UDC 621

FILIP LISOWSKI, PROF. DR HAB. INŻ. EDWARD LISOWSKI
Cracow University of Technology
Computer Graphics and Modeling Students Research Group

CFD MODELING OF RADIAL ROTATING IMPELLER FOR AIR CONDITION AT RAIL VEHICLE

Мета статті - створення в Solidworks geometric моделі радіального impeller, що обертається, використовуваного у вентиляції і систем кондиціонування в транспортних засобах рейки і виконати аналіз повітряного потоку КОСМОС Flowworks.

CFD, моделювання завдяки дискретизації і числовим методам для вирішення рівнянь роблять можливою оцінку приблизного значення параметрів потоку - швидкості, тиску, температурі і іншим параметрам, важливим протягом геометричної оптимізації.

Цель статьи – создание в SolidWorks geometric модели радиального вращающегося impeller, используемого в вентиляции и систем кондиционирования в транспортных средствах рельса и выполнить анализ воздушного потока с КОСМОС FlowWorks.

CFD, моделирование благодаря дискретизации и числовым методам для решения уравнений делают возможными оценку приблизительного значения параметров потока – скорости, давлению, температуре и другим параметрам, важным в течение геометрической оптимизации.

Purpose of paper is creating at SolidWorks geometric model of radial rotating impeller using at ventilation and air conditioning systems in rail vehicles and execute analysis of air flow with COSMOS FlowWorks.

CFD modeling thanks to discretization and numerical methods for solving equations make possible evaluate approximate value of flow parameters like velocity, pressure, temperature and other parameters important during geometrical optimalization.

1. Introduction

Design of modern light rail vehicles burgeon constantly. Low floor, ergonomic space for passangers, efficient ventilation and air condition systems are standards. Using all this innovation is relevant with design issuees and also with projecting air condition and ventilation structures. Despite units from this structures are hidden – so invisible for passengers they need a lot of space. Central units of air condition system are based in the different places of roof so have to be some ventilation channels to transport air around the vehicle's inside. One of basic component of ventilation systems are fans. Depending on work structure and expected flow parameters fans have many different designs of impellers. One of most popular fan using, cause of good flow parameters and quite small dimensions, at modern tram construction is radial centrifugal fan.

1.1 Definition of radial centrifugal fan

Fan is a flow machine for creating current of air and transport it inside ore outside room. At radial centrifugal fans energy is carry to gas in centrifugal direction. Flow direction change to axial only inside impeller's casing.

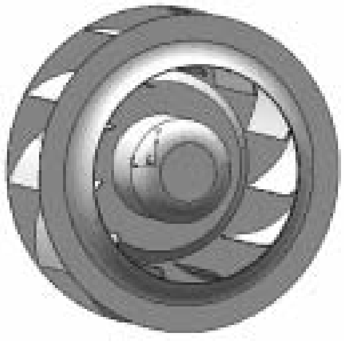


Fig. 1. Radial centrifugal fan impeller

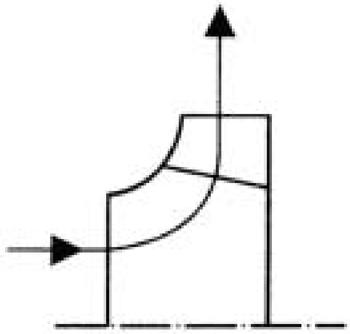


Fig. 2. Air flow trajectory through impeller

2. Geometrical model of fan impeller

Geometrical model of rotating impeller was prepared at CAD application SolidWorks. Assembly used in flow analysis consist of four parts. The main and at once the most complex component is impeller (1). It is based inside cylindrical casing (2). Fan casing is not a standard part of fan but backup geometry that is use to define border condition for air flow. Cylindrical surface (3) is not a wall for air because it is out of flow. Intel surface (4) is also backup geometry out of flow. It is necessary to define inlet conditions.

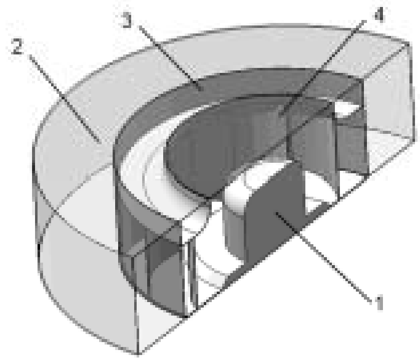


Fig. 3. Cross section of geometrical model of

3. CFD analysis

For CFD (Computation Fluid Dynamics) analysis was used toll from COSMOS platform FloWorks. It is toll for fluid-flow simulation integrated with SolidWorks application. With its CFD analysis capabilities, it is possible to simulate liquid and gas flow in real world conditions, run “what if” scenarios, and quickly analyze the effects of fluid flow, heat transfer, and related forces on immersed or surrounding components.

centrifugal impeller

4. Analysis definition

To define flow analysis following border conditions were determinated (Fig.4.):

a) Inlet efficiency $0{,}3\left[\frac{m^3}{s}\right]$

b) Standard pressure $101325[Pa]$

c) Fixed walls

d) Impeller rotation $1800\left[\frac{obr}{\min}\right]$

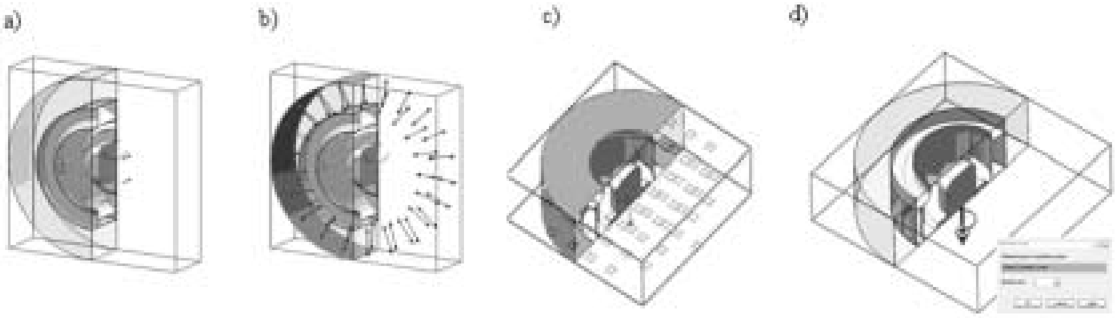


Fig. 4. Definition of border conditions

Fluid is set as air with temperature and pressure. Those are default parameters from COSMOS data base. There is also possibility to define user parameters.

5. Analysis results

In the first step of analysis was generated three-dimensional mesh grid (Fig.5). In this example 3D grid consist of three types of finite volumes.

Fluid cells – cells where fluid flow is detected

Partial cells – backup cells, generated in places with complex geometry

Solid cells – cells where fluid flow is not detected

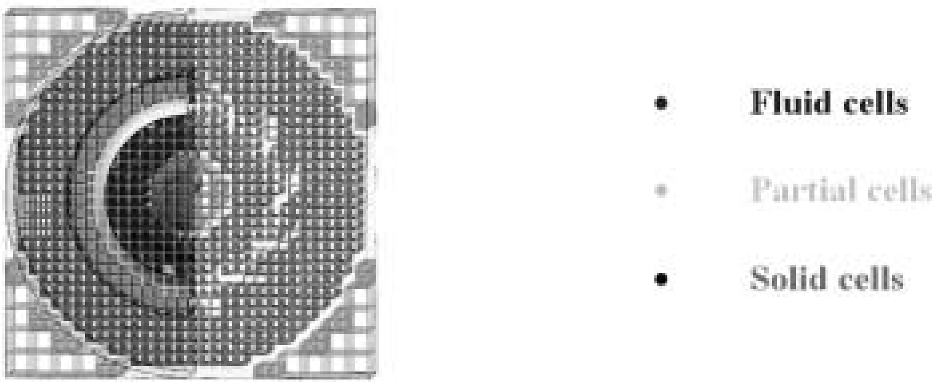
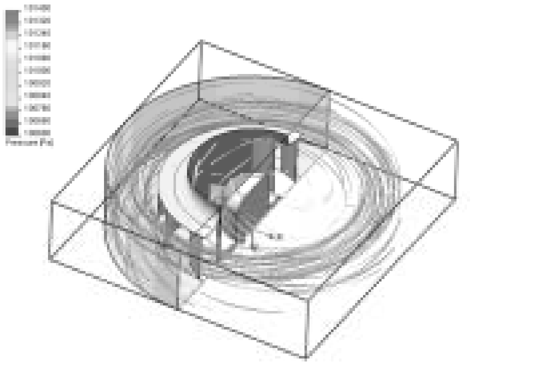


Fig. 5. three-dimensional mesh grid

Next step is generating flow trajectories (Fig.6.) and maps of flow parameters like: average flow pressure (Fig.7) and velocity vectors field (Fig.8) in normal section in the middle of impeller thickness and also surface map with average pressure existing on rotating impeller (Fig.9).



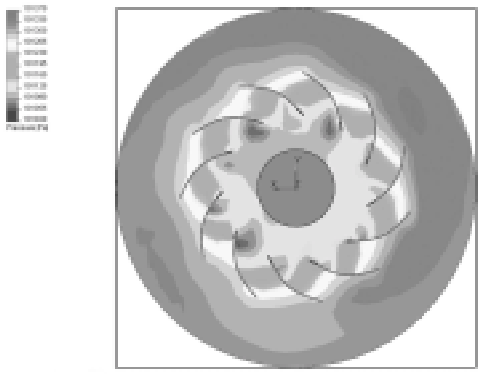
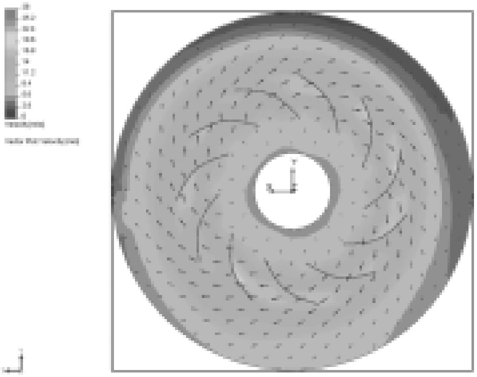


Fig. 6. Average pressure distribution along flow trajectories

Fig. 7. Average pressure distribution in normal section



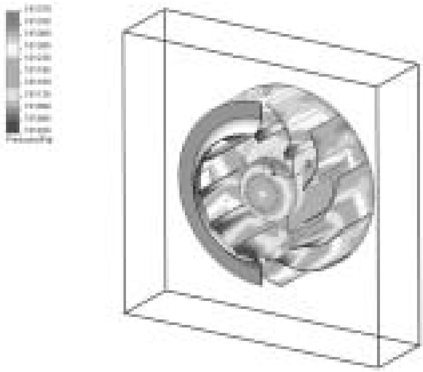


Fig. 8. Velocity vectors field in normal section

Fig. 9. Average pressure distribution in the surfaces of impeller

6. Automatic efficiency calculation

FloWorks makes possibilities to declare mathematic equations. Thanks to this option that base on evaluated average value of flow parameters impeller efficiency was found.

$$\eta = \frac{N_u}{N_s}$$

$$\eta = \left| \frac{\left(P_{inlet} - P_{outlet}\right) \cdot Q}{\Omega \cdot M} \right|$$

η - impeller efficiency
N_u – power output
N_s – engine shaft power
P_{inlet} – inlet pressure
P_{outlet} – outlet enviroment pressure
Q – intel volume flow
Ω – impeller rotation
M – torque on impeller

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
Equation Goal 3	[ ]	0,828180789	0,823269	0,814043	0,832973	100	Yes	0,007088807	0,087928244

Iterations: 79
Analysis Interval: 21

Fig.10. FloWorks result window

Calculated impeller efficiency is about 82 %. It is a bit higher then in real environment. It result from analysis idealization. Efficiency of centrifugal axial impellers, depending of ventilation structures architectures, is from 60 to 90 %

7. Conclusions

Preparing geometrical model and CFD analysis execution let to found flow trajectories, distribution of average pressure along those trajectories, flat maps with flow parameters distributions, velocity vector field and surfaces map of average pressure distribution on rotating impeller. The last one is very useful during designing processes because it clearly illustrate places where local extremes exists. Additionally based on thermodynamic equation impeller efficiency was found.

Using integration CAD application with CFD system, correct definition of flow conditions and also equation definition (optimalization functions) make possible automatic calculation variants of solutions.

Literature

1. Stanisław Fortuna: Wentylatory, TECHWEBT, Kraków 1999.
2. EbmPapst: Centrifugal fans and blowers.

3. Praca zbiorowa: Heating, ventilation and air conditioning.

Надійшла 2.12.2008 р.