30. Setkání kateder mechaniky tekutin a termomechaniky

HYDRO TERMO

22.-24.6. 2011

Špindlerův Mlýn

Jednotlivý příspěvek ze sborníku

TECHNICKÁ UNIVERZITA V LIBERCI

evropský sociální fond v ČR
EVROPSKÁ UNIE
MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY
OP Vzdělávání pro konkurenceschopnost
INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

Numerical Simulation of the Flow Field Near Hump

Milan MATĚJKA[1], Tomáš HYHLÍK[2]

[1] Ing. Milan Matějka, Ph.D., Czech Technical University in Prague, Faculty of Mechanical Engineering, Technická 4, 16607 Praha, milan.matejka@fs.cvut.cz
[2] Ing. Tomáš Hyhlík, Ph.D., Czech Technical University in Prague, Faculty of Mechanical Engineering, Technická 4, 16607 Praha, tomas.hyhlik@fs.cvut.cz

Abstract: *Paper deals with numerical and experimental solution of the flow field of hump. Main goal of this work is to study vortex structures. Studying of character of vortex structures is important to understand possibilities how to reduce their negative effect to the flow field. The hump is located in closed measurement area of Eiffel type wind tunnel. Commercial code Fluent was used to perform numerical solution.*

1. Introduction

Studying of character of vortex structures is important to understand possibilities how to influence them, namely how to reduce their negative effect to the flow field. Vortex structures have significant impact to the flow field. Focusing on comparison of numerical simulation and experimental data of the flow field is important part of research. Main reason of this work is to visualize and identify vortex structures [1, 2 and 3], in the flow field and to verify numerical model. Use of numerical simulation is very important, because describing and visualizing of vortex structure in three-dimensional space from experimental data is very difficult.

2. Numerical simulation

Steady numerical simulation of the flow field near model of hump was done by using commercial code Fluent. Mesh consists of about 6 million cells, see Fig. 1, for more information see [4]. Numerical simulation is based on pressure correction SIMPLE method. Convective terms are discretized using second order upwind. For the turbulent modeling RANS approach with SST k - ω model is used.
Unsteady numerical simulation of the flow field with influence of synthetic jet was also done using commercial code Fluent. In this case half of the channel is simulated on 12 million cells using non iterative time advancement method with second order implicit scheme. Fractional step scheme is used for pressure velocity coupling. Convective terms are discretized using second order upwind scheme. Turbulence modeling is based on Detached Eddy Simulation variant of Spalart-Allmaras model.

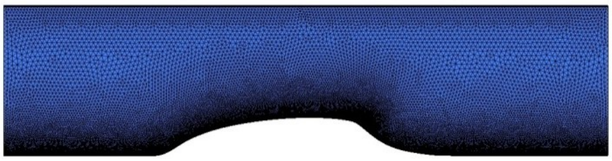

Fig. 1 *Mesh of calculated area*

3. Results

In Figure 2 the position of parallel plains with respect to hump location is marked. Fig. 3 represents velocity distribution - parallel plain in position 445 mm for input velocity 8 m.s^{-1}. In Fig. 4 vortex structures represented by Swirling strength are visible. Colors in Fig. 4 represent values of swirling x-velocity. Blue one corresponds to a clockwise direction and red one to the counter clockwise.
Synthetic jet with carrying frequency f_c = 370 Hz and modulation frequency f_{AM} = 60 Hz was used. More information about amplitude modulation see [5]. Fig. 5 represents half of channel (on the left site of the channel is center of the real channel) with vortex structures represented by swirling strength intensity. Changing (diameter and character) of cylindrical vortex incipient due to

interaction of free stream and synthetic jet is clearly visible. At the starting point is vortex as a cylinder with small diameter. Further in the direction of the flow due to interaction between the consecutive vortexes, their diameters grow and inside structure consist of small vortexes. Comparing of the flow filed with and without synthetic jet is clearly visible, that size and character of vortexes structures is different.

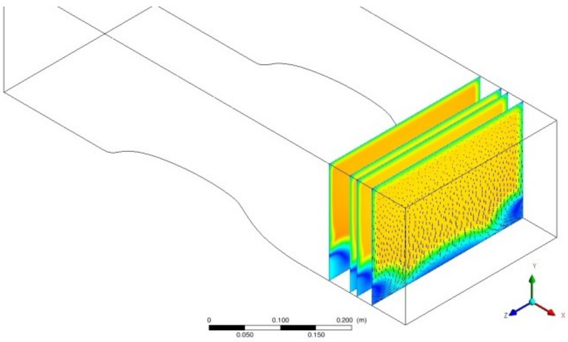

Fig. 2 Parallel plains for comparison of numerical solution and experimental data. Distance of plains is 445, 486, 500 and 530mm from starting point of hump. Dimension of plain 200 x 300 mm.

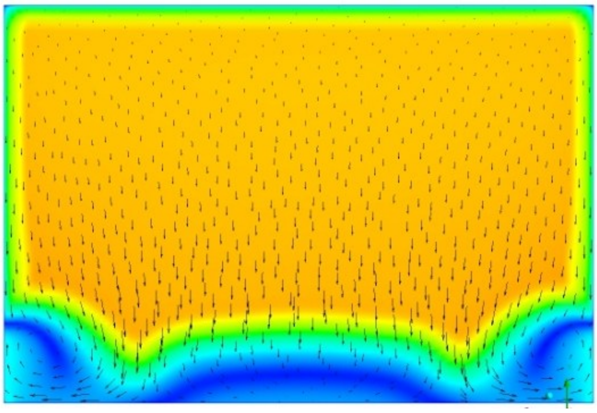

Fig. 3 Velocity distribution - parallel plain in position 445 mm for input velocity 8 m.s^{-1}, steady simulation.

In Fig. 6 is development of magnitude vertex average velocity in dependence on time. On the left side is visible influence of synthetic jet to the average vertex velocity. Synthetic jet was turn off on time approximately 0.29 s. On next three figures 7, 8 and 9 represents flow field of half channel after turning off synthetic jet on different time. There is no visible significant formation of vortex structures which are visible from the stationary calculation, Fig. 4.

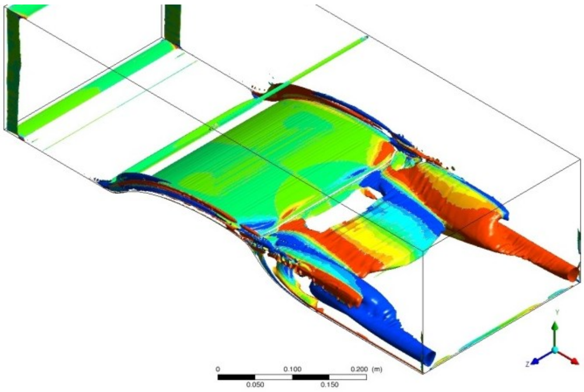

Fig. 4 Swirling strength for velocity 8 m.s^{-1}, steady simulation.

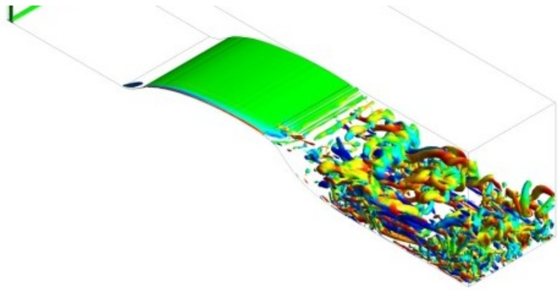

Fig. 5 Swirling strength for velocity 8 m.s^{-1}, unsteady simulation with influence of synthetic jet

.

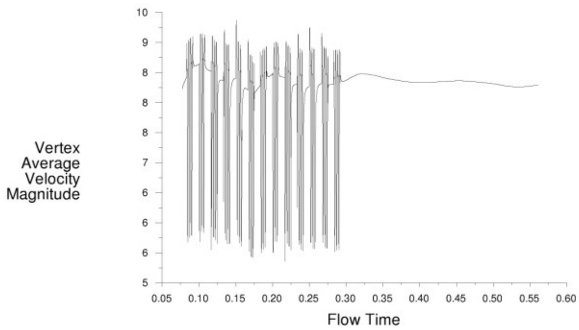

Fig. 6 Development of magnitude vertex average velocity in dependence on time, 2 mm from output slot of synthetic jet.

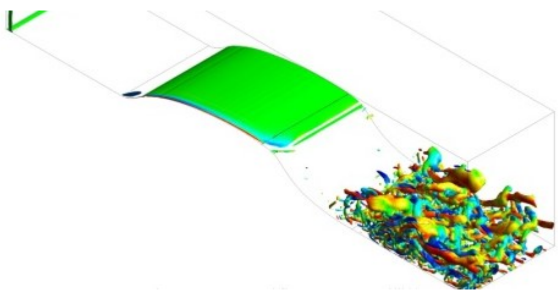

Fig. 7 Swirling strength for velocity 8 m.s^{-1}, unsteady simulation without influence of synthetic jet, t = 0,293 s

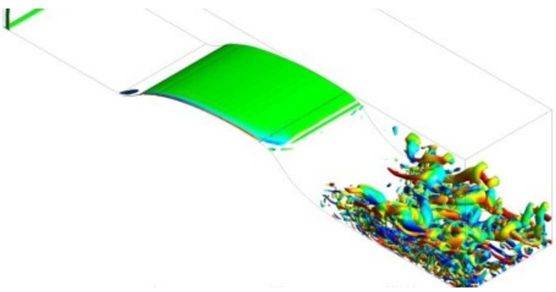

Fig. 8 Swirling strength for velocity 8 m.s^{-1}, unsteady simulation without influence of synthetic jet, t = 0,443 s

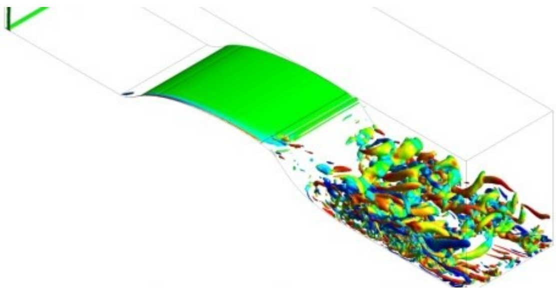

Fig. 9 Swirling strength for velocity 8 m.s^{-1}, unsteady simulation without influence of synthetic jet, t = 0,556 s

4. Experimental data

Experimental data were obtained by using traversing hot wire probe in position 445 mm from starting point of hump. Input velocity was set up to the value 8 m.s^{-1}. Intensity of turbulence in the input part of measurement area was from 0.4 to 0.8 with respect to the velocity. Sizes of measured areas are smaller due to limits of measuring techniques – about 240 x 182 mm.

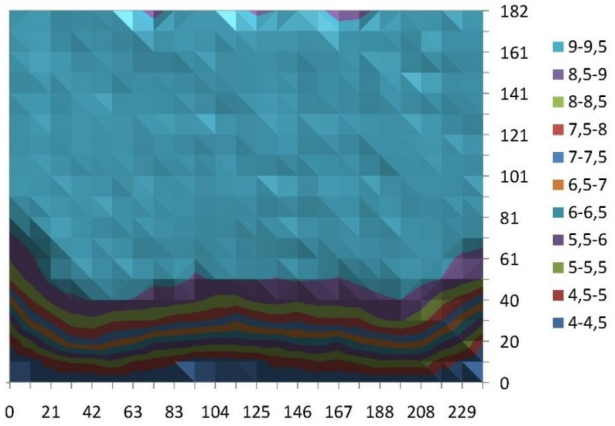

Fig. 10 Velocity flow field without influence of synthetic jet, in parallel plain in position 445 mm for input velocity 8 m.s^{-1}, HW.

Comparison of measured and calculated velocity of the flow field for steady case (without synthetic jet), see Fig. 9, the appreciable differences for 8 m.s^{-1} obtained from experimental measurements and numerical solution is not visible.

5. Conclusion

In the case of steady simulation without influence of synthetic jet big and stable vortex structures were created. Character of the flow field behind hump in the second case, with influence of synthetic jet, is different. Stable vortex structures by complicated vortex structures were replaced. That is caused due to the synthetic jet. In case of unsteady numerical simulation, after turning off synthetic jets, no dominant structures were created. That can be caused due to unsuitable numerical model or boundary conditions.

6. Acknowledgement

The work has been supported by Ministry of Education, Youth and Sports of the Czech Republic within project No. 1M06059. Support by the Czech Science Foundation under grants No. GA 101/08/1112

7. References

[1] Kolar, V. (2007) Vortex identification: New requirements and limitations, in: International Journal Heat and Fluid Flow 28 (2007), pp.638-652.

[2] C. H. Berdahl and D. S. Thompsont (1989) Eduction of Swirling Structure Using the Velocity Gradient Tensor, in: AIAA JOURNAL, Vol. 31, No. 1, January 1993, pp. 97-105.

[3] Zhou, J., Adrian, R.J., Balachandar, S., Kendall, T.M., (1999) Mechanisms for generating coherent packets of hairpin vortices in channel flow. J. Fluid Mech. 387, 353–396.

[4] M. Matějka, T. Hyhlík, P. Pick: Comparison of Numerical and Experimental Methods of Solution of the Flow Field of Hump, 17th International Conference Engineering Mechanics 2011 Svratka, Czech Republic, 9 – 12 May 2011

[5] Matějka, M. - Pick, P. - Procházka, P. - Nožička, J.: Experimental Study of Influence of Active Methods of Flow Control on the Flow Fileld Past Cylinder. Journal of Flow Visualization and Image Processing, 2009, vol. 2009, no. 4, p. 353-365. ISSN 1065-3090.