Numerical investigation of the flow instabilities in centrifugal fan

BRANIMIR MATIJAŠEVIĆ Faculty of Mechanical Engineering and Naval Architecture Chair of Turbomachinery Ivana Lučića 5, 10000 Zagreb CROATIA

STANISLAV SVIDEREK Faculty of Mechanical Engineering and Naval Architecture Chair of Turbomachinery Ivana Lučića 5, 10000 Zagreb CROATIA TIHOMIR MIHALIĆ Faculty of Mechanical Engineering and Naval Architecture Chair of Fluid Mechanics Ivana Lučića 5, 10000 Zagreb CROATIA

Abstract: It is well known in the practice that rotor blade geometry and shape of channels influence flow pattern and stability of the centrifugal fan. Practical recommendations on how to reduce the instabilities as source of energy losses and noise are based on experience and experimental approach. The research is based on the numerical approach, instead of experimental one, for flow stabilities investigation. The calculation of flow instabilities for two different types of the shroud and two different number of blades are performed. The results are transferred, via FFT analysis in frequency domain, where additional conclusions about instabilities phenomena are carried out.

Key-Words: Radial fan, turbulence modeling, unsteady, DES, FFT

1 Introduction

Fluid flow in turbomachinery is highly turbulent and unsteady. Turbulent flows have been studied mostly by means of experimental fluid dynamics research. Measuring techniques that allow sampling of flow characteristics in a single point have increased the range of experimental measurements. Experimental research has still its limitations, especially in the field of rotating turbomachinery, ranging from the time required to produce experimental model and its cost to the difficult, if ever possible, measurements of flow characteristics in fast rotating system.

Hence, experimental methods in turbomachinery applications are often limited to measurement of integral quantities, which do not provide insight of the internal flow structures. Experimental research remains still very important for the validation of new models.

Computational fluid dynamics is more flexible and cost effective than experimental methods. It has advantage that any flow quantity may be sampled at any point in the field. Instantaneous field values can be obtained for the whole domain, thus providing full insight of the flow characteristics and intricate flow structures. Major drawback of the numerical simulations is their inability to guarantee accuracy under all conditions. Strictly speaking, results of numerical simulations must be validated by experimental data.

Still, even without experimental support, numerical simulations can be valuable in design analysis. They provide useful insight of flow characteristics and comparison between different geometry configurations.

In this study, unsteady turbulent fluid flow in a radial fan impeller is investigated by use of computational fluid dynamics methods. Two geometry configurations are compared at several flow rates. Results include frequency analysis of the flow unsteadyness.

2 Turbulence Modeling

First approach to the practical numerical simulation of turbulent flow in a turbomachine could be, the least expensive and fastest, solving of steady Reynolds Averaged Navier-Stokes (RANS) equations in rotational frame of reference, coupled with appropriate k- ϵ turbulence model with wall functions.

In RANS, flow variables are split between one time-averaged mean part and one turbulent (fluctuating) part. The latter is modelled with a turbulence model such as k- ϵ , Spalart-Allmaras or Reynolds Stress Model.

The RANS results show acceptable agreement with analytically predicted as well as experimentally measured integral quantities such as flow rate, head and efficiency in the vicinity of the design working point.

However, in off-design working conditions, when unsteadiness and flow separation become significant, this approach gives less acceptable results. Steady RANS equations are designed to smear out not only the high frequency unsteady turbulent fluctuations but also the low frequency main flow unsteadiness. The wall functions which assume attached boundary layer flow do not cope well with flow separation.

Another approach to modeling of turbulent flows is Large Eddy Simulation (LES). Idea is to compute the contributions of the large, energy-carrying structures to momentum and energy transfer and to model the effects of the small structures, which are not resolved by the numerical scheme. In LES only small, isotropic turbulent scales are modeled. Large scale fluctuations, which strongly depend on the geometry and boundary conditions are fully resolved.

LES is based on a spatial filtering, which decomposes any flow variable into a filtered (large-scale, resolved) part and into a sub-filter (unresolved) part. In finite volume method, a volume filter defined by computational grid is used to form governing equations. Volume filter filters out fluctuations in space with scales smaller then grid (cell) size. Furthermore, filtering is implicit which means that equations are discretized and filtered over the same control volume. Turbulent scales which cannot be resolved by the grid are then modeled by subgrid scale model.

LES is less demanding on computational resources than DNS since not all scales need to be resolved, but considerably more demanding than steady state RANS due to higher resolution requirements and its unsteady nature. Furthermore, presence of solid walls in the flow domain requires very fine grid which makes LES impractical in modelling of wall bounded flows at higher Reynolds numbers.

In order to avoid requirement of very fine mesh near the wall and extend LES to high Reynolds number flows, several methods have been developed. These include Unsteady Reynolds Averaged Navier-Stokes (URANS), Detached Eddy Simulation (DES) and Hybrid LES-RANS.

URANS equations are usual RANS equations but with the transient term retained. However, URANS is not a simple transposition of RANS (steady-state flows in statistical equilibrium) in unsteady flows. Turbulence models and discretization schemes used in URANS are adapted to unsteadiness and detachment of flow. URANS models are able to accurately predict flows under adverse pressure gradients, including flows with small separation bubbles.

DES is a mix of LES and RANS. Dominant eddies in massively separated flows are highly geometry-specific and have not much in common with the standard eddies of the thin shear flows RANS models are designed to model. The general idea of DES is to model attached boundary layer vertices in RANS mode and capture the outer detached eddies with LES. Small, subgrid scale vertices in LES region are also modelled, but they have much less influence then the boundary layer vertices. The name Detached Eddy Simulation owes its name to the fact that only detached eddies are simulated while other are modelled.

2.1 Detached eddy simulation

DES is an approach whereby RANS turbulence modeling and mesh spacing is used in the boundary layer, while LES is employed in the core and separated regions of the flow. Thin shear layer turbulence model controls the solution in the RANS region. In the LES region, turbulence model has little effect since the larger, energy carrying vortices are resolved. Through the use of the near-wall RANS model, DES reduce computational cost over conventional LES by removing wall parallel resolution requirements. Typically, wall normal resolution requirements remain still high due to the required spacing in wall units (y+) of the order of one, similar to the low-Reynolds number RANS turbulence models.

DES model proposed by [8] solves set of URANS equations:

$$\frac{\partial U_j}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \bar{U}_i}{\partial t} + \frac{\partial (\bar{U}_i \bar{U}_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_i} + \nu \frac{\partial^2 \bar{U}_i}{\partial x_j \partial x_j} - \frac{\partial \overline{u_i'' u_j''}}{\partial x_j}$$
(2)

where the Spalart-Allmaras turbulence model [7] is used to compute Reynolds-stress tensor (3):

$$-\frac{\partial \overline{u_i'' u_j''}}{\partial x_j} = -\frac{\partial}{\partial x_j} (\overline{U_i U_j} - \overline{U}_i \overline{U}_j) = \frac{1}{\rho} \frac{\partial}{\partial x_j} \tau_{ij}^R \quad (3)$$

The standard Spalart-Allmaras model uses the distance to the closest wall as the definition for the length scale d, which plays a major role in determining the level of production and destruction of turbulent viscosity. The DES model replaces d everywhere with a new length scale \tilde{d} , defined as:

$$\tilde{d} = \min(d, C_{\text{des}}\Delta) \tag{4}$$

where the grid spacing, Δ , is based on the largest cell edge length in the x, y, or z direction, $\Delta = \max(\Delta x, \Delta y, \Delta z)$. The empirical constant C_{des} has a value of 0.65. In the boundary layer $d < C_{\text{des}}\Delta$ and model operates in RANS mode. Outside of the boundary layer $d > C_{\text{des}}\Delta$ model operates as subgrid-scale LES model.

In standard DES, RANS portion of the simulation is strongly grid dependant. RANS requires that grid spacing in both wall parallel directions be larger then the boundary layer thickness at that wall location. This is a restrictive lower limit for the grid resolution, which may easily be violated. Further grid refinement



Figure 1: Two impeller shapes, flat and cone shaped shroud, with major dimensions

below this limit, which is equivalent to the placing of the LES/RANS interface deeper into the boundary layer, can result in non-physical grid induced separation.

3 Model Details

An inlet/exit diameter ratio $D_1/D_2 = 0.56$ is upper allowable limit for radial impellers with constant impeller width (flat shroud) [1]. For higher values of D_1/D_2 ratio, a reduction of impeller width is recommended in order to keep diffusivity ratio of the blade channel under control. An impeller with inlet/exit diameter ratio $D_1/D_2 =$ of exactly 0.56 is chosen to compare effect of the reduced impeller width on flow characteristics of the impeller.

Thus, two geometrically different configurations (Figure 1) are modeled, one with flat shroud and the other with conical shroud.

Basic geometry parameters are common to both considered impellers: diameter of the suction tube $D_s = 500$ mm, impeller inlet diameter $D_1 = 560$ mm, impeller inlet width $b_1 = 110$ mm, impeller exit diameter $D_2 = 1000$ mm. Blades are formed as single circular arc with 732.2 mm radius. Inlet angle $\beta_1 = 30^\circ$, outlet blade angle $\beta_2 = 44^\circ$.

Because of rotational periodicity of the impeller only a single blade channel needs to be modeled. Periodic boundaries form an angle which is equal to 2π divided by number of blades.

In order to allow simulation of only one portion of the domain, instead into volute, impellers discharge into a vaneless diffuser of $R_3 = 800$ mm outer radius. Width of diffuser is matched to the exit impeller width. On the upper end of the diffuser, there is an



Figure 2: Computational domain and boundary conditions. Single cyclic symmetric sector which contains one blade passage is modelled

outlet formed as axial 12.5 mm wide annulus on the both sides. Such outlet is chosen in order to minimize outlet boundary impact on upstream flow, to prevent possible backflow condition on outlet and to secure the same outlet cross-section area for any diffuser width (Figure 2).

Computational domain upstream of the impeller is extended to form a suction pipe which is widened at the inlet, to minimize effect of inlet boundary on the flow inside impeller. Inlet is formed as a spherical surface.

Rotational moving of the impeller is numerically taken into account by usage of multiple reference frames. Inlet section of the suction tube as well as the outlet section of the diffuser are calculated in stationary reference frames. Blades as well as other moving walls (backplate and shroud) are fixed to the moving reference frame. This approach seems to be reasonable since there is no stationary blades before or after the impeller. In fact, whole domain could be represented by a single rotating reference frame.

Angular velocity of the impeller was set at $\omega = 25 \times 2\pi \text{ s}^{-1}$, except in one comparison case where it was set at $16.66 \times 2\pi \text{ s}^{-1}$.

A constant total pressure and normal velocity direction are prescribed on inlet. On outlet, static pressure is given as a function of normal velocity component and prescribed nondimensional pressure drop coefficient in the form:

$$p_{out} = k_L \frac{1}{2} \rho v^2$$

Near design flow rate was achieved with the pressure drop coefficient value of $k_L = 10$. Values of 25 and 200 were used to simulate flow regimes at smaller flow rates. This boundary condition mimic behavior of throttling valve and approximates natural outlet



Figure 3: Mesh of the blade leading edge. Cells near the solid walls have very high aspect ratios as required by the DES turbulence model.



Figure 4: Signal of pressure and relative velocity is taken at sampling point inside the blade passage

conditions better than fixed static pressure especially in vicinity of zero flow rate.

The computational grid was generated using the Gambit preprocessor. Domain is discretized into 800 000 hexahedral cells. Approximately half of the grid points is found in the near-wall region. Spalart-Allmaras DES turbulence model used in this study require very fine wall-normal grid spacing, of the order of 1 in wall units. On the other hand, the wall-parallel spacing must be larger then local boundary layer thickness. This requirements lead to wall adjacent cells with very high aspect ratio (Figure 3).

Numerical simulations were carried out using Fluent 6.2, finite volume based unsteady solver of the pressure-correction type.

Results were obtained with Spalart-Allmaras DES [8] turbulence model. Bounded central differencing scheme for spatial discretization of the momentum, second order pressure interpolation scheme and second order implicit temporal discretization for time-advancement were used.

Unsteady calculation were carried out with a physical time step of 5×10^{-5} seconds. In order to obtain frequency data, static pressure and relative velocity magnitude were sampled at each time step in a point inside the blade channel. The sampling point is located in the middle of the channel, at 368 mm ra-



Figure 5: Mass flow rate history for different values of k_L



Figure 6: Static pressure signal

dius (Figure 4). Each case was sampled at least 5000 times.

4 **Results**

After short transient period, quasistationary flow is established in the blade passage. While global mass flow rate fluctuates in time only slightly (Figure 5), local static pressure and relative velocity oscillate considerably (Figure 6).

Examination of the flow field shows that a zone of strong flow separation can be identified (Figure 7). This flow zone is characterized by low relative velocity (it travels with blade) and lot of small swirls. Instantaneous structures of separated flow are clearly visible by relative velocity isosurfaces (Figure 8). Flow separation is more pronounced as flow rate gets smaller as there is less and less fluid to occupy available cross section of the blade channel.

Pathlines at the midsurface of the blade passage at different flow rates (Figure 11) demonstrate spreading of the separated flow across the blade passage and its growing as flow rate decreases. Small swirls are clearly visible. Homogeneous flow occurs only in



Figure 7: Schematic of the flow separation bubble in meridional view



Figure 8: 3D view of separated flow structures in the blade passage

small area near the blade passage eye.

Impeller with conical shaped shroud supresses spreading of the separated flow bubble inside blade passage. Flow is more homogeneous and regular then in the case of flat shroud. As the outlet cross section area is lower with conical shroud impeller, diffuser effect of the blade passage is also reduced. Pathlines (Figure 12) confirm more regular flow patterns then in the case of the flat shrouded impeller.

Axial view of the blade passage reveals another flow pattern, shedding of vortices which travel downstream along the suction side of the blade. Vortices rotate in the direction of impeller rotation (Figures 9 and 10). Those vortices are so dominant flow pattern that they are resolved even by k- ϵ URANS turbulence model which tends to smear out all flow irregularities.

Origin of these vortices is not completely clear. At lower flow rates there exists negative angle of attack between blade direction and flow direction. In this case vortices are shedding from the blade leading edge. At higher flow rates, when the flow is parallel to the blade direction, vortices emerge further downstream of the leading edge and they seem to be con-



Figure 9: Schematic of the vortex shedding in a blade passage



Figure 10: Pathlines in axial view reveal vortex shedding patterns



Figure 11: Pathlines in passage midsurface between two blades at various flow rates, flat shrouded impeller



Figure 12: Pathlines in conical-shrouded impeller at various flow rates



Figure 13: Flat-shrouded impeller: effect of flow rate on static pressure frequency spectrum



Figure 14: Conical-shrouded impeller: effect of flow rate on static pressure frequency spectrum



Figure 15: Effect of impeller shape on static pressure frequency spectrum

nected to the flow separation mentioned earlier.

Vortex shedding causes strong pressure and relative velocity oscillation inside the blade passage. It takes place in all simulated cases with almost the same frequency as shown by the FFT analysis of the pressure signal. Intensity of the oscillations varies across the different cases though.

FFT analysis of the pressure signal reveals continuous spectrum of frequencies which is typical for turbulent flows. Dominant frequencies which correspond to the biggest turbulent flow structures can be recognized.

At higher flow rates, vortex shedding frequency is virtually independent of impeller shape and flow rate. Spectral distribution, or shape of the FFT plot, is virtually independent of flow rate and impeller shroud shape.



Figure 16: Effect of rotational speed

Significant difference to this rule exhibits frequency characteristics at small flow rate (Figure 13) where spectral distribution differs, dominant frequency is higher and amplitude smaller then at higher flow rates. More research is needed to explain this behavior, but this could result from the fact that separation bubble spreads over the sampling point in this case.

Frequency spectra comparison between the two impeller shapes uncovers almost fourfold difference in amplitude of pressure oscillations, while at the same time, spectral distribution and dominant frequency are the same. This reflects difference in relative velocity amplitude and confirms earlier observation about more regular flow inside conical shaped impeller.

Effect of the rotation speed (Figure 16) is in accordance with similarity laws.

5 Conclusion

Unsteady simulation of the turbulent flow inside radial fan impeller is performed. Flow instabilities are investigated for two different impeller shapes. DES turbulence model is used. Results are transferred into frequency domain using fast Fourier transformation.

Results of the simulations show that the fluid flow is more regular and stable with cone shaped impeller shroud. Further research is necessary to reveal effects of the flow instabilities on performance losses and efficiency of the impeller. On basis of the calculated flow patterns it is possible to suggest new impeller shapes and investigate flow inside them.

FFT analysis provides additional knowledge of unsteady flow patterns. It is a good index to temporal accuracy of numerical models. It is easier to measure pressure fluctuations in a single point then instantaneous pressure and velocity fields inside a volume of interest. Thus FFT can be useful in experimental validation of the CFD results.

Numerical CFD analysis, used with caution, provides detailed insight of unsteady flow inside the turbomachine. Caution is necessary because of imperfection of mathematical models and numerical algorithms. Experimental validation is highly needed to assure confidence in numerical results.

Acknowledgments: During the preparation of this paper computational resources of the Isabella cluster at Zagreb University Computing Center (Srce) were used.

References:

- [1] B. Eck, Fans, *Springer Verlag*, Berlin/Heidelberg/New York, 1973
- [2] C. Pfleiderer, Die Kreiselpumpen, *Springer Verlag*, Berlin/Göttingen/Heidelberg, 1955
- [3] J. H. Ferziger and M. Perić, Computational Methods for Fluid Dynamics, 3rd. ed., Springer-Verlag, New York, 2002
- [4] L. Davidson, Lecture Notes on Turbulence Modelling, Chalmers University of Technology, www.tfd.chalmers.se/doct/comp_turb_model/
- [5] J. Blazek, Computational Fluid Dynamics: Principles and Applications, *Elsevier*, 2001
- [6] T.H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu, A new k-epsilon Eddy-Viscosity Model for High Reynolds Number Turbulent Flows -Model Development and Validation, *Computers Fluids* 24(3), 1995, pp. 227–238.
- [7] P.R. Spalart and S.R. Allmaras, One-equation turbulence model for aerodynamic flows, *In Proc. AIAA ASM*, Reno, NV, 1992
- [8] P.R. Spalart, W.-H. Jou, M. Strelets and S.R. Allmaras, Comments on the feasibility of LES for wings, and on hybrid RANS/LES approach, *In Advances in DNS/LES*, (ed. C. Liu and Z. Liu), Columbus, OH, Greyden, 1997
- [9] S. Sviderek, Diploma Thesis, Faculty of Mechanical Engineering and Naval Architecture Zagreb, Zagreb, 2001