CDF CALCULATION OF RADIAL FAN STAGE WITH VARIABLE LENGTH OF SEMI BLADES
SVOČ – FST 2012

Ing. Roman Gášpár,
Západočeská univerzita v Plzni,
Univerzitní 8, 306 14 Plzeň
Česká republika

NOMENCLATURE

A – area
\( A = \pi D^2 / 4 \)

\( p_T; T_T; M \) – total pressure, total temperature, moment,
\( (p)' , (T)' , (M)' \)

\( \eta \) – isentropic efficiency
\( \eta(p) = q \cdot \Delta p_T / (M \cdot \omega); \eta(T) = (T^1 - T^2) - (T - T^{is}) \)

\( \varphi \) – flow coefficient
\( \varphi = \dot{m} / A u_r \)

\( \psi \) – pressure coefficient
\( \psi = 2 \cdot \Delta p_T / \rho \cdot u_r^2 \)

INDEX

\( (') \) – average over the entire measuring plane
1,2,3 – measuring planes - stage inlet, stage outlet, case outlet

ABSTRACT

The study is focused on numerical investigation of complex 3D model of a centrifugal fan. A numerical model includes an impeller and a spiral casing of forward curved blades of a centrifugal stage. In design process flow simulation data with an optimized method will be used. The commercial code FLUENT will be applied. In the solution application of passive flow control in the impeller by means of splitter blades of a high pressure fan is considered.

INTRODUCTION

Aerodynamic performance of a centrifugal fan was computed by means of commercial code Fluent 6.3.26. This study has described one part of current numerical investigation. First, initial, steady computation and following unsteady simulations were carried out. Utility necessary for data evaluation was created under Matlab. The evaluation procedure corresponds with the experimental data evaluation. The computed data will be compared with experimental measurements. The article is focused on computational investigation at design and off-design points of a centrifugal fan. Currently, there is a universal rise of demands on a centrifugal compressor and industrial centrifugal flow fans. Requirement both for pressure ratio and efficiency was increased. It is very important to determine aerodynamic characteristics of new turbomachine in a design process. Commercial CFD codes are now used for solution of this task e.g. [1], [2], [3]. The presented paper concentrates on prediction of performance parameters of centrifugal fan with the application of the Fluent – code.

MODEL

Mathematical model:

\(^\uparrow\) System of Navier-Stokes equations for turbulent flow of compressible fluid.
\(^\uparrow\) State equation of the ideal gas added to the equation set.
\(^\uparrow\) Viscosity and the heat capacity are considered constant.
\(^\uparrow\) Differential transport equations complete the system. Modeling of turbulence – RSM
\(^\uparrow\) Non-equilibrium wall functions.

Numerical model:

\(^\uparrow\) Segregated solution Method – FLUENT 13
\(^\uparrow\) Both order accuracy in time and fixed coefficients was used.

Geometry, mesh and boundary conditions:

\(^\uparrow\) Hybrid mesh including 8 million cells (Fig. 1)
\(^\uparrow\) Mesh was adapted close to rotor blade wall to specify calculation of boundary layer phenomena. (Fig. 2)
\(^\uparrow\) Moving mesh B-C was used for simulation of rotor blade rotation
Mass flow inlet and atmospheric pressure outlet were used.
Different lengths of semi-blades were used for CFD calculation (Fig. 5)

Fig. 1: Geometry and boundary conditions of centrifugal fan

RESULTS
The numerical simulations started as steady simulation. During initial calculation the residuals and pressure coefficient were monitored. As soon as the trend was stabilized we stopped the steady computation.

Then unsteady computation was started (switch to moving grid) [5]. During unsteady simulation the stabilized trend of residuals after several revolutions could be expected. Several values (mass flow, efficiencies, pressure coefficients etc.) were monitored. When periodical solution was obtained, the numerical simulation was stopped (Fig. 3).
The actual geometry contains 12 blades and 12 semi-blades (Fig. 1). The displacement of blade by the distance between two blades was defined in 20 time steps and each time step was calculated in 35 iterations. One rotor revolution was broken into 480 time steps. Numerical simulation seems to be convergent after 10 revolutions which results in approximately 5000 time steps (Fig. 5). Subsequently, slight oscillation occurs. For precise data analysis it is necessary to correctly averaged data through the blade surface and time. Graphs in fig. 5 shows impact of accuracy order settings of non-stationary solver to convergence of solution. Instability of solution with second order accuracy near by design point is obvious in fig. 4 (bottom).

Fig. 3 Evolution of mass flow rate

Fig. 4 Blade geometry variants
CONCLUSIONS

- Aerodynamic performance of several variants of radial compressor was predicted by means of the CFD code FLUENT 13.
- Effect of length of semi-blade was determined by CFD calculation:
  It is more significant at higher mass flow coefficient (Fig. 6). Optimal length of semi-blade is observed around ½ of blade length (Fig. 4).
- Precession of numerical simulation of off-design point of aerodynamic characteristic are more sensitive to boundary conditions settings as inlet pressure or mass flow, temperature and turbulence profile, inlet and outlet positions and geometry simplification, wall function, etc. [4].
- Default settings of numerical model have been too dissipative and results are unrealistically stable and optimistic.
ACKNOWLEDGEMENT
Research was supported by Grant of Czech Republic - Project No. 101/09/1959.

REFERENCES


