18th International Conference
ENGINEERING MECHANICS 2012
Svratka, Czech Republic, May 14 – 17, 2012
pp. 122–123
Paper #303

DES SIMULATION OF SEPARATION CONTROL FOR FLOW OVER HUMP

T. Hyhlík*, M. Matějka, V. Skála*****

Abstract: *Article deals with numerical solution of the separation control using Detached Eddy Simulation turbulence model. Numerical computation was done using commercial code Fluent. Vortex structures was visualized from the numerical solution. The strong influence of side walls to the character of the flow field is visible. The effect of the flow control was observed. Computed flow field is compared with experimental data obtained from hot wire measurement in the traversing plane. Significant deviations between numerical simulation and hot wire measurement are found. Numerical simulation of the three dimmensional flow controlled flow field is still challenge for present CFD.*

Keywords: ***Detached Eddy Simulation, Flow Control, Hump, Vortex Structures***

1. Introduction

Separation occurs in the wide range of application in the engineering practice. Separated flow has negative impact on performance because it reduces pressure loss, increase noise etc.

Control of turbulent flow by using oscillatory perturbation generated by synthetic jet actuator can be effective in influencing flow separation. The main advantage of flow control by using synthetic jet actuators against steady suction or blowing is their energy consumption and not least they do not require piping system. Oscillatory flow control is effective mainly in the case when introduced two dimensionally because there are generated vortex structures which are responsible for the momentum transfer across the shear layer.

Our geometrical configuration is inspired by the work of Seifert & Pack (2002) which also forms the basis for few CFD validation workshops but the numerical simulation of flow controlled separation is still an open problem. Our previous works were focused to visualization and identification of vortex structures, pressure measurement, constant temperature anemometry measurement and to the methods of flow control.

2. Numerical Simulation

Unsteady numerical simulation of the flow field with influence of synthetic jet was done by using commercial code Fluent. In this case half of the channel is simulated on 14.5 million cells by using non iterative time advancement method with second order implicit scheme. Fractional step scheme is used for pressure velocity coupling. Convective terms are discretized by using bounded central differencing scheme in momentum equations otherwise second order upwind scheme is used. Turbulence modelling is based on Delayed Detached Eddy Simulation variant of SST k-ω model. The inflow velocity of $8\ ms^{-1}$ was set up, i.e. $Re = 215000$. Velocity boundary condition close to the synthetic jet actuator slot exit was set with carrying frequency $fc = 370\ Hz$ and modulation frequency $f_{AM} = 60\ Hz$ and with amplitude $17\ ms^{-1}$.

* Ing. Tomáš Hyhlík, Ph.D.: CTU in Prague, FME, Department of Fluid Dynamics and Thermodynamics, Technická 4, 166 07, Prague, CZ, e-mail: tomas.hyhlik@fs.cvut.cz

** Ing. Milan Matějka, Ph.D.: CTU in Prague, FME, Department of Fluid Dynamics and Thermodynamics, Technická 4, 166 07, Prague, CZ, e-mail:milan.matejka@fs.cvut.cz

*** Ing. Vladislav Skála: CTU in Prague, FME, Department of Fluid Dynamics and Thermodynamics, Technická 4, 166 07, Prague, CZ, e-mail:VladislavSkala@seznam.cz

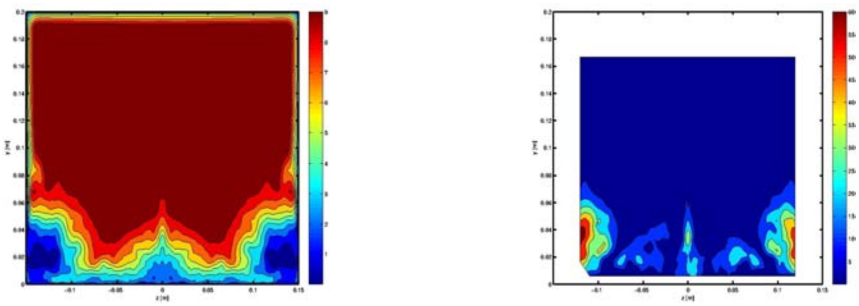
3. Results

The figure 1b shows complex three dimensional vortex structures behind the hump. Vortexes are visualized by using swirling strength method where colour is based on component of swirling velocity in the direction of the main flow. The influence of synthetic jet generator can be observed on the creation of downstream traveling vortex packs. Observed vortex packs are present through the whole width of the channel.

Velocity profile from the numerical computation in the traversing plane (traversing plane is marked in red in the figure 1a) is presented in the figure 2a. Significant deviation in the regions close to the side walls can be seen. Maximum observed deviation close to the left side wall is very close to 60 %.

(a) Location of traversing plane behind the hump *(b) Visualized vortex structures behind the hump*

Fig. 1: Computational domain



(a) Velocity profile from the numerical computation in [ms^{-1}] *(b) Normalized deviation of the measured and computed mean velocity in [%]*

Fig. 2: Mean velocity profile in the traversing plane

4. Conclusion

Investigation of the flow field in the channel with wall hump which is affected by the synthetic jet generator has been made. The focus was placed on the comparison of measured and computed velocity data in the traversing plane. The significant deviations close to the side wall was observed. Deviations can be caused by the influence of traversing probe to the flow field and mainly by lack of knowledge about boundary conditions in the slot exits.

Acknowledgement

The work has been supported by Czech Science Foundation under grant No. GA 101/08/1112.

References

Seifert, A. & Pack L. G. (2002) Active Flow Separation Control on Wall-Mounted Hump at High Reynolds Numbers. *AIAA Journal*, Vol. 40, No. 7, pp. 1363-1372