems. When the CFD solutions look unphysical, people may blame the turbulence models, the grid density and quality of the model, the numerical schemes, and etc. However, this paper also demonstrates that discreetly choosing the type of interface between the moving component and the stationary components is also important in modeling turbomachinery fluid problems. With everything else kept the same, just changing the interface in the CFD model can cause difference in the results.

APPLICATION OF INTERFACES IN TURBOMACHINERY CFD MODELS

All the CFD results presented in this section were obtained from either TASCflow or STAR-CD by solving the Navier-Stokes equations. The flow is considered as turbulent and compressible. The fluid is assumed to be
Newtonian and the ideal gas equation with the right mole weight or gas constant is used in the models. The standard $k-\varepsilon$ turbulence model with wall functions was used in the computations.

A CFD model of a centrifugal compressor stage typically needs two interfaces, one to couple the stationary inlet region to the rotating impeller and a second one to couple the rotating impeller to the downstream stationary components such as diffuser, return bend, and return channel or volute, as depicted in Figure 1. The interfaces applied before and after the impeller are addressed separately in the following.

**Impeller Inlet Interface**

Two types of inlet configuration are commonly utilized in industrial centrifugal compressors. The first type of inlet uses an axial duct to feed gas directly to an impeller and is called the Direct Inlet. All the Dresser-Rand PDI machines for the pipeline market have this type of inlet. Studies done on this type of machines can be found in references [8-9]. Since the gas enters the impeller without turning, the pressure loss and the flow distortion in the inlet are insignificant. Inlet guide vanes (IGV) are generally not required in this case and this makes the flow even cleaner. Consequently, there is not much interaction between an impeller and an axisymmetric direct inlet. It was found that either the Circumferential Averaging interface or the Frozen Rotor interface is adequate for this type of application. The transient sliding interface may be an overkill in this case since the typical rotor-stator interaction phenomenon is not strong at all.

Another type of inlet used in industrial centrifugal compressors feeds gas radially to an impeller. When an inlet duct in a compressor is perpendicular to the axis of the impeller, the package restraint requires a radial inlet to "bridge over". Since a radial inlet is non-axisymmetric, as shown in Figure 1, the inlet flow distribution variation in both the circumferential direction and radial direction is difficult to prevent. Typically, inlet guide vanes are installed in the radial inlet to guide the flow to the impeller, and this further enhances the rotor-stator interaction phenomenon, especially when IGV trailing edges are close to the impeller. Under this kind of circumstances, the choice of the interface type and interpretation of the solutions resulted from the CFD model using a specific type of interface are very important to get useful information from the analysis.

In order to find out the effect of the interface on the CFD results, all the three types of interface were individually applied to the same CFD model, with everything else kept the same in the model. Figure 2 compares the impeller velocity magnitude contour computed by the CFD models using those three types of interface. The circumferential flow variation resulted from the upstream non-axisymmetric radial inlet causes the flow in the impeller passages to be non-uniform in the Frozen Rotor solution. The Circumferential Averaging model predicted a more axisymmetric flow field in the impeller passages since it averaged out the circumferential flow variation from the upstream inlet. The same velocity contour plot computed by the transient sliding mesh model is shown in Figure 2(c). Since the flow is transient, a time averaging process was performed on the transient CFD results stored at many time steps to obtain the time averaged flow field. Comparing the three plots of speed contour at the same location of the impeller gives the impression that the Frozen Rotor CFD model over-predicts the flow variation in the circumferential direction. Both the Circumferential Averaging model and the transient sliding mesh model show that the flow variation from impeller passage to passage is not significant and the effect of the circumferential non-uniform flow distribution resulted from the radial inlet on the flow distribution in the impeller passages is small.

The interface used to couple the radial inlet to the impeller also affects the predicted flow field in the upstream radial inlet. A comparison of the gas velocity distribution at the exit of the radial inlet was made in Figure 3. The Circumferential Averaging model predicts that the flow exiting the inlet is symmetrical about the symmetry plane of the inlet geometry, which agrees well with the result from the inlet alone CFD model. The downstream rotating impeller has little influence on the circumferential flow distribution in the inlet according to the Circumferential Averaging CFD result. However, the transient CFD result shows that the flow leaving the symmetrical radial inlet is not purely symmetrical due to the effect of the rotating impeller. Another difference in the two plots is the location of the high velocity regions. The transient sliding mesh CFD result shows that the maximum speed occurs to the vicinity of the 90 degree position while the Circumferential Averaging model predicts that high-speed gas streams exit at both the 90 degree and 180 degree locations.

**Impeller Discharge Interface**

An interface is normally needed to couple a rotating impeller to a diffuser in the CFD analysis of an industrial centrifugal compressor. There are two types of diffuser commonly used in centrifugal compressors, the vane diffuser and the vaneless diffuser. For a compressor with a vaneless diffuser, the type of interface used between the impeller and the vaneless diffuser in the CFD model is not very critical since the coupling effect between the impeller and the vaneless diffuser is not strong. Both the Circumferential Averaging and the Frozen...
Rotor interfaces give similar results. The Frozen Rotor interface is preferred by the authors in this kind of application, as this can be seen in reference [8-9].

The insertion of discharge vanes in the diffuser of a compressor makes the flow in the diffuser more dynamic and unsteady because of the rotor-stator interaction phenomenon. The intensity level of the interaction is influenced by the gap distance between the impeller tips and the leading edges of the discharge vanes. The smaller the gap is, the stronger the interaction is. It was found in reference [12] that the reduction of the gap distance makes the choice of the interface type used in the CFD model very critical. CFD studies of two different gap distance are presented in this study.

The CFD results presented in Figure 4 represent the baseline diffuser design of a field compressor manufactured by Dresser-Rand. There are 15 Low Solidity Diffuser (LSD) vanes in the diffuser, with a gap distance of 15.3% of the impeller diameter. The Frozen Rotor interface was first used in the CFD model and the resulted velocity vector plot and the Mach number plot are shown in Figure 4(a). The Frozen Rotor interface was then replaced by the Circumferential Averaging interface and the CFD results were shown in Figure 4(b). Comparison of the flow filed resulted from both types of interface indicates that the difference is small.

The diffusion from the original LSD diffuser design of the field compressor was found to be insufficient in reference [12]. One of the design iterations was to increase the number of LSD vanes from 15 to 17 and to enlarge the LSD vanes, as shown in Figure 5. The gap distance between the impeller and the LSD vanes was reduced to 12.35% of the impeller diameter. The modified diffuser was remodeled and both the Frozen Rotor interface and the Circumferential Averaging interface were used, respectively, in the new CFD model. Comparing the solutions presented in Figure 5 shows big difference in the two solutions. The flow field resulted from the Circumferential Averaging interface is much better behaved than the solution from the other interface. A small recirculation zone near the suction side of the LSD vane trailing edge is predicted by the Frozen Rotor model. To clarify the difference between the two models, the third CFD model using the transient sliding mesh method was built and run. The time averaged velocity vector and Mach number plots were presented in Figure 5(c). The gray scale used in Figure 5(c) is opposite to the gray scale used in plots 4(a) and 4(b) because they were created by different software packages. Similar to the Circumferential Averaging CFD result, the transient CFD model does not predict any recirculation zone in the diffuser. This is another demonstration that the Frozen Rotor solution tends to over-predict the non-uniformity of the flow field.

CONCLUSIONS

Three types of interface to couple a rotating impeller to its neighboring components were discussed and compared in this study. Before choosing which interface to be used in a centrifugal compressor CFD model, one has to be aware of the coupling effect between the rotating part and the stationary parts. When the coupling effect is weak, all three interfaces give similar results. However, the results deviate as strong aerodynamic interaction occurs to the coupled rotating and stationary parts. Two such cases were presented in this study. In the first example, it is mainly the circumferential flow variation resulted from the radial inlet that causes the differences among the three interface models. In the second case, the decrease of the gap distance between the impeller and the LSD vanes enhances the rotor/stator interaction effect and causes the Frozen Rotor model to predict the flow field to be very different from the other solutions. In both cases, the Frozen Rotor interface tends to over-predict the non-uniformity of the flow field in the down stream inter region. The Circumferential Averaging CFD results are more similar to the solution of the transient sliding mesh models than the Frozen Rotor CFD results are. For steady state CFD simulations, the Circumferential Averaging interface should be more often used since the Frozen Rotor interface may give misleading information under special cases. Although the transient sliding mesh method is computationally intensive, it is the necessary approach to predict the inherently unsteady flow field of a centrifugal compressor. Only the transient sliding mesh CFD is capable of simulating the aerodynamic interaction due to the impeller rotation relative to either upstream inlet guide vanes or downstream discharge vanes.

REFERENCES

Figure 1 – CFD model of an industrial centrifugal compressor
Figure 2 – Velocity magnitude inside impeller

(a) Frozen Rotor  (b) Circumferential averaging  (c) Transient sliding mesh

Figure 3 – Velocity contour plots at the outlet of a radial

(a) Transient sliding mesh  (b) Circumferential averaging
(a) Frozen Rotor solution

(b) Circumferential Averaging solution

Figure 4 – Velocity vector and Mach number plots in a LSD diffuser
Figure 5 – Velocity and Mach number plots in a LSD diffuser with shorter gap distance from the impeller