FSI MODELLING OF AN INDUSTRIAL CENTRIFUGAL COMpressor STAGE OPERATION AT STABLE AND UNSTABLE OPERATING POINTS

K. Kabalyk

Lodz University of Technology, Institute of Turbomachinery, Lodz, Poland, kirill.kabalyk@p.lodz.pl

ABSTRACT
The paper describes an effort to numerically define the temporal dependences of mechanical stresses in a centrifugal compressor impeller caused by unsteady aerodynamic loads at nominal and diffuser stall conditions. The object is a moderate Mach number shrouded wheel followed by a vaneless diffuser. The method applied is one-directional fluid-structure interaction with fully-transient fluid-flow- and structural simulations. The results indicate a clear distinction between stable and unstable flow-induced forces acting on the blades. On the other hand, the mechanical response predictions show rather marginal sensitivity to the operating point studied.

NOMENCLATURE

\( b \) meridional blade (channel) height (m)
\( D \) diameter (m)
\( \text{FE} \) finite element
\( \text{FFT} \) fast Fourier transform
\( \text{FSI} \) fluid-structure interaction
\( I \) turbulence intensity
\( L \) meridional length (m)
\( \dot{m} \) mass flow rate (kg/s)
\( M_u \) specific Mach number (based on the impeller tip speed)
\( n \) rotation speed (rev/min)
\( NP \) nominal point
\( p_{tot} \) total pressure (Pa)
\( RS \) rotating stall
\( T_{tot} \) total temperature (K)
\( \text{VLD} \) vaneless diffuser
\( Z \) axial coordinate (m)
\( \beta_{bl} \) blade sweep angle at impeller outlet (deg.)
\( \theta \) circumferential coordinate (deg.)
\( \mu_T/\mu \) eddy viscosity ratio
\( \Pi_{2s} \) impeller pressure ratio (based on total parameters)
\( \tau \) current simulation time (s)
\( \Delta \tau \) time step size (s)
\( \omega_{rot} \) impeller (rotor) angular frequency (rad/s)
\( \omega_{cell} \) rotating stall cell angular frequency (rad/s)
INTRODUCTION

Off-design operation of an industrial centrifugal compressor may be a consequence of such factors as e.g. partial modernization of the technological circuit it used to feed (reactor replacement, etc.), temporal changes in network resistance and others. In cases when higher pressure ratio is required and no increase in rotational speed is available the multi-stage unit will always meet a risk of local unstable flow phenomena inception, especially at the last stages (Aungier, 2000). According to the same author what is most likely to occur will be vaneless diffuser rotating stall (accepting the stage is VLD-based). It is typically reported as a pattern of circumferentially distributed higher- and lower pressure zones rotating slower than impeller and thus generating additional dynamic load on the passing wheel’s cascade. Recently Mischo et al. (2018) demonstrated a reasonable potential of numerical fluid-structure coupled simulations to correctly predict the dynamic stresses caused by vaned diffuser stall in an unshrouded turbocharger impeller. This work, attempts to apply FSI to search for possible differences in impeller stress response when operating at nominal and moderate VLD rotating stall conditions since the latter is less likely to be recognized by the control automatics.

DESCRIPTION OF THE TEST CASE

Object compressor stage

The object of the investigation is a single centrifugal stage illustrated in Fig. 1. The rig abbreviated as SP-1.4 is being currently built at the authors’ employing institution on the platform of the former VRK-3 three-stage integrally-geared industrial compressor. The elements borrowed include the shrouded fifteen-blade impeller (pos. 2), the bull-gear casing (pos. 4) and the ex-stage 2 volute casting (pos. 3). In the present design, the newly manufactured shaft is supported by high-precision ball bearings and is directly coupled to the variable-speed electric motor (pos. 5) with maximal power output of 50 kW. The nominal operating conditions estimated on the basis of an internally developed one-dimensional procedure leaning on the open literature sources, e.g. Seleznev et al., 1990, correspond to: \( \dot{m}_{NP} = 0.83 \text{ kg/s}, \) \( \Pi_{2NP} = 1.29, \) at \( n = 15000 \text{ rpm}. \)

The rig is being developed as a part of a grant-research project devoted to a search of a sophisticated and robust active surge control system. Altogether, there are 23 thoroughly distributed wall static pressure drills for high-frequency transducers (\( \text{FP}_{0'} - \text{FP}_{4'} \)) coupled with industrial pressure (\( \text{SP}_{in}, \text{SP}_{out} \)) and temperature (\( \text{ST}_{out} \)) gages situated in suction (pos. 1) and discharge (pos. 6) pipelines for stationary performance tests. The list of characteristic dimensions of the stage’s flowpath is in Tab. 1.

| Table 1: Main geometric parameters of the examined stage, [mm] |
|-------------|-------------|-------------|-------------|-------------|
| \( D_0 \) | \( D_2 \) | \( D_2' \) | \( D_4 \) | \( D_4' \) | \( b_2 \) | \( b_{VLD} \) | \( Z_0 \) | \( Z_0' \) | \( L_{ax} \) | \( \beta_{bl2} \) |
| 130 | 265 | 324 | 390 | 360 | 13 | 12.5 | 82 | 107 | 137 | 75° |

Definition of the fluid-flow problem

The core of the two fluid (air ideal gas) numerical domains used in the study (Fig. 2) embodies the impeller and VLD full-annulus sections meshed with hexahedra in ANSYS TurboGrid 18.2. The rest is meshed in ICEM CFD with tetrahedra and hexahedra chosen for the suction pipes and for the crossover respectively. The decision to alter the version created initially.
and applied successfully for the nominal point run (Fig. 2, left) was justified by rather poor simulation convergence after the operating condition was switched to rotating stall. Reduction of the pipe length to the extent shown in Fig. 2, right accompanied by an addition of the short crossover section (following the experience of Mischo et al. (2018) whose similar study reported the absence of volute domain influence on diffuser pressure field modelling results) eliminated the convergence issues and provided an extra-shift of the outlet boundary from the diffuser exit. The eventual grid sizes are $2 \cdot 10^6$ nodes ($1.8 \cdot 10^6$ elements) for ”NP” domain and $2.2 \cdot 10^6$ nodes ($2.1 \cdot 10^6$ elements) for ”RS” domain. The mesh independence study has been carried out ensuing the recommendations in Celik et al. (2008). The reported grids revealed the grid convergence indices of less than 1% for the majority of the monitored parameters.

The results reported in this work concern transient URANS simulations completed in ANSYS CFX 19.2. However, the proper initial and boundary condition set required a conduction of a series of preliminary steady-state runs. The latter manifested that zero-incidence or NP condition was achieved at a higher mass flow than initially evaluated by 1D procedure.

Every unsteady simulation resolved a single impeller revolution in 64 timesteps adhibiting the second order backward Euler scheme. It must be said that even for a vaneless diffuser-based stage this temporal step size seems very large. The state-of-the-art papers (e. g. Dickmann et al. (2006)) claim employing the steps of at least four times lower magnitudes. The value assumed in this work, is mainly a consequence of computational power bounds (single node with an eight-core CPU (Intel Xeon 2.7 GHz) and 32 GiB RAM). The latter were also the reason for a shorter total run time at RS condition (see Tab. 2). In spatial discretization, the high-resolution scheme was selected both for the conservation and turbulence transport equations. The simulations were run exploiting two-equation $k – \omega$ SST turbulence model.

The rotational speed of impeller domain was set to $n = 15000$ rpm ($M_a = 0.6$). The
connection of the impeller grid to the surrounding ones (suction pipe and VLD) was executed with transient rotor-stator (sliding mesh) interface model whereas the VLD-crossover interface utilized the stationary general grid interpolation (GGI) method. The simulations did not account for the external heat exchange and considered the wall surfaces as hydraulically smooth ones.

The inlet boundary condition pattern involved total pressure, total temperature, eddy viscosity ratio and turbulence intensity. Mass flow rate was specified at the outlet. The exact values of these parameters are available in Tab. 2 with respect to the operating condition simulated.

Table 2: **Boundary and temporal conditions set used in fluid-flow simulations**

<table>
<thead>
<tr>
<th>Inlet</th>
<th>Outlet</th>
<th>Temporal</th>
</tr>
</thead>
<tbody>
<tr>
<td>$p_{tot}$</td>
<td>$T_{tot}$</td>
<td>$\mu_T/\mu$</td>
</tr>
<tr>
<td>Pa</td>
<td>K</td>
<td>-</td>
</tr>
<tr>
<td>NP 101325</td>
<td>293</td>
<td>1</td>
</tr>
<tr>
<td>PRS 101325</td>
<td>293</td>
<td>1</td>
</tr>
</tbody>
</table>

**Definition of the structural analysis problem**

The FE mesh of the impeller shown in Fig. 3 is tetrahedral with 233.3k elements (403.6k nodes). The wheel material is X5CrNi 13-4 steel with a relatively high yield strength of 940 MPa. Nevertheless, to simulate the possible plastic deformation effects, the bilinear isotropic hardening model was tagged to material properties.

The cylindrical support specified on the shaft mounting hole surface restricts tangential and axial motion, but leaves the possibility of radial displacement. The gas forces acting on the cover and main disk surfaces from the sides of impeller-casing gaps were substituted by constant pressure loads. The values of the loads (depending on the operating point) were borrowed from the CFD part and equalled to area-weighted static pressures at impeller-VLD interface. The
rotation-induced loads were accounted for by defining the rotational speed of 15000 rpm.

Fluid-structural solvers’ communication was implemented in three coupling iterations occurring every timestep. The same number of internal iteration loops was specified for the CFD solver. The mesh deformation due to FSI was omitted leaving the coupling to stay one-directional. This approach has been successfully implemented in earlier works, e.g., in Lerche et al. (2012). The data-exchanging interfaces were the shroud, the blades and the hub surfaces respectively.

**Figure 3:** A FE mesh of impeller used in structural analysis

**FLUID-FLOW SOLUTION ANALYSIS**

**Monitor points distribution**

The array of monitor points schematically shown in Fig. 4,a was generated a) to duplicate the locations of taps for high-frequency pressure transducers in the test rig and b) to enable a detailed inspection of the fundamental flow parameters’ (static pressure, static temperature, velocity, etc.) time-histories throughout the main areas of interest. The latter comprised the suction pipe’s segment close to impeller eye (points ”In...”), the vicinity of two neighbouring blades’ walls (points ”B...”) and the entire VLD domain (points ”D...”). Each point (except blade-adjacent ones) was translated over circumferential coordinate with the step of either 30° or 15°. The majority of the results discussed below concern the data exported from those monitors after the corresponding runs had converged.

**Pressure signals in stationary elements**

The maps compiled in Fig. 5 are the temporal-spatial static pressure signal resolutions computed correspondingly at 90% span at impeller inlet (a, b), 10% span at VLD inlet (c, d) and VLD outlet (e, f). At these locations, the flow stability occurred to express the highest sensitivity to the operating point.

The diagrams for \( \dot{m} = 1.17 \text{ kg/s} \) indicate a rather expected signal behaviour. The ”InS1” station reveals lack of circumferential disturbances whereas the transient pressure fluctuations
lie within the range lower than 250 Pa, which should be either a nozzle- or numerically provoked effect. At VLD inlet ("DH1"), the fluctuation character is obviously explained by the periodically disturbed flow leaving the wheel. For each out of 24 bands shown in Fig. 5,c the FFTs (not presented here) revealed the strong dominance of the 3750 Hz frequency peak, which is exactly the blade passing frequency. At "DH4" the impeller cascade-provoked oscillations are still present, but are quenched down enough to be enclosed by a span of less than 250 Pa, which practically implies stable and uniform flow.

Switching the mass flow to $\dot{m} = 0.75 \text{ kg/s}$ brings up significant transformations to the patterns present in the maps. The signals at impeller eye (Fig. 5,b) proclaim the appearance of a circumferentially non-uniform pressure pattern rotating approximately 2.1 times faster than the impeller. Similar circumferential pressure disturbances travelling 1.9 times the impeller speed appear both at VLD inlet (Fig. 5, d) and outlet (Fig. 5, f). The latter are additionally visualized in Fig. 4,b along with surface streamlines revealing the presence of the so-called diffuser stall cell, see e. g. Japikse, 1981. The black point denotes the corresponding locations of a reference blade’s trailing edge with respect to the stall cell.

The rotational speed lines of $2.1 \omega_{rot}$ and $1.9 \omega_{rot}$ pointed out in the lower row of Fig. 5 do not seem to sound as a reliable result from the standpoint of the vaneless diffuser rotating stall. As was said in the introduction, the frequencies of rotating stall cells rarely overcome the interval of $0.5 - 0.7 \omega_{rot}$ whereas usually lie within $0.1 - 0.3 \omega_{rot}$ (Izmailov, 1995). An additional coupled simulation of six impeller revolutions with a four times lower timestep of $1.56 \cdot 10^{-5} \text{ s}$ (results are not presented here) has been carried out, but revealed no significant differences with respect to the $\omega_{cell}$. However, after the FSI coupling had been disabled and a standalone fluid-flow run of ten impeller revolutions had been completed at $\dot{m} = 0.75 \text{ kg/s}$ (not presented here) the angular frequency of the cell reduced to $0.3 \omega_{rot}$. Understanding of this effect is left as a crucial point for further research.

Figure 4: **Left:** a scheme of monitor points distribution employed in fluid-flow analysis (meridional plane), **Right:** static pressure distribution snapshots along the surface 10% far from diffuser hub wall, $\dot{m} = 0.75 \text{ kg/s}$, $\tau = 0.0185 - 0.02 \text{ s}$


Pressure signals in rotating frame of reference

The CFD data representing the transient course of the fluid-induced loads acting on a single blade (B1 in Fig. 7) are in Fig. 6 as static pressure signals (a, c) and Fourier amplitude spectra (b, d). Each signal map (to a bigger extent, the one at $\dot{m} = 1.17 \text{ kg/s}$) reveals the effects of inlet meridional curvature, incidence and blade loading. However, in context of this study, a characteristic shift from absolutely flat curves at $\dot{m} = 1.17 \text{ kg/s}$ to periodically fluctuating ones at $\dot{m} = 0.75 \text{ kg/s}$ is more crucial. The corresponding FFTs reveal the dominant harmonics of either 212 Hz or 255 Hz arising only at lower flow rate. Since these monitors were explicitly set in rotating reference frame and assuming the $1.9 - 2.1 \omega_{\text{rot}}$ being the main excitation source the harmonics’ frequencies prediction looks correct. In order to define the potential resonance condition the modal analysis has been carried out on the same structural grid and revealed the impeller first natural frequency mode of 1475 Hz, which excludes the possibility of resonant excitation.

STRUCTURAL SOLUTION ANALYSIS

The distribution of equivalent stresses over the wheel’s surfaces corresponding to the case of $\dot{m} = 1.17 \text{ kg/s}$, $\tau = 0.04 \text{ s}$ is given in Fig. 7 along with the blade numbering. The regions of the most intensive stress concentration (highlighted) occurred at the leading edge-cover joint and at the trailing edge corners. The areas were, therefore, selected for a more thorough analysis.

Fig. 8 collects the time-dependences of von Mises stress maxima computed individually for...
Figure 6: Time-resolved computed static pressure fluctuations along leading and trailing edges of the "B1" blade (upper row) and their corresponding Fourier amplitude spectra (lower row). **Left**: $\dot{m} = 1.17$ kg/s, **right**: $\dot{m} = 0.75$ kg/s

Figure 7: An example of equivalent stress distribution along the impeller surfaces obtained in a fluid-coupled structural analysis (left) and definition of stress concentration areas at trailing (upper right) and leading edges (lower right) each out of 30 edges at two operating conditions. The diagrams reveal marginal sensitivity of the stress responses to the mass flow value. At each location, there exists a certain convergence
Figure 8: Numerically obtained pseudocolor maps of maximal equivalent stress on impeller blades. Upper row: $\dot{m} = 1.17$ kg/s, lower row: $\dot{m} = 0.75$ kg/s. Left column: leading edge, right column: trailing edge

period followed by a complete stabilization of the stress level till the end of the simulation. Since at $\dot{m} = 0.75$ kg/s the impeller operates at a higher pressure ratio than at $\dot{m} = 1.17$ kg/s the stresses at RS are averagely lower than at the nominal point by 5 MPa at the leading edge and by 10 MPa at the trailing edge. The reason for the observed differences in the individual blades’ stresses levels must be purely numerical and not fluid-induced, which was proved by conducting a corresponding transient structural analysis without FSI.

CONCLUSIONS

A one-directional FSI analysis initiated to estimate the effect of VLD RS on stress response of an industrial centrifugal compressor impeller unfolded the following effects:

1. URANS CFD analysis with moderate time and space resolutions appeared to be accurate enough to reproduce the principal differences in flow behaviour between nominal and rotating stall regimes. The latter resulted in the appearance of a single cell localized at the hub surface of VLD closer to its exit.

2. The angular frequency of a rotating stall cell identified during the coupled runs was averagely twice higher than the one of the impeller, which implies a certain discrepancy to the experimental observations published in literature. On the other hand, an identical simulation of 10 rotor revolutions without coupling indicated the $\omega_{cell}/\omega_{rotor}$ ratio of 0.3, which sounds more rational from the state of the art viewpoint (e. g. Aungier, 2000).

3. The structural solutions of the coupled system showed the indifference of the impeller’s dynamic stress-response to the operating regime imposed. The static von Mises stress
level computed at stall condition appeared to be slightly lower than the one at the nominal point.

ACKNOWLEDGEMENTS

The work was accomplished under support of Polish National Centre for Research and Development, Grant No Lider/447/L-6/14/NCBR/2015. The author also wishes to acknowledge the contribution of Dr M. Kulak who shared the most of CFD grid data and provided critical remarks concerning the final manuscript.

REFERENCES


