ZÁPADOČESKÁ UNIVERZITA V PLZNI

FAKULTA STROJNÍ

KATEDRA ENERGETICKÝCH STROJŮ A ZAŘÍZENÍ

ZÁPADOČESKÁ UNIVERZITA V PLZNI

POWER SYSTEM ENGINEERING, THERMODYNAMICS & FLUID FLOW

2012

JEDNOTLIVÝ PŘÍSPĚVEK ZE SBORNÍKU

evropský sociální fond v ČR

EVROPSKÁ UNIE

MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY

OP Vzdělávání pro konkurenceschopnost

INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

SIMULATION OF FLOW WIHT HEAT TRANSFER INTO BLADE

BĚTÁK Vojtěch, FÜRST Jiří

This paper deals with simulation of flow with heat transfer into the blade using open source simulation tools especially OpenFOAM. The effect of viscous heating is discussed and its implementation to the structure of temperature field is desribed.

Klíčová slova: OpenFOAM, heat transfer, blade, viscous heating

1. Introduction

There are a lot of open source simulation tools. We can mention for example Code Saturne, Elmer, COOLFluiD or OpenFOAM which is quite popular in recent time. This code is based on finite volume method where the variables are stored in the cell center and the equations are solved sequentially using PISO or SIMPLE algorithms. The package contains a lot of tools for incompressible, compressible flow, flow with chemical reaction etc. Our previous work [1] shows that it is possible to solve combustion together with heat transfer with the wall of combustion chamber. Therefore we decided to compare the results of the OpenFOAM and an in-house code for the case of flows through a turbine cascade with heat transfer between the fluid and the solid blade.

2. Governing equations

The steady-state solution of the Favre averaged Navier-Stokes equations (1)-(3) for the fluid field and the Laplace equation (4) for the heat conduction in the blade is sought.

$$\frac{\partial}{\partial x_j}(\rho u_j) = 0 \tag{1}$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right)\right] \tag{2}$$

$$\frac{\partial}{\partial x_j}(h u_j) = \frac{\partial}{\partial x_j}\left[\left(\alpha + \frac{\mu_t}{Pr_t}\right)\frac{\partial h}{\partial x_j}\right] + \frac{\partial}{\partial x_j}\left[u_i(\tau_{ij})_{eff}\right] \tag{3}$$

$$\lambda\frac{\partial^2 T}{\partial x_j^2} = 0 \tag{4}$$

here $\rho$is the density, $u_i$is the component of fluid vector, $p$is the effective pressure, $\mu$is the dynamic viscosity, $h$is the enthalpy,$\alpha$is the coefficient of heat diffusion,$\mu_t$is th turbulent viscosity and $(\tau_{ij})_{eff}$is the effective stress tensor. $T$is the temperature and $\lambda$is the heat flux coefficient in the solid region.
The term (5) is often called as viscous heating term. This term is neglected in number of pressure-based solvers. If the Brinkman number (6) is greater than 1 (compressible flow),

$$\dot{q}_{vis} = \frac{\partial}{\partial x_j}\left[u_i(\tau_{ij})_{eff}\right] \tag{5}$$

we can not neglected this term. The Brinkman number is given by (6) where $\kappa$is the thermal conductivity of fluid, $T_w$is the temperature of wall and $T_0$is the bulk temperature of the fluid.

$$Br = \frac{\mu \parallel u_i \parallel}{\kappa(T_w - T_0)} \tag{6}$$

For the simulations we assumed the ideal gas with parameters corresponding to hot steam an the$k - \omega SST$model given by following equations

$$\frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\Gamma_k \frac{\partial k}{\partial x_j}\right] + G_k - Y_k \tag{7}$$

$$\frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j}\left[\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right] + G_\omega - Y_\omega + D_\omega \tag{8}$$

$$\mu_t = \frac{\rho k}{\omega}\left[max\left(\frac{1}{\alpha^*}, \frac{SF_2}{a_1 \omega}\right)\right]^{-1} \tag{9}$$

$$(\tau_{ij})_{eff} = (\mu + \mu_t)\left[\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right] \tag{10}$$

where $k$is a kinetic energy of turbulence and $\omega$is a specific speed of dissipation. Model functions and constant are defined in [2].

3. Boundary condition

The specific boundary condition on the wall for the turbulence model and heat transfer have to be chosen. We have used Low-Re modification of the turbulence model where the boundary values are defined in (11)

$$k_w = 10^{-10}, \omega_w = \frac{6\mu}{\beta_0 y_1^2} \tag{11}$$

The boundary condition for heat transfer must satisfy the equality of temperature and heat flux from the both sides (fluid, solid) of wall. This boundary condition is defined in (12).

$$T_w = \varphi T \mid_{\Omega_2} + (1 - \varphi) T \mid_{\Omega_1} \tag{12}$$

The functions $\varphi$and $\delta\lambda$are defined as

$$\varphi = \frac{(\delta\lambda) \mid_{\Omega_2}}{(\delta\lambda) \mid_{\Omega_2} + (\delta\lambda) \mid_{\Omega_1}} \tag{13}$$

$$(\delta\lambda) \mid_{\Omega_i} = \frac{\lambda}{(\vec{n}, \vec{c_f} - \vec{c_c})} \mid_{\Omega_i} \tag{14}$$

where $\vec{c_f}$are coordinates of wall face center and $\vec{c_c}$are coordinates of wall layer cell center.

4. Implementation of viscous heating term

There are two ways how to implement viscous heating term. The first one is based on definition of effective stresses in turbulence library and it is shown in following piece of code.

```
volSymmTensorField TauEff = -(turbulence->devRhoReff());
volScalarField viscousHeat = fvc::div( TauEff.T() & U );
```

Code 1: Viscous heating term using turbulence library

The second one is based on approach that is used in ***rhoCentralFoam*** solver. These two approaches will be compared.

```
const surfaceScalarField sigmaDotU
   (
      fvc::interpolate(turbulence->muEff(), "interpolate(muEff)")
      * (  mesh.magSf()*fvc::snGrad(U)
          +
           ( mesh.Sf() &
             fvc::interpolate(dev2(Foam::T(fvc::grad(U))), "interpolate(gradU)")
           )
        )
           & fvc::interpolate(U,"interpolate(Uv)")
      );
volScalarField viscousHeat = fvc::div(sigmaDotU)- 2./3.*fvc::div(phi,turbulence->k());
```

Code 2: Viscous heating term using rhoCentralFoam approach

5. Results

Figure 1 shows the results for basic steady-state solver and its extensions for the solver with heat transfer. If we compare this data with results obtained by in-house code from Fürst and Louda [3] (Fig. 2.), we can see some differences. The first one is in value of maximal and minimal temperature and the second one is in the position of temperature profile for the case of flow with heat transfer. This profile should be inside the profile without heat transfer as is shown in figure 2.

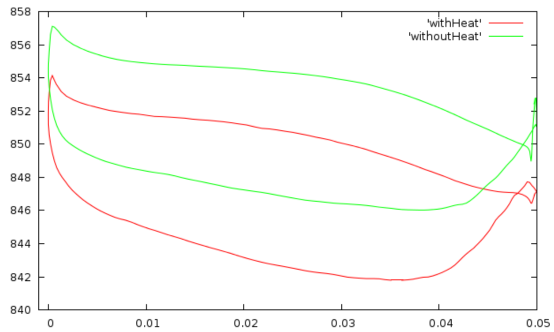

Fig. 1.: Temperature profiles for OpenFOAM solver without heat transfer

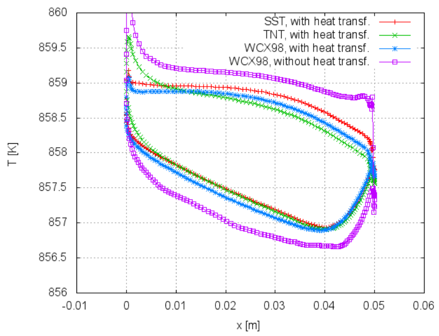

Fig. 2: Temperature profiles for solvers developed by J.Fürst and P. Louda

Figures 3 and 4 show the results for the OpenFOAM solvers with viscous heating and without heat transfer. Results with adiabatic wall are similar to the solution of J. Fürst but profiles are not perfectly smooth as expected. Results for the solvers with heat transfer indicate that the viscous term implementation based on turbulence library is not correct.

6. Conclusion

This paper showw the effect of viscous heating term to the temperature profile on surface of a turbine blade. The assumption of small Brinkman number, that have been adopted by many OpenFOAM solvers, is not correct in our case. Two methods have been proposed how to correct this defficiency and the method based on rhoCentralFoam approach have been chosen as better.

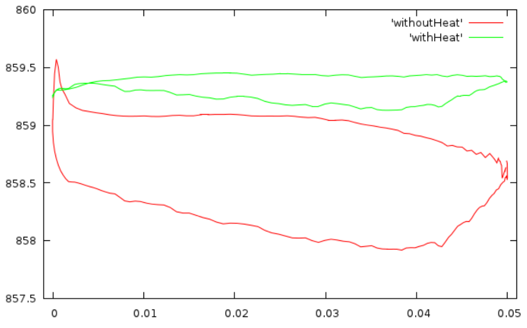

Fig. 3.: Temperature profile for OpenFOAM solver using viscous heating code 1

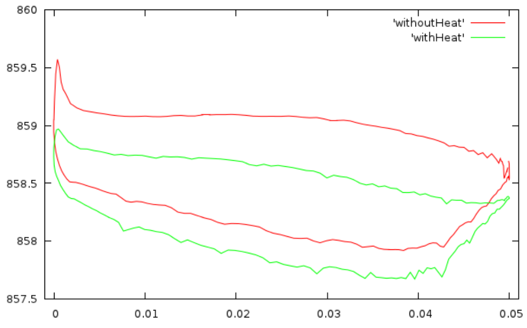

Fig. 4.: Temperature profile for OpenFOAM solver using viscous heating code 2

The oscillations in the temperature profiles are probably caused by the OpenFOAM spatial discretization schemes. It seems that the schemes have some accuracy problems for high aspect ratio cells which were dictated by Low-Re approach. Our calculations with the same mesh using coupled in-house codes (see Fig. 2) and with other segregated solver (Fluent) show that the mesh was fine enough to get smooth temperature profiles.

The future work will be oriented to develop of fully implicit schemes based on AUSM (Advanced Upwind Spliting Methods) schemes and application of advanced RANS turulence model.

References

[1] BĚTÁK, V.; KUBATA, J., TŮMA, J.: Numerical Study of Reacting Flow with Heat Transfer through the Wall into Neighborhood, Topical Problems of Fluid Mechanics 2012, Czech Academy of Science, Department of Thermodynamics, February 2012

[2] PŘÍHODA, J.; LOUDA, P.: Matematické modelování turbulentního proudění,CTU, Prague, 2007

[3] FOŘT, J., FÜRST, J., et al.: FVM-FEM Coupling and its Application to Turbomachinery, Finite Volumes for Complex Applications VI,Prague, 2011, ISSN 2190-5614

Acknowledgment

This work was supported by the Grant Agency of the Czech Technical University in Prague, grant No. SGS 10/243/0HK2/3T/12.

Ing. BĚTÁK Vojtěch, Department of technical mathematics, Faculty of mechanical engineering, CTU Prague, Karlovo nam. 13, 121 35 Praha 2,Czech Republic, betakvojtech@gmail.com
Doc. Ing. FÜRST Jiří PhD.,Department of technical mathematics, Faculty of mechanical engineering, CTU Prague, Karlovo nam. 13, 121 35 Praha 2,Czech Republic, Jiri.Furst@fs.cvut.cz