ACTA TEHNICA CORVINIENSIS – Bulletin of Engineering
Tome VII [2014] Fascicule 1 [January – March]
ISSN: 2067 – 3809

[1.] Pavel ZÁCHA

STEAM ENGINE – CFD SIMULATIONS WITH MOVING MESH

CTU in Prague, Faculty of Mechanical Engineering,
Department of Energy Engineering,
Technická 4, Prague, CZECH REPUBLIC

Abstract: The first steam engine was designed almost 250 years ago. Nevertheless, even today there are its applications linked to energy utilization which are of interest. Especially in some technological processes were throttling of steam generates losses, which generate no useful energy. This paper briefly analyses problems arising from the CFD simulation of steam flow in a piston machine. Based on a prototype facility a simplified 2-dimensional model, with computational grid of the single-acting steam engine in pre-processor GAMBIT, was set up. In the first part of this paper the model generation is described and difficulties linked with moving and dynamic meshing are discussed. In the second part, the boundary conditions for the simulations are defined and results of simulations performed in the CFD code ANSYS-FLUENT are presented. The main outputs of the performed simulations are indicator diagrams that can subsequently be used for determination of the time-dependent forces acting to the piston.

Keywords: steam flow, CFD simulation, prototype, model generation, simulations

INTRODUCTION

In the last year PolyComp Ltd. Company was developing a piston steam engine with plan to transform even a small part of steam thermal energy which is lost through the throttling loss. Piston machines make it possible to transform some thermal energy to the energy of piston motion and next to generate electrical energy.

Imagine a case of using two levels of steam parameters (namely temperature and/or pressure) for different parts of one technological process. You have two general possibilities to solve it:

a) to separate the process into two parts leading to generating final levels of steam parameters;

b) to produce all the steam at higher level of these parameters and subsequently reduce demanded part of the steam to the lower level.

If the second case is better than the first one in his economic point of view, we need use non-reversible process – throttling loss. The first choice involves higher capital costs, the second one causes growing of the operating costs. However, in many cases these processes are already in operation, so that the possibility b) was chosen before and economies are founding afterwards. As we know, in throttling process all the steam superfluous energy is lost. Therefore we look into some machine designs with sufficient efficiency leading to the early economic return. For many cases with sufficient power, we can find solution in small steam turbines. But, if is there too small power to use, the investment to turbine machine is uneconomic. For small electric output (approximately up to 300 kW) it is better to use modified series manufactured piston engines (in Czech conditions e.q. Tedom engines). One of the main visible modifications is cams replacement of sleeve-valves. This innovators solution has some difficulties though, e.g. sleeve-valves sealing, heat stress during starting of the unit or strength limits of the crank mechanisms.

MODEL DESCRIPTION

Figure 1 shows a 2-D model consisting of input chamber with sleeve-valve, entry channel, steam cylinder with moving piston, output channel and output chamber with sleeve-valve. In reality the input and output chamber includes always 6 *parallel sleeve-valves which connect 6 parallel cylinders. Sleeve-valves have three pairs coupled to each other with 120° angle relative slewing. This linkage ensured more fluent steam distribution to the piston and speed-torque force to the crankshaft. The steam engine works as a two-stroker. When the inlet sleeve-valve is open, higher pressure steam expands to the cylinder and push to the downward moving piston. In this moving section useful work is done. When outlet sleeve-valve is open, the upward moved piston pushes steam away to the outlet chamber. This cycle works in nominal rotation speed 450 rpm [1].*

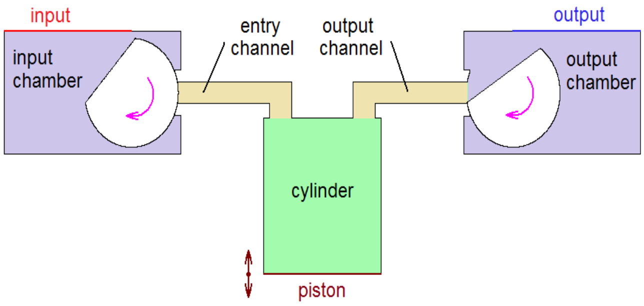

Figure 1. Boundary conditions setting and computational areas labeling of simplified 2D model geometry

MODEL CONDITIONS SETUP

To obtain sufficiently accurate steam flow description the ANSYS-FLUENT CFD code needs to solve a number of specific geometrical and computational parameters [3]:

- *Sliding and Dynamics Mesh Models – control area contains two different moving parts: piston and rotational sleeve valves.*
- *Two-face flow – intensive expansion of the superheated steam can lead to subcooling of steam under saturation temperature. We need to add in the solution suitable two-dimensional model which covers the condensation process.*
- *Transient – in merits of the case, this model causes high-unsteady behaviour of the flow. As a result it has very small time step.*

Sliding Mesh Model

Sleeve-valves generate rotating circle with cut-off segment (see Fig. 2). For this movement it can be suitable to use the Sliding Mesh Model which accounts the relative motion of stationary and rotating components. In the Sliding Mesh Model two main cell zones are used. Each cell zone is bounded by so-called Interface zone where it meets the opposing cell zone. The Interface zones of adjacent cell zones are associated with one another to form a Grid interface. The two cell zones will move relatively to each other along this Grid interface, which must be positioned so that it has fluid cells on both sides. On the right side of Figure 2 the detailed area around Grid interface (red line) is shown. For a better description of fluid behaviour there are more cells rows on both sides.

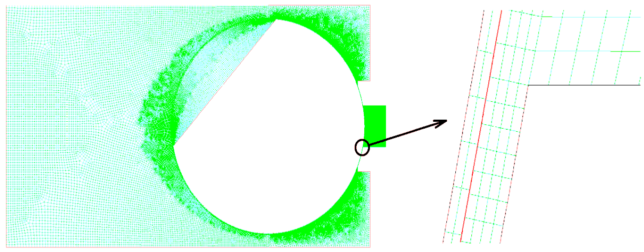

Figure 2. Computational mesh in the input chamber region with auxiliary view of slot area

Dynamic Mesh Model

The Dynamic Mesh Model uses the ANSYS-FLUENT solver to move boundaries and/or objects, and to adjust the mesh accordingly. Suitable cases include the piston moving inside an engine cylinder. We can use dynamic layering to add or remove layers of cells adjacent to a moving boundary, based on the height of the layer adjacent to the moving surface. The Dynamic Mesh Model allows specifying an ideal layer height on each moving boundary. The layer of cells adjacent to the moving boundary (layer j in Figure 3) is split or merged with the layer of cells next to it (layer i) based on the height h of the cells in layer j. ANSYS-FLUENT code also enables to set the in-cylinder options (crank shaft speed, starting crank angle, crank radius, etc.) which are used to convert between flow time and crank angle.

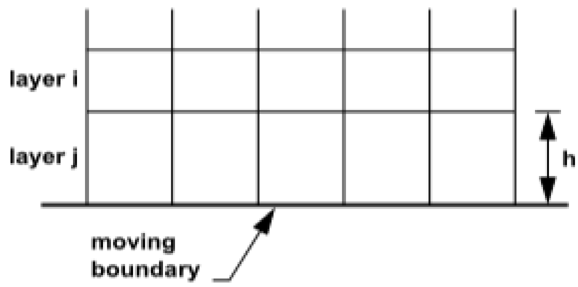

Figure 3. Dynamic Layering

Transient

Sliding and Dynamic Mesh Models are the most accurate methods for simulating flows in multiple moving reference frames, but also they are the most computationally demanding.

For CFD simulations it is necessary to find a convergent solution. In order to find it, we need to set useful especially physical convergence criteria. For this case the temperature and pressure course criteria was set. Criteria conditions are satisfied when the controlled variable had coincident course with minor deviations in two consecutive (crankshaft) revolutions. In reality, pressure relative deviations do not exceed limit of 2% and average relative deviation under 0.3%. The temperature relative deviations were a little higher because of steep pressure rising during the time, when sleeve-valves were just opening. Nevertheless, the maximum temperature relative deviation does not exceeds 5%. One example of a successful convergence of temperature course is on Figure 4 [5].

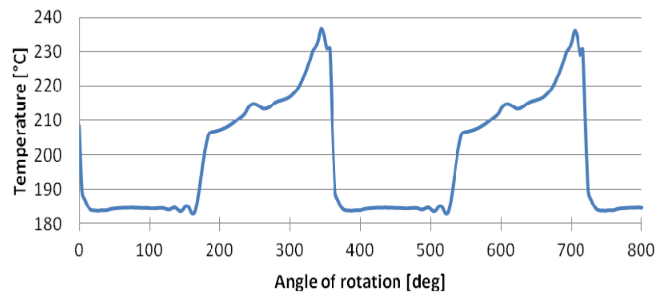

Figure 4. Example of cylinder average temperature course (load time 5°)

To reach convergent solution we need 5 or 6 revolutions with 0.1° slewing time step (i.e. 3.704×10^{-5} s for 450 rpm). With 20 iterations per one time step it leads approximately to 400,000 iterations. Using the 16 processors of Intel Nehalem server we totally need just between 80 and 100 hours of computational (real) time.

Condensation

For solved technological processes, the pressure and temperature conditions of admission steam are near to the saturation state. During a rapid expansion of steam, a condensation process will take place shortly after the state path crosses the vapour-saturation line. The expansion process causes the super-heated dry steam to first subcool and then nucleates to form a two-phase mixture of saturated vapour and fine liquid droplets. For these cases ANSYS-FLUENT code implements Wet Steam Model, which uses the Eulerian-Eulerian approach. The flow mixture is modelled using the compressible Navier-Stokes equations, in addition to two transport equations for the liquid-phase mass-fraction (□), and the number of liquid-droplets per unit volume (□). The phase change model, which involves the formation of liquid-droplets in a homogeneous nonequilibrium condensation process, is based on the classical nonisothermal nucleation theory. Physical restrictions of the Wet Steam Model are the following: the velocity slip between the droplets and gaseous-phase is negligible; the interactions between droplets are neglected; the mass fraction of the condense phase is small (□ < 0.2) and droplet sizes are typically very small (between approximately 0.1 and 100 mm) [3].

Above-mentioned restrictions are in good accords with solved case conditions (□ exceed over 0.2 values only locally) therefore the Wet Steam Model can be used.

SIMULATIONS SETUP

Basic operating parameters setup of the simulations includes: inlet operating pressure (0.1 MPa); inlet gauge pressure (2.0 MPa); inlet temperature (213 °C); outlet gauge pressure (1.0 MPa); sleeve-valves and crank shaft rotation speed (450 rpm); crank radius (75 mm) and connecting rod length (250 mm). For transients problem setup time step was 3.704×10^{-5} s which corresponds to the angular rotation of sleeve-valves and crank shaft by 0.1°. According to the model variants we set up lead times (5°, 7° respectively). The lead time presents the angle of rotation by which the open input/output sleeve-valves preceded the piston top/bottom dead centre.

Basic simulation parameters are: numeric schema – Density-Based Solver; viscous model – turbulent Standard k-□ model with Standard wall function; discretization schemas - Upwind second order [5].

RESULTS

Simulations results for the PolyComp Company include:

- *Sliding and Dynamics Mesh Models – control area contains two different moving parts: piston*
- *Quantity of steam leaving from input chamber in a closed state - steam outflow through slots was analysed.*
- *Indicator diagrams – applies for cycle efficiency determination in different settings (load times*

or pressure levels) and for determining of the forces acting on the piston.

- *Temperature fields on the walls – enables to better temperature stress definition in construction materials (stress analysis).*
- *Mass flow quantity of the aqueous phase – steam moisture quantity checking towards limit values of the used model (□ < 0.2).*
- *Videos of the tracked variables –visualization providing better process understanding.*

The most important results are indicator diagrams, whose values for subsequent strength calculations have been used. Examples of indicator diagrams for two different load times (5° and 7°), inlet pressure p_{in} = 2MPa and outlet pressure p_{out} = 1MPa are shown in Figure 5. Diagrams show the average values for the static pressure in the cylinder, depending on the angle of rotation. Zero angle was set up to the top dead centre.

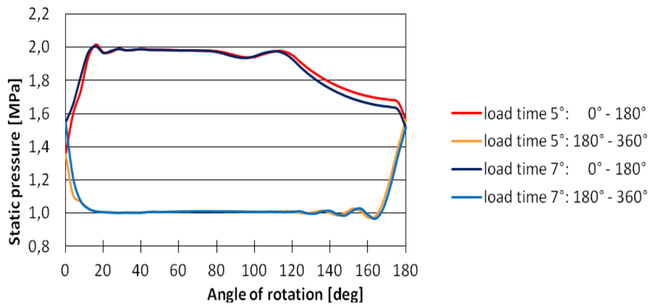

Figure 5. Indicator diagrams for p_{in} = 2 MPa and p_{out} = 1 MPa; load times 5° and 7°

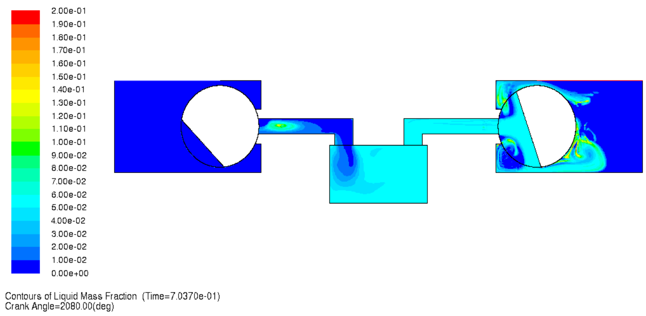

Figure 6. Weight ratio process of aqueous phase in the model for selected rotation angle

When we compare the two shown diagrams, we can notice minor effect of the load times setting onto performed work, or on the forces acting on the piston as it is by the piston internal combustion engine. A more important point of view to the load times setting is creation and diffusion of supercooled steam, which influences the formation of condensation. The process of aqueous phase diffusion for the selected rotation angle is shown in Figure 6. In general, it can be concluded that a higher aqueous phase fraction is formed but in the output chamber.

CONCLUSIONS

2D steam flow simulation on the CFD model of the piston engine was carried out. Sliding and Dynamics Mesh Models and also Wet Steam Model including condensation process were used. As a main result chosen indicator diagrams were described. 2D symmetry cannot substitute a full 3D simulation of the entire machine. The results are used especially as an alternative method of monitored parameters course determination (temperatures, pressures and condensation). The chosen solution method also indicated present possibilities of the CFD simulations in this area.

ACKNOWLEDGEMENT

Financial support for this work was under a project number TIP FR-TI1/379 "Development and production of a prototype of a steam cogeneration unit for fluctuating steam flow with output 100 to 300 kWel" is gratefully acknowledged.

REFERENCE

[1] *Drawing documentation of the LIAZ M1 engine.*
[2] *GAMBIT 2.4 User's Guide, Fluent Inc., May 2007.*
[3] *FLUENT 6.3 User's Guide, Volume 1, 2, 3 and 4, Fluent Inc., September 2006.*
[4] *ZÁCHA, P., GREGOR, K., ŽELEZNÝ, V.: Simulace proudění páry v pístovém stroji, Research report no. 12107-JE/2009/01, ČVUT v Praze, 2009.*
[5] *ZÁCHA, P., GREGOR, K.: Simulace proudění páry v pístovém stroji – 2. část, Research report no. 12107-JE/2010/01, ČVUT v Praze, 2010.*

ACTA Technica CORVINIENSIS
BULLETIN OF ENGINEERING
ISSN:2067-3809