Innovative Diagnosis for Instability in Turbomachinery

Simulation helps to predict subsynchronous vibrations and rotordynamic stability for centrifugal compressors.

By J. Jeffrey Moore, Program Manager (Rotordynamics Program), Rotating Machinery & Measurement Technology Section, David L. Ransom, Senior Research Engineer, Rotating Machinery & Measurement Technology Section and Flavia Viana, Research Engineer, Fluid Dynamics & Multiphase Flow Section, Southwest Research Institute, Texas, U.S.A.

The energy industry — particularly the natural gas and hydrocarbon segments — depends on centrifugal compressors to produce, process, liquefy and transport many different gases. As the pressures in a compressor increase, the dynamic behavior at shaft and impeller seals, axial thrust balance pistons and impellers becomes more complex, with vibration ultimately becoming a concern.

There are two types of vibration of concern in industrial compressors: synchronous vibration and subsynchronous vibration. Synchronous, or running-speed vibrations, normally are excited by residual unbalance resulting from small imperfections in the manufacturing and assembly processes. The second and more troubling type of vibration, subsynchronous vibration, occurs when non-conservative whirling forces (cross couplings) act to excite a lateral natural frequency, which occurs in cases in which these fall below running speed. The excitation forces generated at seals and impellers have components that act at right angles to the displacement vector. Cross-coupling effects tend to sustain whirling motion at a subsynchronous natural frequency when insufficient damping is present. The whirling motion is referred to as self-excited rotordynamic instability, and it can lead to serious damage if not properly controlled. Researchers at Southwest Research Institute (SwRI), a nonprofit applied engineering research and development organization headquartered in the United States, have examined the possibility that computational fluid dynamics (CFD) can be used to study subsynchronous vibrations and rotordynamic instability for centrifugal compressors.

A CFD analysis by Moore and Palazzolo[1] used a grid perturbation method (GPM) approach with a 3-D structured computational mesh to demonstrate how cross-coupled stiffness for liquid (incompressible) pump impellers could be determined. Gas forces in a compressible fluid tend to be smaller, more difficult to predict and more difficult to model than the liquid forces that were predicted by Moore and Palazzolo. Furthermore, the energy equation and an equation of state are required to completely describe the fluid flow. A lack of accurately predicted operational specifications for compressor designs can result in unexpected, dangerous, and damaging instabilities and subsynchronous vibrations, making the identification of accurate analysis methodologies essential to the industry.

Description of Computational Model

A compressor manufacturer provided SwRI with the complete geometry, process and rotordynamic information for a centrifugal compressor that previously had experienced subsynchronous vibrations. The centrifugal compressor was equipped with only four out of 10 stages and had undergone development testing a number of years prior. While testing at a speed of 21,500 rpm and a discharge pressure of 2,300 psi using nitrogen as the testing medium, the compressor encountered classical rotordynamic instability. The frequency corresponded to the first natural frequency of the rotor. A second instability was reached while operating at 23,000 rpm. Engineers at SwRI identified this particular compressor as suitable for a case study because the impeller aerodynamic cross coupling was the dominant effect on the machine's stability. Because the exact conditions at which the compressor went unstable were available from test records, the CFD could be tested under the same conditions.

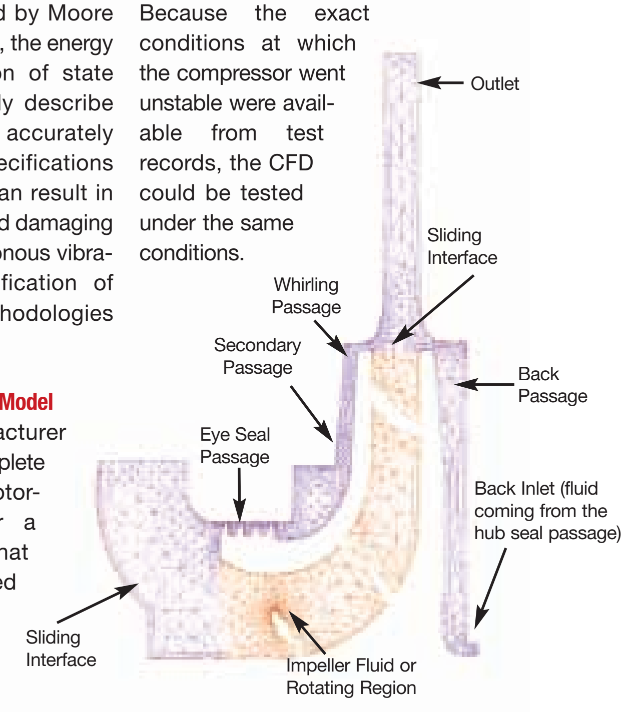

Figure 1. The compressor mesh with the boundary condition surfaces and sliding interfaces

To generate a complete CFD model of this impeller — including both the primary and the secondary flow around the impeller/diffuser, shroud, back face and seals — the engineering team used ANSYS CFX technology. The shroud was displaced in the radial direction. Although the physical problem appeared to be inherently time dependent, a transient CFD solution of this problem was not required if a simple reference frame transformation was performed. Since the shroud region was solved in the whirling frame of reference, while the primary impeller passage was always solved in the rotating frame, a sliding interface was employed. Researchers chose the frozen rotor sliding interface approach exclusively, so as not to artificially constrain the circumferential pressure field. The rotordynamic influence of the labyrinth seal was modeled in a rotordynamics model using a traditional bulk flow seal code.

Researchers evaluated the rotordynamic force coefficients of the impeller by determining the impedance at a minimum of three precessional frequencies. For improved accuracy over a wide range of precessional frequencies, more than three would need to be calculated and a least-squares curve fit to the linear second-order model was performed. The coefficients of the curve fit would yield the impeller's stiffness, damping and mass force coefficients.

Validation of Results

In this stage of the project, the team validated the results from the CFD and rotordynamic analyses using real-world data. They used two separate verification methods for the SwRI CFD model for impeller force work performance. In the first test case, researchers essentially reproduced the results of Moore and Palazzolo[1], though they used an unstructured mesh. This case demonstrated good correlation to previous predictions and experiment, validating the use of an unstructured grid. In the second verification, the team required a comparison of CFD-based stability predictions for a centrifugal gas compressor against measured subsynchronous vibrations on the test compressor.

Overall, SwRI engineers found the CFD results to be in reasonable agreement with the performance data. The flow field in the secondary passage was highly recirculating. Using a second-order curve fit, the full set of force coefficients was computed. Since the team performed a CFD analysis on only stages one and three, normalized parameters were used to calculate the coefficients for stages two and four. These derived force coefficients were close to the CFD values, validating the method used. Researchers also performed a rotordynamic analysis to analytically determine total dynamic behavior of the rotor at high rotational speeds and the stability of the compressor rotor — including the effects of rotor flexibility, bearing stiffness and damping, eye seal stiffness and damping, balance piston stiffness and damping, and aerodynamic excitation.

Engineers analyzed two compressor instability cases: instability point one (21,500 rpm) and instability point two (23,000 rpm). Even though the speed increased for point two, the discharge pressure at the point of instability was approximately the same. Therefore, the predicted rotordynamic stability varied only slightly between the two conditions. For each case, the aerodynamic cross coupling was varied from 0 to about 25,000 lbf/in to define the slope of the stability curve. The point at which the lines intercepted the vertical axis represented the system stability without the effects of aerodynamic cross coupling. The point at which the lines crossed the horizontal axis was the stability threshold, beyond which the machine was predicted to be unstable. These two lines provided insight into the sensitivity of the rotordynamic stability as a function of aerodynamic cross coupling (Figure 6).

The CFD results showed much-improved agreement in overall magnitude in comparison to the application programming interface API and SwRI methods, which are empirically based equations that have been used in the industry for many years. CFD predicts similar levels of cross coupling for the two instability points, while the empirically based methods do not.

The SwRI team then performed a parametric study to determine the various parameters known to affect the flow field inside the impeller and their effect on rotordynamic forces. Based on this

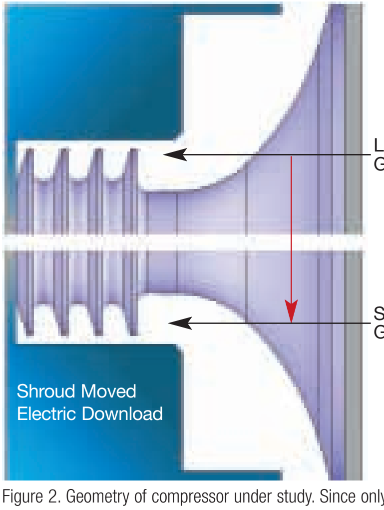

Figure 2. Geometry of compressor under study. Since only shroud forces are of interest in this study, only the shroud region is made eccentric.

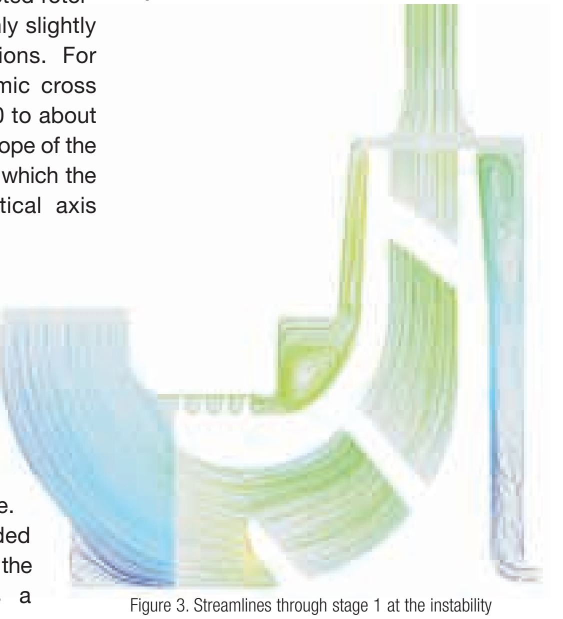

Figure 3. Streamlines through stage 1 at the instability point #1, 21,500 rpm

study, they developed a new formula to describe impeller cross coupling. The formula stated that the cross coupling was proportional to the dynamic pressure and the axial length of the impeller, and inversely proportional to relative flow due to the change in the exit flow angle of the impeller, as shown below.

$$K_{xy} = \frac{C_{mr}\,\rho_{dis}\,U^2\,L_{shr}}{Q/Q_{design}}$$

in which

K_{xy} = cross-coupled stiffness of impeller (lb/in) [N/m]

C_{mr} = constant for a given impeller design

ρ_{dis} = discharge density (lbm/ft^3) [kg/m^3]

U = impeller tip speed (ft/s) [m/s]

L_{shr} = axial length of shroud from impeller eye seal to impeller tip (in) [m]

Q/Q_{design} = flow relative to design flow

Subsequent studies have indicated that C_{mr} can vary for different impeller geometries, and it is typically in the range of 4 to 7.5.

As demonstrated in this study, the SwRI engineering team was the first to develop analytical methods capable of analyzing the rotordynamic forces on a centrifugal compressor impeller using CFD. The results compared favorably when predicting the instability of a full-scale compressor. Based on this result, the team concluded that the majority of the destabilizing force of a centrifugal impeller arises from the shroud passage, not the impeller-to-diffuser interaction. These results are described in more detail in Moore, Ransom and Viana[2]. ■

References

[1] Moore, J.J., Palazzolo, A.B., "Rotordynamic Force Prediction of Centrifugal Impeller Shroud Passages Using Computational Fluid Dynamic Techniques with Combined Primary/Secondary Flow Model," *Journal of Gas Turbines and Power*, Vol. 123, October 2002, pp. 910–918.

[2] Moore, J.J.; Ransom, D.L.; and Viana, F. "Rotordynamic Force Prediction of Centrifugal Compressor Impellers Using Computational Fluid Dynamics," GT2007-28181, ASME Turbo Expo, May 14–17, 2007, Montreal, Canada.

STAGE 1		CFD Value	Applied Cond.	Difference
Compressor Speed	[rpm]	21,500		
Pressure Ratio		1.238	1.264	2.06%
Delta Tot. Temperature	[F]	40.36	48.10	16.09%
Polytropic Efficiency (from Power)		0.746	0.786	5.10%
Isentropic Efficiency		0.830	0.779	6.56%
Isentropic Efficiency (from Power)		0.736	0.779	5.40%

STAGE 3		CFD Value	Applied Cond.	Difference
Compressor Speed	[rpm]	21,500	21,570	
Pressure Ratio		1.234	1.234	0.02%
Delta Tot. Temperature	[F]	47.60	52.30	9.00%
Polytropic Efficiency (from Power)		0.734	0.763	3.80%
Isentropic Efficiency		0.815	0.755	7.87%
Isentropic Efficiency (from Power)		0.713	0.755	5.66%

Figure 4. Summary of compressor performance for compressor stages 1 and 3

Method	Instability Pt 1	Instability Pt 2
SwRI	18,453 lbf/in	23,441 lbf/in
API	13,848 lbf/in	17,301 lbf/in
CFD	12,098 lbf/in	11,908 lbf/in

Figure 5. Comparison of modally weighted aero cross-coupling values using the various prediction methods

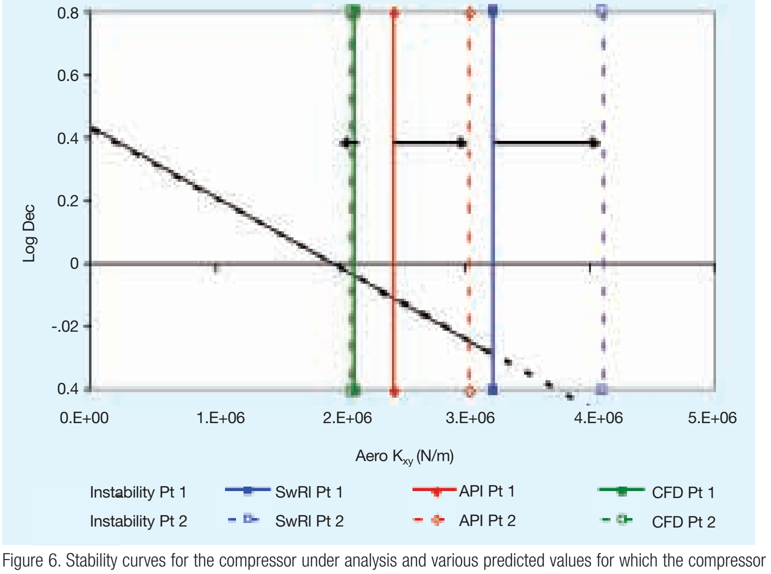

Figure 6. Stability curves for the compressor under analysis and various predicted values for which the compressor would become unstable (i.e., prediction values for where the curves would cross the x-axis of this plot; the CFD results provide the most accurate prediction).