Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 2, 2008, vol. LIV,
article No. 1630

Zdeněk PORUBA[*], **Jan SZWEDA**[**]

FLOW PROBLEM IN THERMAL MOTOR ANALYSIS

ÚLOHA PROUDĚNÍ V TEPELNÉ ANALÝZE MOTORU

Abstract

The paper presents the relations used for CFD simulation of the flow around the body. The procedures for creation of practically usable computer model of bigger mechanical parts and application of its boundary conditions are described, obtained results are presented. The aim of the work was the analysis of temperature field and cooling parameters of current motor geometry with expectation of further optimization to achieve better cooling power and to reduce the mass of the motor.

Abstrakt

Předložený článek prezentuje vztahy uživané při numerické CFD analýze obtekaného tělesa. Jsou popsány postupy pro tvorbu prakticky použitelného počítačového modelu strojního celku větších rozměrů, aplikace okrajových podmínek a jsou prezentovány získané výsledky. Cílem práce byla analýza teplotního pole a parametry chlazení současné geometrie motoru s výhledem na případnou optimalizaci tvaru za účelem lepší chladivosti a snížení hmotnosti motoru.

1 INTRODUCTION

The problem of inadequate high temperature loading of mechanical part can be often observed in engineering practice. In case of electromotor the undesirable conductivity aggravation and additional temperature loading of mechanical parts can happen due to the high temperatures. Because of this fact the designers are forced to solve the problem of motor temperature minimisation. Performance of mathematical simulation of those processes is rather complicated discipline. It is necessary to dispose of both theoretical, necessary for choice of proper mathematical model, and practical knowledge to create the computer model itself. An important role play also related costs (proper software, human resources) and last but not least a big time severity of whole numerical simulation process. The mentioned reasons predestine the more complicated problems in technical practice to be solved within the research activity at the universities. The aim of the work was the analysis of temperature field and cooling parameters of current used motor geometry with expectation of pertinent shape optimisation to achieve better cooling power and to reduce the mass of the motor.

The numerical modelling of many physical phenomena is closely related to the modelling of certain form of motion by mathematical instruments. The mathematical model is based on the definition of equation describing processes such as laminar and turbulent flow even in more complicated geometries, compressible and incompressible flow, stationary and non-stationary flow, heat transfer and heat convection. Respecting the fact that mentioned processes are in generally three-

---

[*] Ing. Ph.D., Department of Mechanics, Faculty of Mechanical Engineering, VŠB-Technical University of Ostrava, 17. listopadu 15, Ostrava, tel. (+420) 59 732 3228, e-mail zdenek.poruba@vsb.cz

[**] Ing. Ph.D., Department of Mechanics, Faculty of Mechanical Engineering, VŠB-Technical University of Ostrava, 17. listopadu 15, Ostrava, tel. (+420) 59 732 5283, e-mail jan.szweda@vsb.cz

dimensional and time-dependent, they are described by the system of partial differential equations which has to be solved by numerical methods.

2 MATHEMATICAL MODEL EQUATIONS

2.1 Fluid

Mathematical model of the general three-dimensional non-stationary fluid flow with heat transfer is described by system of equations which is consist of equation of continuity, momentum transfer equation and heat convection equation. It is necessary to use the statistic turbulence model based on time averaging of the turbulent flow values and on subsequent time averaging procedure of balance equations to solve the given turbulent flow problem.

Let us consider the equations for time averaged values and equation of continuity in the shape

$$\frac{\partial \rho}{\partial t}+\frac{\partial\left(\rho \bar{u}_j\right)}{\partial x_j}=0 \ , \tag{1}$$

where:

ρ – density $[\mathrm{kg} \cdot \mathrm{m}^{-3}]$,

t – time $[\mathrm{s}]$,

$\bar{u}_j$ – averaged velocity in generalized direction $[\mathrm{m} \cdot \mathrm{s}^{-1}]$,

x_j – generalized direction $[\mathrm{m}]$.

The compressed shape of equation (1) can be expanded according to the Einstein’s summing rule for relevant problem dimension, i.e. for 3D problem can be written

$$\frac{\partial \rho}{\partial t}+\frac{\partial\left(\rho \bar{u}_x\right)}{\partial x}+\frac{\partial\left(\rho \bar{u}_y\right)}{\partial y}+\frac{\partial\left(\rho \bar{u}_z\right)}{\partial z}=0 . \tag{2}$$

Furthermore for numerical CFD analysis momentum transfer equation can be written in the form

$$\frac{\partial\left(\rho \overline{u_i}\right)}{\partial t}+\frac{\partial\left(\rho \overline{u_i u_j}\right)}{\partial x_j}=-\frac{\partial \bar{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\left(\mu+\mu_t\right) \frac{\partial \overline{u_i}}{\partial x_j}\right)+\underbrace{\rho \delta_{i3} g}_{buoynancy\ \ forces}+\underbrace{\rho f_c \varepsilon_{ij3} \overline{u_j}}_{Coriolis\ 's\ forces}+\rho f_i \ , \tag{3}$$

where:

p – averaged pressure value $[\mathrm{Pa}]$,

μ – dynamic viscosity $[\mathrm{Pa} \cdot \mathrm{s}]$,

μ_t – turbulent viscosity $[\mathrm{Pa} \cdot \mathrm{s}]$,

δ_{i3} – Kronecker’s δ $[1]$,

g – gravitational acceleration $[\mathrm{m} \cdot \mathrm{s}^{-2}]$,

f_c – Coriolis’s parameter $[\mathrm{s}^{-1}]$,

ε_{ij3} – Einstein’s summing tensor $[1]$,

f_i – generalized volume force $[\mathrm{N} \cdot \mathrm{kg}^{-1}]$.

For numerical analysis it is neccesary to addjust the momentum transfer equation by turbulence model equations – a set of additional equations and empiric formulas which build together with equations (1) and (3) the solvable equation system.

Let us consider the k-e turbulence model defined equation for disipation rate in the form of

$$\frac{\partial \varepsilon}{\partial t}+\frac{\partial \bar{u}_j \varepsilon}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\frac{\nu_t}{\sigma_{\varepsilon}} . \frac{\partial \varepsilon}{\partial x_j}\right)+c_{1 \varepsilon} \nu_t\left(\frac{\partial \bar{u}_j}{\partial x_l}+\frac{\partial \bar{u}_l}{\partial x_j}\right) \frac{\partial \bar{u}_l}{\partial x_j}-c_{2 \varepsilon} \frac{\varepsilon^2}{k} \ , \tag{4}$$

where $c_{1\varepsilon}$, $c_{2\varepsilon}$ and σ_ε are the empiric coefficients (all [1]) and:

ε – disipation rate $[\mathrm{m^2 \cdot s^{-3}}]$,

ν_t – turbulent viscosity $[\mathrm{m^2 \cdot s^{-1}}]$,

j, l – Einstein’s summing indexes [1] ,

k – turbulent kinetic energy $[\mathrm{m^2 \cdot s^{-2}}]$ and is defined as

$$k = \frac{1}{2}\left(\overline{u_1'^2} + \overline{u_2'^2} + \overline{u_3'}\right) = \frac{1}{2}\overline{u_j'^2} \quad , \tag{5}$$

where:

$\overline{u'^2_j}$ – averaged value of squared fluctuational velocity component in generalized direction $[\mathrm{m^2 \cdot s^{-2}}]$.

The heat transfer in fluids is described by the equation

$$\frac{\partial}{\partial t}\left(\rho \overline{h}\right) + \frac{\partial}{\partial x_j}\left(\rho \overline{u_j} \overline{h}\right) = \frac{d\overline{p}}{dt} + \frac{\partial}{\partial x_j}\left(\left(\lambda + \lambda_t\right)\frac{\partial \overline{T}}{\partial x_j}\right) + \frac{\partial}{\partial x_l}\left(\overline{\tau_{jl}}\,\overline{u_j}\right) \quad , \tag{6}$$

where:

$\overline{h}$ – averaged value of enthalpy $[\mathrm{J \cdot kg^{-1}}]$,

λ – molecular temperature conductivity coefficient $[\mathrm{W \cdot m^{-1} \cdot K^{-1}}]$,

λ_t – turbulent temperature conductivity coefficient $[\mathrm{W \cdot m^{-1} \cdot K^{-1}}]$,

T – temperature $[\mathrm{K}]$,

τ_{ij} – viscous stress tensor $[\mathrm{Pa}]$.

2.2 Solid

The object of the work whose temperature field is the objective of this paper is a solid body. Let us consider the heat convection equation for description of temperature field in the form

$$\frac{\partial}{\partial t}\left(\rho c_p T\right) = \frac{\partial}{\partial x_j}\left(\lambda \frac{\partial T}{\partial x_j}\right) \tag{7}$$

where:

c_p – specific heat capacity $[\mathrm{J \cdot kg^{-1} \cdot K^{-1}}]$.

The equation systems for solid and fluid phase are mutually linked by boundary conditions at the boundary shared by both phases, i.e. at the surface of the body flowed by liquid. After completing of equation system by appropriate outer boundary conditions (for the flow problem and for problem of heat convection and transfer) it is possible to approach the numerical simulation of boundary or mixed problem.

3 SOLVED TECHNICAL APPLICATION

The aim of the work is to solve the surface temperatures of electromotor frame. Because of the complicated shape and difficult determination of the flow speeds (its knowledge is necessary for solution of heat transfer problem), especially in the space among the ribs, it was suitable to analyse the flow by numerical simulation. In dependence of used numerical model can be, except required surface temperatures on the frame, obtained another results such as flow speeds or heat transfer coefficients, etc.

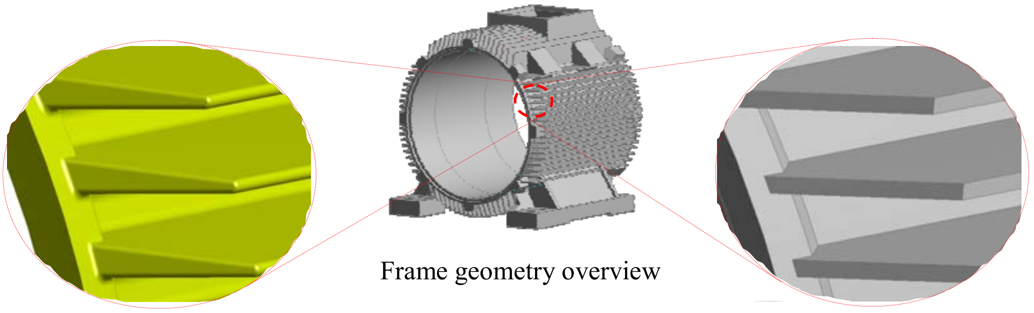
3.1 Real shape versus numerical model geometry

The numerical model is always certain approximation of the reality because it substitutes the real behaviour by chosen mathematical model. The rate of reality approximation is given especially by the difficulty to determine the parameters of mathematical model and by accessibility of HW and SW equipment.

The preparation and creation of computational models for numerical simulations is usually accompanied by the solution of compromise between model details and computational requirements. Exceedingly actual is this question in solution of flow problems where it is usually required to model relatively large space which has to be meshed very fine in the locations important for solution results. Those spaces are in case of flow modelling especially the places with eddies, in case of heat transfer solution it is the interference layers where the value y_+ is defined for judgement of mesh quality.

In case of solved electromotor with axial height 315mm it was found out by simple experiment on stand that stream field of the cooling air reaches the distance approximately 200mm from the electromotor surface. For the length of the motor 700mm is the volume of modelled fluid domain approximately 0.365m^3. Assuming all geometrical details and all recommendations for mesh finesse it was necessary to mesh ½ of mentioned volume (the task was solved in plane symmetry) by approximately 15 millions cells (finite volume elements). To create the mesh with this number of elements takes approximately 8 hours and the manipulation with such model was only hardly possible. Based on this fact certain geometrical simplifications were performed which reduce the number of needed elements. By observation of created mesh topology it was found out that the large number of elements is located close to the fillet of all sharp junctions and rib edges. After fillet removing the geometrically simplified model is created which needs for its meshing only approximately 1.5 millions elements.

Meshing, solution and subsequent analysis have to be realized on 64bit platform (32bit platform of Windows does not offer sufficient HW/ SW capacity). The configuration of working station was following: Intel Core2 Quad, 8GB RAM, OS Windows XP x64, application software Ansys Workbench v11 CFX module, parallel solver for 4 CPUs.



Real shape detail

Frame geometry overview

Model geometry detail

Fig. 1 Geometry difference between real shape and numerical model

3.2 Numerical model properties

The created model consists of two parts – solid domain a fluid domain. The electromotor frame is made from cast iron, the constant temperature conductivity coefficient ($50\,\mathrm{W\cdot m^{-1}\cdot K^{-1}}$) was assumed. The frame cooling was realized by air flow with following material parameters: density ($1.185\,\mathrm{kg\cdot m^{-3}}$), dynamic viscosity ($1.831\cdot10^{-5}\,\mathrm{Pa\cdot s}$), heat conductivity ($2.61\cdot10^{-2}\,\mathrm{W\cdot m^{-1}\cdot K^{-1}}$). The problem was solved as quasi-static, the k-epsilon turbulence model was considered.

The boundary conditions for solid domain were applied in the form of heat flux located on inner cylindrical part of the frame. For the fluid domain it is necessary to define following boundary conditions: *inlet* on the incoming surface (mass flow applied), *outlet* on the outgoing surface, *wall* or *opening* on other surfaces.

The created numerical model consists of 1681366 elements in fluid domain and 39719 elements in solid domain. Overall time needed for the solution on above mentioned workstation configuration was 5 hours and 13 minutes.

3.3 Adjustment of numerical model and calculated results

The heat flux applied on the inner part of the electromotor frame was for first approximation applied according to the heat losses mentioned in product sheet. The losses in iron-, copper- and aluminium motor section are considered. It is difficult to determine exactly the value of heat flux for appropriate boundary condition because this value includes the contributions of all distinguished loss components. The results of primary frame cooling simulation were used especially to obtain the rough overview and for preparation of measurement by thermovision camera and by thermoelements, i.e. for determination of suitable locations for thermoelements, etc.

In the next step the frame surface temperature measurement was performed with the aim to calibrate the model and to verify the influence of simplified ribs modelling. Within the measurement the motor was scanned by thermovision camera from number of different angles and temperatures were measured by thermoelements in chosen locations. For the calibration purposes the values of electric input, moment of load and revolutions were recorded as well. On the base of those values it was possible to tune the heat flux values so that the agreement between measured and calculated results was sufficient.

With respect to the aim of the work and its further use the most important result is temperature distribution on the electromotor frame surface. The good agreement between temperatures obtained by measurement and by calculation shows the possibility to accept the performed simplifications (ribs geometry, way of boundary conditions application) and to consider the calculation model as sufficient, i.e. proper for further numerical simulations. This good results agreement results from comparison at Fig. 2 and 3.

Except the calculation of temperature field on the electromotor frame another, from the motor cooling point of view, important values can be evaluated, i.e. heat transfer coefficients distribution, speed of cooling flow in defined plane or streamline visualization for determining of the air flow efficiency.

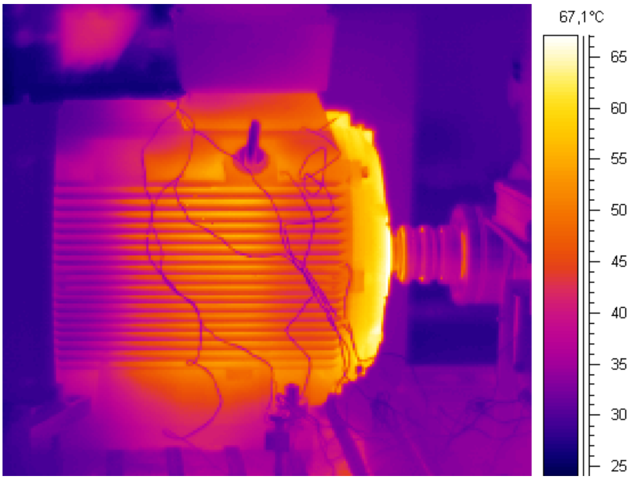


Fig. 2 Temerature distribution measured by thermovision camera

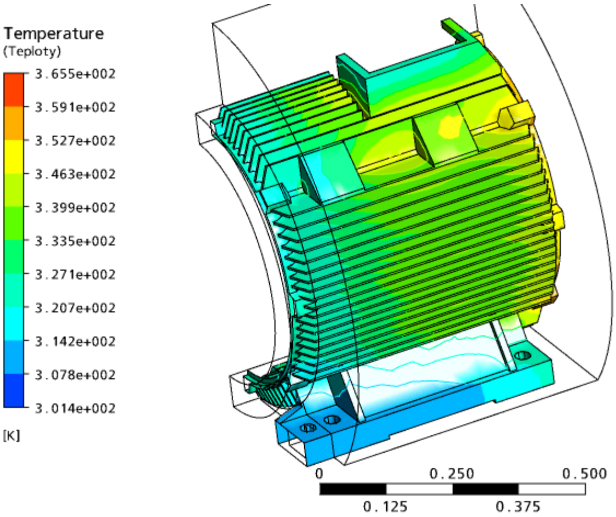


Fig. 3 Temerature distribution calculated by CFD analysis

4 CONCLUSION

This paper presents the formulas used for CFD analysis of the flow around the solid structure. The processes for creation of practically usable computer model and boundary condition application are described and obtained results are presented. The calibration of used numerical model is performed on the base of experimental data in the form of motor surface temperatures.

It can be stated that:

1. The CFD analysis of larger mechanical parts is only hardly realisable without certain simplification: CFD model of real geometry contains approximately 15 millions of cells, the model of simplified geometry approximately 1.5 million ones. Also in this case is the model rather demanding on the HW equipment and solution times are in order of hours.
2. Assuming of simplified geometry modeling it is necessary to provide the calibration of numerical model. This calibration was performed on the base of values measured by thermovision camera and thermoelements. Used procedure is applicable only in cases when experimental data are available.
3. The well created and calibrated model can be used for determination of values which can be only hardly obtained via experiment. In such way obtained results are in technical problems highly appreciated and are contributions for optimization of current devices or for development of new ones with similar geometry.

ACKNOWLEDGEMENT

The work has been supported by the research project MSM6198910027 of the Ministry of Education, Youth and Sports of the Czech Republic what is highly appreciated.

REFERENCES

[1] RODI, W. & FUEYO, N. *Engineering Turbulence Modelling and Experiments 5*. 1st ed. Oxford : Esevier, 2002. 1010 pp. ISBN 0-08-044114-9.

[2] LESIEUR, M., ZAGLOM, A. & DAVID, F. *New trends in turbulence*. Berlin : Springer-Verlag, 2001. 554 pp. ISBN 3-540-42978-6.

[3] MIKOLAJEK, J., KOVÁŘ, L., KOZUBKOVÁ, M. & BOJKO, M. Numerické řešení disociace uhlovodíkových plynů při průtoku dmyšnými systémy s uvážením přívodu tepla k proudícímu médiu. *Sborník vědeckých prací VŠB-Technické univerzity Ostrava. Řada strojní*. 2001, XLVII. Nr. 1/part 2, pp. 1-16. ISSN 1210-0471.

[4] BOJKO, M., KOZUBKOVÁ, M. & KOVÁŘ, L. Chlazení ústí trysky plynnými palivy u konvertoru Oxyvit. In *Sborník „9. uživatelská konference Fluent 2003“*. Valašské Klobouky: Techsoft Engineering, 2003, pp.(2-103)-(2-112). ISBN 80-239-1647-5.

Reviewers:

prof. Ing. Antonín Skoc, DrSc, Silesian University of Technology

prof. Ing. Dr. Milan Sága, University of Žilina