Abstract
Experimental investigations on a single stage centrifugal compressor showed that asymmetric suction elbows can cause unacceptable blade vibrations. For turbocharger applications it is important to assess this issue in an early design phase. Although computational methods are known to estimate the influence of the asymmetric flow structure on referring blade vibration resonance by means of unsteady CFD and FE vibration analysis, those are not suitable for the design process, as they are too time and CPU consuming.

Thus, the focus of this project is to define a procedure which determines the blade excitation by simplified models that can easily be integrated into the compressor design loop.

For the evaluation of blade excitation due to asymmetric incoming flow, the flow fields at the inlet of the impeller using calculation methods of varying accuracy are compared. The potential energy of the harmonic blade excitations at the inlet of the impeller at different axial and radial positions is determined and evaluated by Fast Fourier Transform analysis over the circumference.

The flow field of the unsteady calculation acts as a reference, since it was used to reproduce the measured blade vibrations in an accurate way in [1]. The blade excitation potential due to asymmetric incoming flow is examined by means of several steady CFD calculations, each differing in the model resolution. It is shown that the potential blade vibration excitation due to suction elbows or asymmetric aspiration ports is accurately determined by a standard steady-state calculation and therefore the computational effort can be minimized.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pPS</td>
<td>static pressure on pressure side</td>
</tr>
<tr>
<td>pSS</td>
<td>static pressure on suction side</td>
</tr>
<tr>
<td>Dp</td>
<td>blade loading (pressure difference: pPS - pSS)</td>
</tr>
<tr>
<td>ML</td>
<td>Meridional Length</td>
</tr>
<tr>
<td>dt</td>
<td>Time step</td>
</tr>
<tr>
<td>OPcC</td>
<td>Operating point close to choke</td>
</tr>
<tr>
<td>OPcS</td>
<td>Operating point close to surge</td>
</tr>
<tr>
<td>harm.</td>
<td>Dp harmonic blade loadings norm. normalized</td>
</tr>
<tr>
<td>DpCF,EO</td>
<td>weighted pressure fluctuations on circumferences for different engine orders</td>
</tr>
<tr>
<td>x1,…..xi</td>
<td>Values for consideration of leverage effects</td>
</tr>
<tr>
<td>A-F</td>
<td>locations on circumferences of Fourier analysis in front of the impeller</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Method</td>
</tr>
<tr>
<td>Q3D-</td>
<td>method Quasi 3 Dimensional method</td>
</tr>
</tbody>
</table>

Introduction
Centrifugal compressors of turbochargers operate in a wide range of rotational speeds, which depends on the load of the supercharged (Diesel) engine. Consequently, there is a significant number of possible intersections between the excitation order and the natural frequencies of the impeller blades with a potential for resonant vibrations. Therefore, it is of great importance for a safe design of centrifugal compressors, to determine the structure’s vibration behavior so that it can be properly accounted for operation under resonance conditions.
Current designs of turbocharger compressors exhibit high efficiencies accompanied by high flow capacities [2]. Consequences of aerodynamic optimization are high mean stress values in the blades due to centrifugal loading as well as dynamic stresses due to blade vibrations. Blade vibrations in a turbocharger compressor are assumed to be predominantly excited by unsteady aerodynamic forces. These forces are caused by a variety of effects influencing the flow. These influences on the flow are known from geometrical characteristics, such as the flow channel, elbows and the diffuser vanes or struts. Therefore, an understanding of the fluid-structure interaction is essential for further design optimizations.

The calculative determination of blade vibrations in centrifugal compressors is difficult. The most common procedure is to carry out unsteady Navier-Stokes CFD calculations of the respective compressor stage. In this process, unsteady static pressure data on surfaces of blades are transmitted from the CFD mesh to the FEM mesh. Through the use of Fast Fourier Transformation analysis harmonic excitations are determined. Dickmann et al. [1] and Srivastava et al. [3] build upon this approach. Assuming a given amount of damping, the amplitudes of blade vibrations and maximum stresses of the impeller are calculated with an FE model. Recent developments in industrial CFD-codes indicate that this process can be handled easily and automatically in the future. The intention of these industrial CFD-codes is to calculate bidirectionally. This means that after every CFD time step, an FE calculation is started. Thereby mesh deformations and their codependent influence on Fluid Structure Interaction (FSI) behavior can be determined. For example Nipkau et al. [4] did such bi-directional investigations by coupling industrial CFD and FE- Codes. Nevertheless, there is still much computational effort for Fluid Structure Interaction calculations in centrifugal compressors. Considering a complete compressor stage, for a single operation point, the calculation effort lies in the range of about 10 weeks using 10 CPUs.

Blade vibrations in single stage centrifugal compressors are complicated through asymmetric elements like volutes with tongues behind the diffuser or suction elbows upstream of the stage. These elements influence the amplitudes of blade vibrations as proven by experimental research. To include these elements in unsteady Navier-Stokes CFD calculations raises the computational effort due to their geometrical extension. In order to speed up these calculations, the models are simplified so that the computational effort can be minimized. For example Gould et al. describe a Q3D-method in [5] where the blade loading due to the interaction between the impeller and the vaned diffuser is calculated easily. The upstream interaction causes potential blade excitations for the impeller. It is demonstrated that this interaction is dominated by different flow quantities like the relative Mach number, the deceleration of the flow in the compressor channel and further parameters. Winter et al. [6] describe a very simplified method which is used to calculate diffuser induced blade excitations of an impeller. Assuming that the vaned diffuser generates harmonic blade excitations at the trailing edge, FE calculations are carried out. Additionally it is assumed, that these forces at the trailing edge grow linearly with increasing rotational speed. For potential eigenmodes of the respective nodal diameters of the dispersion diagram FE simulations with harmonic excitation forces are carried out to calculate the maximum stresses of the impeller. Therefore the vibration behavior of an impeller can be estimated for different resonances without time consuming CFD calculations and potential blade excitations caused by the diffuser can be accounted for in the design process.

**Background**

The CFD simulations of a single stage centrifugal compressor have been carried out by Dickmann et al. [1] with the CFX5.7 code from ANSYS [7]. An impeller with 8 main and splitter blades, an inducer casing bleed system, a low solidity diffuser with 9 vanes, a 90 degree suction elbow and an asymmetric diffuser outlet static pressure distribution, typical for volutes was modelled. The asymmetric pressure distributions at the outlet of the diffuser allows doing away with the modelling of the volute so that the computational domain is smaller and less computational effort is necessary. Average values for total temperature and total pressure, a turbulence intensity of 5 % and flow vertical to the inlet boundary have been applied to the inlet boundary. One impeller revolution has been resolved by 256 time steps using a grid with 730'000 cells. The k-e turbulence model with scalable wall functions has been applied. 2nd order discretization in space has been used. Each time step is converged to 10-3 of all maximal residua after 10 internal time step loops. These maximal residua have been regarded as sufficient concerning the data of interest.

The compressor stage belongs to a turbocharger, which supercharges a state-of-the-art 1,100 kW Diesel engine. Investigations described here have been done on two off-design operating points on a speed line of 83 % design speed. OPcC corresponds to 80.4 % of design volume flow, 67.8 % design total pressure ratio and 103 % of design efficiency, OPcS to 60.0 %, 70.7 % and 100.0 % respectively (figure 1). The relative Mach numbers at the tip leading edge of the impeller are 1.50 for OPcC and 1.20 for OPcS. The absolute Mach numbers at the leading edges of the diffuser vanes are 0.65 for OPcC and 0.60 for OPcS.
The unsteady pressure data on the surfaces of the blades from transient CFD results were used to determine the harmonic excitations. When applied to an FE Model the amplitudes as well as the stresses in the impeller of the first eigenmode could be calculated. Results for the two operating points OPcC and OPcS on the same speed line are compared. The main statement of [1] is that different amplitudes of an inlet induced mode shape could be calculated for operating points close to choke and close to surge on the same speed line and that calculated amplitudes have been in good agreement with experimental research. These results act as a reference for the following analyses.

Motivation
Suction elbows are usually designed by manufacturers of engines in order to save space in the machine room. Hence the incoming flow of centrifugal compressors is often inhomogeneous and non-axisymmetric. A typical suction elbow for turbochargers leads to inhomogeneous velocity distributions in front of the impeller. For example the stage as it was modelled in [1] with a 90 degree suction elbow leads to a steady inhomogeneous velocity distribution as displayed in the upper part of figure 1. This velocity distribution of plane B has been calculated by means of steady-state Navier-Stokes simulations. Plane B is an area in front of the impeller as demonstrated in the middle part of figure 2. Additionally four planes with snapshots of unsteady local pressure distributions, calculated by means of transient Navier-Stokes simulations, have been chosen in the middle part of figure 2 to show influences of circumferentially asymmetric geometries (suction elbow and volute) on the impeller flow. Vertical views can be seen in the lower part of figure 2. Plane A and B illustrate the

1 Performance map excerpt of the investigated compressor [1].
2 Steady allocation of velocities in plane B (upper part), snapshots of local pressure distributions (lower parts) [1].
change of the flow pattern caused by the suction elbow. Plane B extends into the inducer casing bleed system and is located in the center of the upstream circumferential slot. Plane C is located in the center of the downstream circumferential slot while plane D represents the midspan plane of the vaned diffuser. In the upper part of figure 2 it can be seen that the flow is accelerated at the inner radius of the suction elbow. This leads to a lower static pressure distribution as it can be seen in Plane B in the lower part of figure 1. In Plane D the influence of the tongue can be detected, as regions of higher static backpressure around the tongue are present.

The potential of blade excitation due to asymmetric incoming flow depends on the degree of inhomogeneity of the referring suction elbow. Extensive experimental tests for different designs of suction elbows is not possible for economical reasons.

Hence in this work a simplified procedure is presented in order to ensure a safe design of the suction elbow concerning the blade vibration potential, without applying time consuming calculation methods or experimental tests.

**Analysis of the blade excitation potential**

Transient blade loadings characterize blade vibrations. Hence in a first step, the blade vibration potential is visualized based on the unsteady Navier-Stokes CFD calculation results. For this purpose the unsteady static pressure data on the surfaces of the main blades of 50 % channel height are plotted against the meridional length for one impeller position. In figure 3 the static pressure is shown for the pressure side and the suction side for one of the 256 time steps (one revolution =256 dt). In this work static pressure data are normalized by a constant value.
The resulting blade loading is calculated by the pressure difference between the pressure and suction side after equation (1). This difference is shown in the lower part of figure 2.

\[ \Delta p = p_{\text{ss}} - p_{\text{PS}} \quad (1) \]

The blade loading \( \Delta p \) is a function of the meridional length and the time referring to the rotor position. Representing the unsteady loading, for the operation point OPcC this dependency is shown in figure 4 and for OPcS in Figure 5.

In both figures, it is obvious that a diffuser with 9 vanes was used as there are nine maxima at the trailing edge (meridional length = 1) per revolution. The diffuser vanes cause blockages in the flow, which penetrate upstream and influence the impeller flow. For both cases, an influence of the volute tongue is recognized. The influence of the interaction between impeller and vaned diffuser on the impeller flow is stronger for OPcS than for OPcC. While the influence of the diffuser and the volute tongue on the impeller flow field is seen for \( m = 0.3 \) to \( m = 1 \) for the operation point close to the surge line, its upstream effect decays at half of the impeller length for OPcC. This effect is caused by the larger mass flows near the choke margin resulting in larger flow velocities. With the high velocities the interference cannot spread as far as for OPcS. With the high velocities the interference cannot spread as far as for OPcS.

For OPcC a clear influence of the suction elbow is recognized in the leading edge region (ML = 0). This is clarified in figure 6. In this diagram the blade loading near the leading edge (ML = 0.05) in function of circumferential blade position. The curve for OPcC is a point symmetric curve. The centre of the point symmetry is marked by a red point. The point symmetry suggests that this kind of blade loading arises through a half symmetric part like the suction elbow which was used in this case (figure 2). The blade loadings for OPcS with the same exposition are displayed in figure 6. It can be seen that for OPcS higher blade loadings arise and that point symmetry does not arise. Higher
Amplitudes for OPcS in comparison to OPcC have been calculated in [1], too. This is caused by the upstream influence of the volute tongue which reaches as far as to the leading edge. Since the volute tongue has an influence as far upstream as the leading edge, this leads to the conclusion, that this operating point is less sensitive for the potential blade excitations due to asymmetric incoming flow.

For the determination of harmonic blade loading fast Fourier transformations over time have been carried out for every data point on the meridional length. The results are harmonic blade loadings over the meridional lengths that are a potential blade excitation for respective impellers. For OPcC these harmonic blade loadings are demonstrated in Figures 7 - 10 for engine orders 3, 4, 5 and 6. These blade loadings are calculated for 8 different channel heights from hub to shroud. Additionally in the same figures, the harmonic blade excitations for OPcS are illustrated as dashed lines for two different channel heights. In comparison to the harmonic blade loadings of OPcC those for OPcS are clearly higher in the trailing edge region than for OPcC. The rise of these harmonic blade loadings corresponding to decreasing flow rates on the same speed line was recognized in [3] for different operating points as well. The comparison of the two operating points shows much higher excitations for OPcS. But the higher excitations for OPcS appear mainly in the exit region of the main blade. It is assumed that excitations in the exit region of blades only have negligible influences on blade excitations of inlet induced mode shapes. So as inlet induced mode resonance occur in the front of the main blades, additional excitations at the blade exit due not necessarily contribute to higher amplitudes close to surge. The size of the amplitudes depends additionally on the aerodynamic damping which varies according to different operating points and eigenmodes [7] of the stage.

Comparison between the harm. norm. Dp of different engine orders in the front region of the blade shows that for engine order 3 higher norm. harm Dp appear than for engine order 4, 5 or 6. For resonance vibration, it is necessary that intersections between the engine order and the natural frequencies of the impeller appear. Campbell diagrams show these connections. As the norm. harm Dp of engine order 3 are comparatively big it is advantageous that the first eigenmode of engine order 3 is outside of the potential operating conditions. Experimental research showed that potential blade excitations of engine order 5 and 6 of the first eigenmode do not cause critical alternating stresses as engine order 4 does. As resonant vibrations of the first eigenmode of engine order 5 and 6 appear under unequal operating conditions, it is not possible to compare these different harmonic blade excitations.

Investigations of different calculation methods
The goal of this work is to develop a method calculating blade vibration potential caused by the asymmetric incoming flow as simple as possible. In order to investigate this issue in the next step different calculation methods are carried out and are compared for OPcC:
1. Frozen Rotor Calculations
2. Mixing Plane Calculations
3. Unsteady calculations
4. “Method of the half symmetric channel”

For investigation of the different calculation methods the geometries, the meshes, and the boundary conditions from [1] are used except for the “method of the half symmetric channel”.

10 Harmonic blade loadings of engine order 6.

11 Locations where fast Fourier transformation is applied.
The Frozen Rotor approach is a snapshot of an unsteady calculation, whereas flow values at the interfaces are passed through a simulated plane neglecting the influence of the circumferential speed. The mixing Plane model is a steady state calculation passing averaged values over circumferences at the rotorstator interfaces. The unsteady calculation method is the most accurate method of these three as every relative rotor position is considered and the flow values are passed correctly at the interfaces. Due to the fact that the unsteady calculation method is the most accurate method, results of this method will be used as reference for the comparisons. In addition the “method of a half symmetric channel” is investigated. In this calculation method the suction elbow, the hub and the shroud have been modelled, neglecting the impeller blading.

As boundary condition for the symmetry plane, no slip surfaces have been used. To compare “the method of the half symmetric channel” with the other methods, the back pressure was adjusted in order to meet mass flows comparable to OPcC.

Comparison of the calculation methods
In order to compare the different calculation methods, for every method static pressures on circumferences in front of the impeller are calculated. The position of the evaluation points is shown in Figure 10.

It is assumed that harmonic blade excitation due to asymmetric incoming flow can be calculated by means of fast Fourier transformations of the static pressure on circumferences in front of the impeller. For the evaluation of the unsteady calculations averaged pressure data of one revolution (taking into account 256 dt have been used to calculate harmonic blade excitation potential. In order to reduce the influence on a certain relative wheel position for Frozen Rotor calculations, harmonic blade excitations for averaged pressure data of three wheel positions have been calculated. Harmonic blade excitation potential of one steady mixing plane method is used for the evaluations. Different wheel positions are not necessary in this case as flow values are averaged over circumferences at these interfaces.

For the “method of the half symmetric channel”, static pressure data of semi circles are used for the evaluation. The idea of this approach is that excitations are a result due to the inhomogeneity of the flow, thus a result of the geometric asymmetry of the suction elbows and that effects on the flow field depend on geometric mass flows of the unbladed duct and do not depend on flows of the bladed impeller. May be the geometry of the suction elbow by itself causes similar harmonic blade excitations on circumferences in front of the impeller. Thereby computational effort could be minimized as this method is very simple compared to the other methods. Harmonic blade excitations of the different methods are displayed for engine orders 1 to 20 for the location F of Figure 11 in Figure 12.

Mainly the fourth and fifth engine orders are relevant for the assessment of inlet induced modes, nevertheless engine orders from 0 – 20 have been investigated for the comparison of the different calculating methods. Different engine orders are connected with lines in this diagram type although there is no coupling in between. However, this diagram type is very useful to compare the different methods.

Since it is assumed that the results of unsteady calculations are the most accurate, these results are used as a reference. The comparison in Figure 12 of the harmonic blade excitations of the different calculation methods shows that the method of the half symmetric channel delivers much higher blade excitations than the three other methods. Additionally, there is no qualitative accordance of the curve progressions in comparison to the other methods. From this, it is expected that “the method of the half symmetric channel” is not capable to calculate potential harmonic blade excitations due to asymmetric incoming flow. The evaluations of the other three methods show similar results. Variations for several engine orders appear in the comparison of the “Frozen Rotor Method” and the unsteady method in Figure 12.

These methods show comparable results. It would be possible to reduce these variations by considering more different wheel
positions. But in comparison to the Mixing Plane Method this requires an extensive additional effort, as with the Mixing Plane Method very similar results arise compared to the unsteady method. With the assumption that harmonic blade excitations are due to asymmetric incoming flow, steady mixing plane calculations are a relatively simple and sufficiently precise approach for estimations of harmonic blade excitations.

Comparison of the harmonic blade loadings
In the next step it is shown that those harmonic pressure fluctuations on circumferences in front of the impeller are in direct coherence with harmonic blade loadings.

Harmonic blade loadings $D_p$ which have been calculated with the unsteady method on the surfaces of one main blade have been described in the previous section. Those are displayed for $OP_{cC}$ of different engine orders and for different channel heights in Figures 7 - 10. It is assumed that harmonic pressure fluctuations on circumferences in front of the impeller appear as harmonic blade loadings in the forward area on the surfaces of the blades as well. As inlet induced modes of vibration mainly appear in the front region of the blades as it can be seen in Figure 13, this assumption is valid.

Since harmonic pressure fluctuations on circumferences in front of the impeller appear as harmonic blade loadings, this shows that potential blade excitations due to asymmetric incoming flow can be determined or at least be estimated by means of steady mixing plane calculations. To compare harmonic pressure fluctuations over circumferences in front of the impeller with harmonic blade loadings in the forward area on the surfaces of the blades, the maxima of the blade loadings in the forward 20\(\%\) ($ML = 0 - 0.2$) are determined. This was carried out for harm. blade loadings of engine orders 3, 4, 5 and 6 and 6. In these diagrams, for different engine orders, a qualitatively similar curve progression can be seen for both methods of evaluation, whereas the harmonic blade loadings on the surfaces of the blade in comparison to the harmonic pressure fluctuations on circumferences in front of the impeller appear about 10 - 15\% radially further outside.

This can be explained, by the flow expanding due to the spinner of the impeller and so these fluctuations are expanding radial outwards, too. Although for both methods different scales are used, the qualitative correspondence is inherent. The scale factor is dependent on the damping factor. With these comparisons it is evident that by means of steady mixing plane calculations and with their harmonic pressure fluctuations on circumferences in front of the impeller, potential blade excitations can be calculated.

Weighting
For centrifugal compressors the first mode shape of the main blades is mainly influenced by asymmetric incoming flow. The first mode shape of the main blades is a so called “tab mode”, which is shown in Figure 12. To assess different suction geometries in the next step a weighting is introduced for the first mode shape. Thereby potential blade excitations are considered.

Through this weighting, it is considered that harmonic pressure fluctuations, which are radially further outwards, take more influence on blade excitations of the first mode shape because of leverage effects than radially inner lying blade excitations. For this, harmonic pressure fluctuations of the locations A – E from Figure 10 on the circumferences in front of the impeller are weighted after (2). At this an axial position for the locations of circumferences in front of the impeller had to be chosen for the weighting. In Figure 11 it can be seen that data of 3 different axial positions (X, Y and Z) are available.

On the one hand it can be shown that pressure fluctuations of locations that are closer to the impeller are distorted due to the circumferential averaging of the mixing plane rotor-stator interface. On the other hand pressure fluctuations of circumferential locations that are too far upstream of the impeller will not be comparable with harmonic blade loadings over the channel height as in Figure 18-21. That is that the axial circumferential positions for these evaluations have to be optimized for the respective position of the rotor-stator interface.
13 Comparison of calculation methods.

14 Maxima of harmonic blade loadings $ML = 0 – 0.2$ for engine order 3.

15 Maxima of harmonic blade loadings $ML = 0 – 0.2$ for engine order 4.

16 Maxima of harmonic blade loadings $ML = 0 – 0.2$ for engine order 5.

17 Maxima of harmonic blade loadings $ML = 0 – 0.2$ for engine order 6.

18 Comparison of pressure fluctuations and harmonic blade loadings for engine order 3.
In Figure 21 there is a comparison of the different calculation methods under consideration of equation (2).

$$\Delta p_{CR,EO} = x_1 \cdot A + x_2 \cdot B + x_3 \cdot C + x_4 \cdot D + x_5 \cdot E$$

(2)

Also this diagram makes clear, that with steady-state mixing plane calculations nearly the same potential blade excitations can be calculated as it is possible with unsteady calculations. Except for the 4. engine order, variations of the Frozen Rotor-Method do not increase in accuracy as a function of increased computational effort, since different relative wheel positions have to be considered.

**Conclusions**

It was demonstrated that asymmetric parts like suction elbows and volute tongues have a significant influence on potential harmonic blade excitations and that for respective operating points excitations of the different parts are superimposed. It is shown that harmonic pressure fluctuations, which are calculated with the steady mixing plane method on circumferences in front of the impeller appear qualitatively similar as harmonic blade loadings in the front 20 % on the surfaces of the main blades, too. This means that potential blade excitations due to asymmetric incoming flow can be determined by steady calculation. These calculations are associated with considerable less computational effort than unsteady calculations. A weighting with respect to the first mode shape was introduced with the goal that different suction geometries can be compared very easily to each other and a relative comparison of different air intake constructions is possible by this approach.

19 Comparison of pressure fluctuations and harmonic blade loadings for engine order 4.

20 Comparison of pressure fluctuations and harmonic blade loadings for engine order 5.

21 Comparison of pressure fluctuations and harmonic blade loadings for engine order 6.

22 Comparison of weighted calculation methods.
Acknowledgments

The authors would like to thank ABB Turbo Systems Ltd for the permission to publish this work. As this work is based on a collaboration with the “Institut für Thermische Strömungsma-

schinen” (ITSM) of the University Stuttgart the authors like to thank as well. Several colleagues contributed to the success of this project, particularly we are thankful to D. Robinson for his final internal review.

References


Proceedings of ASME Turbo Expo 2009:
Power for Land, Sea and Air
GT2009
June 8-12, 2009, Orlando, Florida, USA
Contact us

For more information please contact:

ABB Ltd
Affolternstrasse 44
CH-8050 Zurich, Switzerland

www.abb.com