17th International Conference
ENGINEERING MECHANICS 2011
Svratka, Czech Republic, 9-12 May 2011

CALCULATION STUDY OF STEAM FLOW THROUGH A CONTROL VALVE

L. Bednář[*], L. Tajč[*], L. Jirka[*]

Abstract: *Outcomes of a numerical simulation of flow through an unbalanced control valve during the turbine's operational modes. A 2D as well as a 3D calculation study is considered. Effect of a protective screen on aerodynamic conditions inside the valve is modeled. Distribution of velocity and pressures in the diffuser is determined. Pressures affecting the valve cone are described.*

Keywords: ***Steam turbine, control valve.***

1. Introduction

Control valves adjust the required output of the turbine by means of the mass flow and admission steam pressure before the turbine. Unbalanced valves have been adequately tested in practical applications. In the course of time, a flat-bottomed valve cone has become established. A valve of this type, however, has never been used for turbines because of the supercritical steam parameters. The calculation study is to evaluate aerodynamic characteristics affecting the valve cone and the output pipe under extreme temperatures and pressures. An effect of the screen and the entry angle of flow rotation on distribution of the velocity and pressures in the space under the valve cone can also be considered. This presentation summarizes outcomes of a numerical simulation of steam flow through the valve (Jirka, 2007).

2. Calculation model of the control valve

A cross-section of the control valve is illustrated in Fig. 1. The valve is equipped with a protective screen of a cylindrical shape made of metal sheeting, perforated with approximately one thousand small round apertures. In its bottom section, channels are milled under 45°. These apertures spin a part of the admission steam. It is supposed that the rotation of the flow helps stabilize the flow under the valve cone and equalize pressure fields, thus reducing the pipe's vibrations. The valve stem is terminated with a flat-bottomed cone and a punch. The input section consists of a cone with the apex angle of $\alpha = 60°$, the output diffuser has a 7° enlargement. The cone, together with the cone seating, forms a nozzle with jump enlargement. The sudden enlargement of the profile stabilizes the flow-separation effect from the cone's surface. This conception is required especially when the valve is only partly open and the pressure ratios are high. Step changes of pressure in impact waves then occur in lower Mach number values. The total pressure on the input p_{1c} = 27.18 MPa and the total temperature on the input T_{1c} = 868.15 K. The extent of the calculated alternatives is illustrated in Tab. 1.

Tab. 1: The extent of the calculated alternatives.

Name	*Marking*	*Extent*		
Relative lift	$\overline{h} = h/D$	*0.15*	*0.3*	*0.4; 0.45*
Pressure ratio	$\varepsilon = p_3/p_{1c}$	*0.955 ÷ 0.965*	*0.88 ÷ 0.99*	*0.98 ÷ 0.999*

h ... the cone's lift
D ... diffuser's throat diameter
p_3 … pressure behind the diffuser

[*] Ing. Lukáš Bednář, Ing. Ladislav Jirka and Ing. Ladislav Tajč, CSc.: ŠKODA POWER s. r. o., Tylova 1/57, 301 28 Plzeň; CZ, e-mails: lukas.bednar@doosan.com, ladislav.jirka@doosan.com, ladislav.tajc@doosan.com

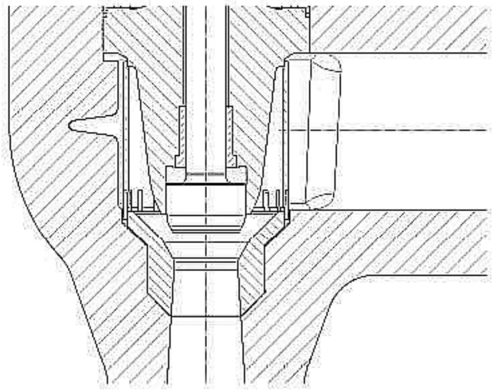

Fig. 1: Cross-section detail of the control valve.

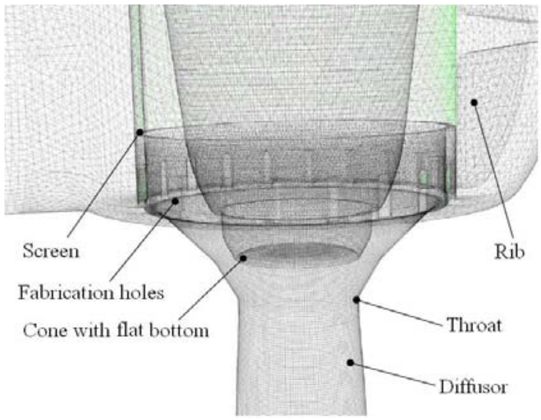

Fig. 2: Detail of important sections of the valve in a 3D model.

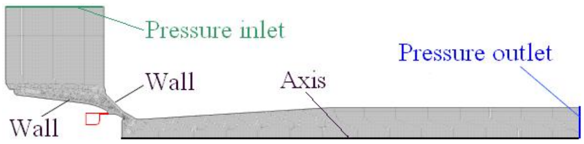

Fig. 3: 2D grid of the axially symmetric model with specified boundary conditions.

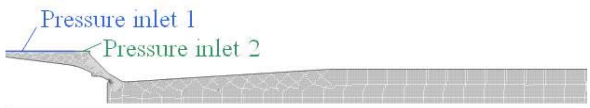

Fig. 4: 2D grid of the axially symmetric model with specified boundary condition on the input.

Two grid alternatives were used for the 2D calculation. They are illustrated in Figures 3 and 4. In the first grid alternative, the protective screen was not modeled. The flow rotation was specified for the entire input diameter. Four deviations from the radial direction were selected, i.e. 0°, 15°, 30° and 45°. The second 2D grid (Fig. 4) contains the protective screen. The input section of the calculation grid was removed because the rotation adjustment for a porous boundary condition under the specified angle could not be preset. This is why the entry is moved to the screen area and is divided into two parts – the top section (pressure input 1) with radially drilled holes and the bottom (pressure input 2) with apertures for steam entry under different angles. Basic simplification of both grids regards axially symmetrical flow.

In creation of the 3D grid illustrated in Fig. 2, attention was paid to the highest possible shape identity with a real valve. Only the protective screen, which contains about one thousand perforations, was not modeled directly, but by means of a porous medium. This concerns an area where the pressure resistance of the medium flow increases. The bottom directional apertures were created directly. For the 3D model, only a calculation with relative lift was performed $\bar{h} = 0.3$. In the calculations, non-dimensional mass flow was considered $q = \dot{m} / \dot{m}_{tkr}$. The calculated mass flow $\dot{m}$ refers to the critical mass flow $\dot{m}_{tkr}$. It is calculated for the flow area inside the diffuser throat.

3. Simulation on 2D axially symmetrical models

On the first 2D grid, stationary and non-stationary calculations with 0° diversions from the radial entry and stationary calculations with 15°, 30° and 45° diversions from the radial entry were simulated. For the first mentioned alternative, a calculation both by application of ideal gas modification and the UDF function of Redlich-Kwong's equation was considered. On the second grid, only stationary calculations were simulated for the deviations of 0°, 15°, 30° and 45° from the radial entry on input 2. The flow characteristics for alternatives on the first grid are illustrated in Fig. 5 and for the second grid in Fig. 6.

It becomes clear that both grid alternatives show almost identical values. Pressure loss in the protective screen is negligible. Deviation of flow from the radial direction does not show visible differences in mass flows through the valve. It does not matter whether the rotation concerns the entire input area or just its section. Modeling of the real gas properties leads to a slight increase of steam flow through the valve. Within the turbine's operational modes ($\bar{h} = 0.3$ a $q = 0.3$) this change will practically take no effect. At the turbine's full output, the pressure loss in the valve is lower than two percent. The simulation also proved that the stationary calculation practically does not differ from the non-stationary calculation. These findings, however, may not be true for short lifts of the valve cone, i.e. when transonic flow takes effect. This, however, was not tested in this case.

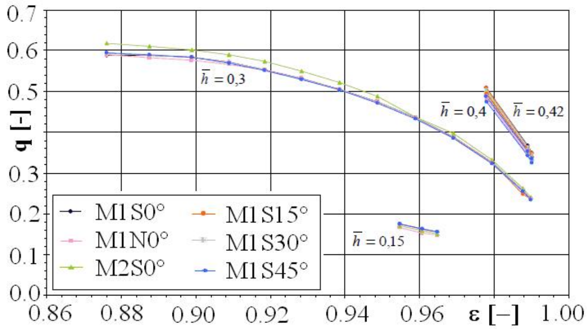

Fig. 5: Flow characteristics for options on the first 2D grid.

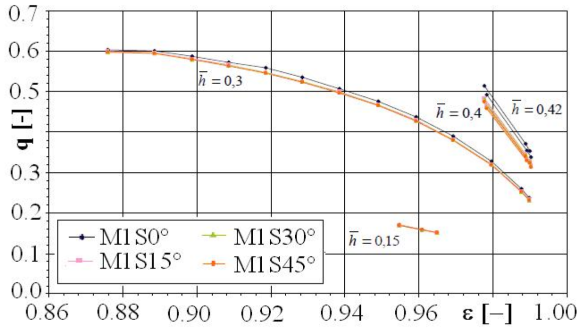

Fig. 6: Flow characteristics for options on the second 2D grid.

Fig. 7 illustrates velocity vectors in the diffuser under the valve cone. There is an area with back-flow immediately under the cone. The maximum velocity in the diffuser occurs in the vicinity of its wall. Alongside the flow, however, the velocity profile gradually becomes equalized (Fig. 8).

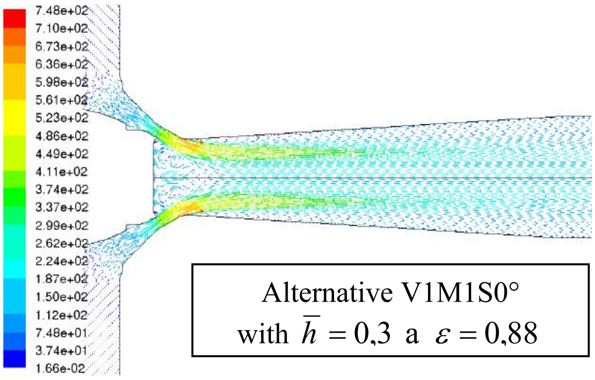

Fig. 7: Velocity vectors.

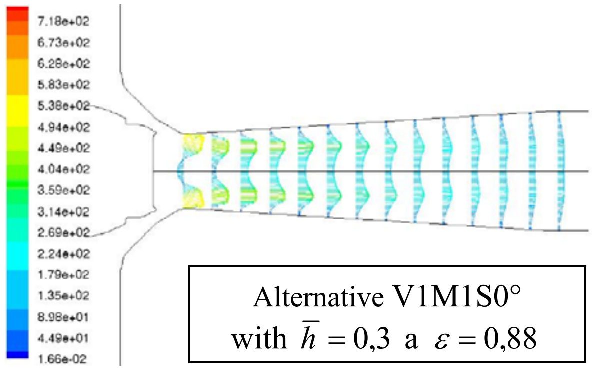

Fig. 8: Velocity profiles alongside the diffuser.

4. Simulation on a 3D model

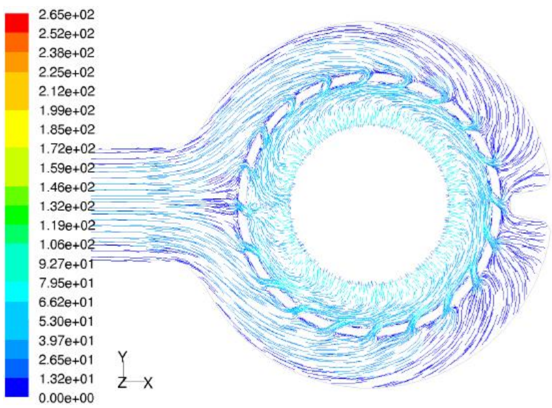

Fig. 9: Flow field in the half-point of the apertures' height; ε = 0.98.

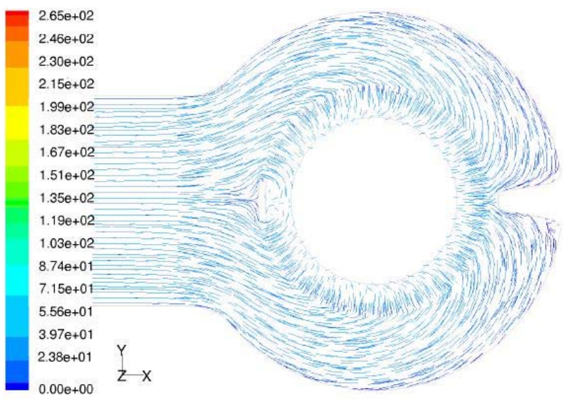

Fig. 10: Flow field in the area of circular apertures; ε = 0.98.

The model was created for the valve's relative lift $\bar{h} = 0.3$ and stationary simulation for three pressure ratios 0.97, 0.98 and 0.99. For the 3D model, the relative mass flow through the valve results in a lower value. This is caused by steam entry in one direction. It means that there are unbalanced conditions for steam entry through the directional apertures on the protective screen. The calculations prove that 79.5 per cent of the total mass flow runs through the porous zone of the screen in all the tested models. The rest flows through the directional apertures. An analysis of a flow field of the horizontal cross-section (Fig. 9) which is performed in the half-point of the apertures' height documents that re-distribution of the mass flow is caused by the direction of the apertures. It is favorable for the steam flow in the right section of the screen, in contrast to the other side. This Fig. 9 also documents a correct operation of the directional apertures, i.e. spinning of the steam. The stream-lines that are the closest to the internal circle belong to the steam which passed through the top apertures of the screen and flows down vertically. The flow field in the area of circular apertures is illustrated in Fig. 10. There are better conditions for steam entry in the front section of the screen than on the lateral sides. A rib on the opposite side enables a better balancing of the steam flow.

A curve at the maximal Mach numbers has a similar progression for the 3D model as for the 2D axially symmetric models, as it is documented in Fig. 11. However, it is moved by a certain amount to lower values. The pressure distribution on the cone is similar, with identical conclusions as with the 2D models, as it is documented in Fig. 12.

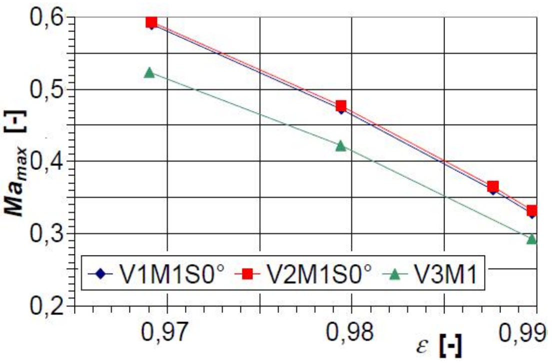

Fig. 11: Comparison of the maximal Mach numbers.

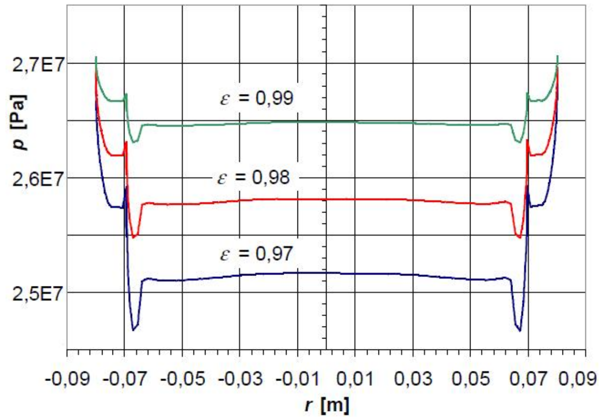

Fig. 12: Pressure distribution on the cone surface.

5. Conclusions

The 2D calculation grid of the symmetrical model simplifies the process of numerical simulation of a flow through the valve. However, it does not guarantee obtaining a reliable figure of the situation inside the valves. There are no significant differences between findings regarding the 2D and the 3D models for operational pressure conditions.

A varied angle of the flow's deviation from the radial direction on the entry into the valve has no effect on the passage of the mass flow through the valve. It is mainly affected by the geometry of the flow section of the valve and the pressure ratio.

A non-stationary calculation for operational pressure ratios on the valve does not produce visible alterations with regard to the stationary valve.

There is a backflow in the area immediately under the flat-bottomed punched cone.

Directional apertures on the protective screen cause unbalanced distribution of the mass flow around the screen's perimeter.

The pressure affecting the bottom of the cone is practically identical in operational modes of the turbine.

References

Jirka, L. (2007) Výpočtová studie proudění páry regulačním ventilem, Thesis, University of West Bohemia, Plzeň, Czech Republic.