Effective computational procedure for high pressure ratio centrifugal compressor

Martin Babák

První brněnská strojírna Velká Bíteš, a.s.

Abstract: This paper describes the matter of a cost effective computational procedure using FLUENT code to be used in case of high pressure ratio centrifugal compressor. The computational procedure was used in case of small scale compressor from PBS jet engine TJ 100A. The results of performed numerical investigations were then compared to experimental data and established computational procedure was verified.

Keywords: centrifugal compressor, CFD, verification.

Introduction

The flow field within a high pressure ratio centrifugal compressor is very complex with transonic and separated flow regions. Moreover due to high speed rotation of the wheel the flow field is unsteady in time. It makes numerical investigation of the flow field very demanding. There are number examples of very detailed CFD investigations in published papers comprising multi-millions grid points and time dependent solutions of sub-microsecond time scales. Such investigation gives very detailed results of high reliability but it also has got breathtaking demands in terms of computer capacity and time. As a consequence the more “cost” effective approaches are widely used in engineering practice where time and resources are always limited. One of such effective computational procedures is described below and obtained results are compared to experimental data to show that steady state approach together with quite small amount of grid points is also able to produce good results. The matter discussed here is related to FLUENT code which as a general purpose tool requires special treatment of some issues in the case of centrifugal compressor.

Centrifugal compressor stage

Small scale centrifugal compressor (CC) from PBS jet engine TJ 100A was selected for presented study (basic data of the engine is given in Tab.1). Compressor stage comprises backswept wheel, radial waned diffuser, 90 degree bend and axial waned diffuser acting as straightener. CC is of high flow capacity and design pressure ratio is slightly above 5.0.
<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum thrust</td>
<td>1 100 N</td>
</tr>
<tr>
<td>El. Power output</td>
<td>750 W</td>
</tr>
<tr>
<td>Maximum diameter</td>
<td>272 mm</td>
</tr>
<tr>
<td>Length</td>
<td>625 mm</td>
</tr>
<tr>
<td>Dry weight</td>
<td>20.6 kg</td>
</tr>
</tbody>
</table>

**Tab. 1 Basic data of TJ 100A small jet engine**

Experimental data presented in the paper were gathered within the scope of PBS test jet engine facility (see Fig.1).

**Computational meshes**

Only one segment for each wane row was used as mixing plane model was utilized for rotor-stator interaction approximation. Additional mixing plane was used between radial and axial diffuser. Multi-block structured mesh condensed at the walls was produced by means of GAMBIT preprocessor (see Fig.2 and Fig.3). Based on previous experience and findings from available publications (for example reference 2), that good quality of the mesh is much more important then its overall density, rather coarse computation mesh was used. Five cell layers was created in the gap clearance between wheel wane tip and stator wall, whole computation model comprised only 274 607 cells.
Boundary conditions

Several combinations of boundary conditions (BC) can be used in case of high pressure CC within the frame of FLUENT code. Native combination of BC for a compressible case is “pressure inlet” at the inlet of the wheel and “pressure outlet” at the last stator outlet. But this set of BC is suitable only near the choke region as is shown on Fig. 4. It is evident that working point is not well defined in flat high pressure region. It will depend on actual performance line shape and numerical
fluctuation during iteration process which one will be “selected” or (more likely) if solutions will be of unstable behavior. One can suggest that “mass flow inlet” BC seems to be a solution of trouble described above. Nevertheless one should keep in mind that it would be necessary to switch between appropriate BC whenever the working point is moved form choke region to flat high pressure region and vice versa. It is not really convenient and it not permits automatic computation procedure (using journal) in case that the performance line is not known prior to computation. Furthermore there are further consequences of using “mass flow inlet” BC as total pressure at the wheel inlet is no more constant and one should handle it properly.

More convenient way is to use “outlet vent” BC at the domain outlet. The intersection of CC performance line and BC characteristic is well defined in the all operating regimes which makes possible to use automatic computation procedure. BC course is defined by combinations of static pressure and pressure loss coefficient Kp. Value of static pressure can be kept to be equal to inlet pressure and working point is moved only by setting different value of Kp. “Outlet vent” BC is acting as well as a throttle-valve and it has benefit in stabilizing solution and it has a native behavior during initializing phase of the computations (as will be discussed below). One should only keep in mind that “outlet vent” BC forces the flow field to be uniform at the boundary and as a consequence the flow field up-stream BC is affected. It means that computation domain has to be prolonged enough to prevent affection the area of interest.

**Computational procedure**

A special treatment is necessary during initial phase of computation in case of high pressure CC. Flow direction changes its sense several times and it usually leads into instability. Surge line is
quite close to choke region in the area of high pressure ratio and flow fluctuations behind surge line are very pronounced thus solutions is forced to “go through” this severe conditions several times until operating point is “found” and one has no guarantee that solutions will be able to “escape” unstable performance region. A procedure how to overcome the difficulty of initial phase and to get converged solution is described below.

1. Computation case is properly established, rotating speed is set to very low value (~10% of design speed) assuring that operating regime will be (due to low velocities) of incompressible nature with no surge behavior. Pressure loss coefficient Kp is set to low value from 5 to 10 in order to be in choke region.

2. Initialization using full multigrid is useful but not necessary. Near to zero initialization is good enough in many cases.

3. Approximately 100-200 iterations are necessary to get stabilized (not converged) solution while CFL number has been gradually increasing. Density based implicit solver is preferable option and 1st order upwind numerical scheme is used for stability benefit.

4. Rotating speed is increasing step by step until desired level is reached. It can take approx 10 steps 20-30 iterations each. Solution is switched to 2nd order of accuracy after stable behavior at design speed is observed.

5. Solution is converged after all relevant monitors are stabilized. It is not enough just watch the residuals. It is necessary to define extra surface monitors, 4 is the minimum: mass flow at the inlet, overall mass balance, total pressure and temperature at the stage outlet. Total temperature monitor (efficiency) is typically stabilized as the last one.

6. Additional working points of the given speed line are computed step by step by increasing loss coefficient Kp as log as is desired or getting stable solution became impossible.

Unstable behavior of the solution does not mean that the surge line is reached. It is hardly possible to establish surge line based on steady state calculation. But it is tough problem even in case of fully time transient calculation using sliding mesh approach.

Another case settings

Density based implicit solver and 2nd order of accuracy was used for getting converged solutions presented here. Air was treated as ideal gas and Realizable k-ε model of turbulence in combination with enhanced wall treatment was used as viscous model.

Overall performance

Computed overall performance characteristics (red lines with points) in comparison to the experimental data (black lines) are given on Fig.5 for three speed lines. Maximum speed line is extremely narrow (small surge margin) and it is tough problem to get a point in high pressure flat region. This is why the last point has somewhat higher deviation from experimental line. Nevertheless maximum difference between CFD an experimental data in pressure ratio is 4 % and in efficiency is 2 % (absolutely). If one take into considerations all possible imperfectness in computational model (from ideal gas to mixing plane and many others) uncertainty of experiment
and real geometry effects (roughness, tip clearance, tolerances and so) it is hardly possible to get much more better results by means of any other computational procedure (in sense of overall performance).

Pressure and entropy contours on CC stage walls are shown on Fig.6 and flow field within the compressor stage is shown on Fig.7.

Fig.5 Overall performance characteristic

Fig.6 Pressure and entropy contours on CC walls
Flow conditions in different operating points are apparent from Fig. 8 where relative Mach number contours within wheel inducer are given in three span-wise sections. Left picture shows choke conditions with high Mach number regions and shock waves. On the other hand pronounced flow separation is evident in case of near surge operations on the left picture.
Conclusion

The results performed numerical investigation and comparison to the experimental data demonstrated the capability of established cost effective computational procedure using FLUENT code to predict overall performance characteristic of high pressure ratio centrifugal compressor. Difference between CFD and experimental data in pressure ratio and efficiency was rather small and it would be not easy to obtain significantly better results (in sense of overall performance) by means of any other more demanding computational procedure.

Acknowledgements

The work presented here was carried out in the frame of the NEWAC Project (FP6-030876) with co-funding from the European Commission within the 6th Framework Program

Reference