Assessment of Turbulence Models for Turbulent Flow over Backward Facing Step

Anwar-ul-Haque, Fareed Ahmad, Shunsuke Yamada and Sajid Raza Chaudhry

Abstract— **Predication of reattachment length of separated shear layers in low Reynolds number turbulent flow is a challenging task to evaluate the capabilities of different turbulence models, especially for MEMS applications. Physical significances of backward facing step and its industrial applications were discussed. Flow analysis of backward facing step in 2D as well as in 3D were carried out by using Finite volume method, incompressible segregated scheme with SIMPLE algorithm for pressure velocity coupling. Although the recirculation created by backward facing step (BFS) is predicted by all turbulence models but ε based turbulence models underpredict the reattachment length of flow where as Shear Stress Turbulence model accurately predict the flow reattachment. Comparison of 3D and 2D results revealed that the separation point is fixed where as the reattachment points were different due to 3D nature of turbulence.**

***Index Terms*—Flow Separation, Turbulence Modeling, Re-attachment Length, Backward Facing Step**

I. INTRODUCTION

The topic under investigation is a very important one as flow over a backward facing step forms the basis of many real flow situations. It is often used as test cases for improvement of numerical schemes and is a classical problem in applied aerodynamics. Turbulent reattachment occurs in most of the engineering applications, like sudden enlargements in pipes and ducks, ignition and stabilization of the flame in a scramjet engine, prediction of wall heat transfer in PCB circuits and multiphase flow phenomenon in piston engines. Overall performance of many devices such as diffusers, turbine blades, micro electrical and mechanical devices *(MEMS),* leading edge vortex control of aircraft by using MEMS transducers and of aerodynamic bodies is greatly influenced by the flow separation. Experimental data for backward facing step is present in the literature for wide range of Reynolds numbers and at different expansion ratios [1], [2], [3] and [4]. Up to date, numerous experimental as well as computational studies have been carried out to analyse the flow over a backward facing step and to estimate the value of reattachment length. In comparison with the 2D results, value of reattachment length (i.e. the point where the separation disappears on the wall aft of the step) in 3D [5] is somewhat higher in the laminar flow ($R_e < 400$), considerably lower in the transition regime ($400 < R_e < 3400$), and to some extent lower in the fully turbulent flow regime ($R_e > 3400$). Geometry of step also plays a dominant role in recirculating regions for turbulent flows. In comparison with sharp step, a blunt edge [6] will produce higher turbulent intensity upstream of the step at fixed upstream velocity, which in result tends to increase the size of separation bubble. This is due to the fact that the additional turbulence added into upstream boundary layer.

Since Direct numerical simulations of most of the engineering and industrial applications are not possible due to limitations of the computational resources. Therefore industry requires need only the average quantities of the flowfield by solving Reynolds average Navier Stokes equations (RANS). Keeping above in view, numerical simulations for the estimation of reattachment length were carried out by using RANS with different turbulence models.

II. COMPUTATIONAL METHODOLOGIES

Numerical predictions of reattachment length with different turbulence models in 2D as well as in 3D were compared with the available experimental data of wind tunnel [7] and the separation point was determined by the numerical solution. This experiment was basically performed for a backward facing step, for a range of Reynolds number (based on step height) from 133 to 3693. At Reynolds number of 380, flow is laminar and transition occurs between Reynolds number of 600 and 1000. The height of step was 4 mm and width of the domain was 150 mm. Major dimensions of this geometry are shown in Table. 1

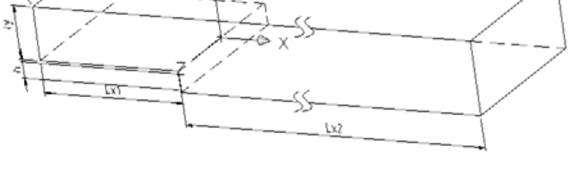


Fig. 1 Computational Domain

Manuscript received March 6, 2007. This work was supported by the office of NESCOM, U.E.T, Taxila by providing the computational support.

Sajid Raza Chaudhry & Anwar-ul-Haque is with Center of Excellences in Science and Applied Technologies (CESAT), Islamabad, Pakistan and also in the visiting faculty of University of Engineering and Technology, Taxila, Pakistan, (Phone No: +92-51-90166237; Fax No: +92-51-9258435; e-mails: sajidraza039@hotmail.com, anwar_haque02000@yahoo.com). Fareed Ahmad is also with CESAT, Islamabad, Pakistan, (e- mail: fareedmuet@yahoo.com)

Shunsuke Yamada is with Department of Mechanical Engineering, Tokyo University of Science, Japan, (E mail: j4505703@ed.kagu.tus.ac.jp)

Table.1 Major dimensions of the selected geometry of BFS

Position of inflow boundary	$x = -Lx_1 = 127.5h$
Position of outflow boundary	$x = Lx_2 = 30h$
Position of upper wall	$y = Ly = 1.5h$
Width of domain	$z = Lz = 37.5h$
Expansion ratio	$\alpha = (Ly+h)/Ly = 1.6$

Fully structured multiblock grid was generated with grid refinement near the wall and in the vicinity of step and is shown in Fig. 2. The grid size in 2D was 0.17 million with first fluid grid cell placed at 9µm, which resulted in y^+ value of unity and is shown as Fig. 3. Lower plot of Fig. 3 shows the distribution of y^+ on the step side wall and upper plot shows the upper side wall. Downstream of the step, 105 points were used in the recirculation region with most of them placed close to the wall. In order to improve the resolution of flow, grid was clustered in the streamwise direction near the recirculation region. Although, in low Reynolds number flow modeling, large computational cost is required due to the use of fine grid. It is because the viscous sublayer near the wall cannot be resolved by using the coarse mesh and the damping functions used to model the singular behaviour of the turbulent kinetic energy (near the wall) gives the false value. Mesh independence was also investigated and the solutions obtained with different turbulence models were also investigated. Negligible effect in the value of reattachment length was observed by using grid of more then .17 million.

A uniform free stream velocity inlet boundary condition for incompressible flow was applied upstream of the step and no restriction on the flow was applied at the step edge. Pressure outlet boundary condition was applied at the outflow plan which was positioned far downstream of the step (with zero gauge pressure) to reduce the influence of the outflow conditions. In one of the numerical study [8], it was found that the computational domain should be sufficiently long (fineness ratio>30) to get a good agreement between the numerical results and experimental data. On the solid surfaces (including the upper wall) noslip boundary condition (zero velocity) was applied. In experiment [7], turbulence intensity at the step edge is about 3%. Since the flow of rotating blower is unstable, therefore as the Reynolds number is increased then the turbulence intensity value is uniform by the stability of blower. Although in the experiment [7] the value of turbulence intensity is known, however a study was carried out to find the effect of turbulence intensity on reattachment length against a fixed Reynolds number. The results are summarized in Fig. 4, which revealed that velocity profile is sensitive to input value of turbulence intensity. Inlet and outlet boundary conditions are set based on the turbulent kinetic energy and turbulent dissipation rate. The free stream turbulent kinetic energy was calculated by using the relationship, given below:

$$K = \frac{1}{2}\left(u'^2 + v'^2 + w'^2\right) \qquad (1)$$

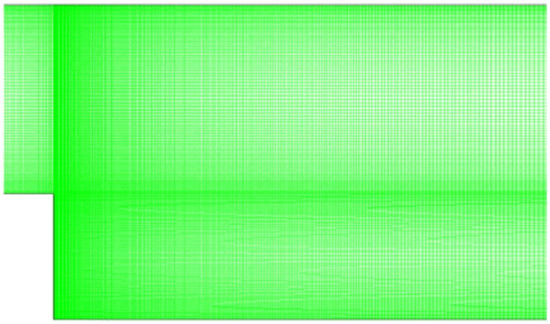


Fig. 2 Computational grid refinement near step

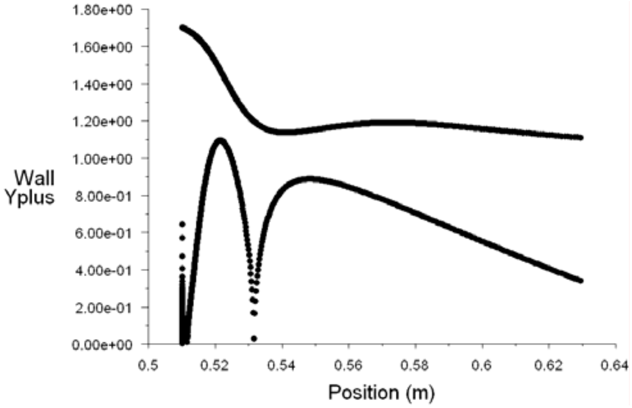


Fig. 3 Distribution of wall y^+ for upper and lower wall aft the step edge

Where, u', v' and w' are the turbulent fluctuation velocity components in x, y and z directions respectively. The dissipation rate 'D' can then be calculated from (2).

$$D = \frac{C_\mu^{\,0.75} K^{1.5}}{\kappa L} \qquad (2)$$

Where $C_\mu = .09$, $\kappa = 0.4$ and L is the characteristic length and is taken equal to the inlet height (10 mm).

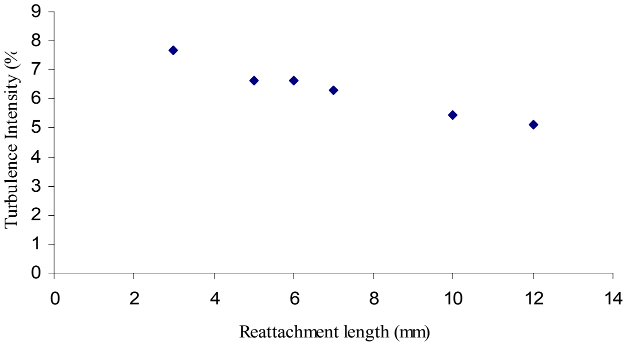


Fig. 4 Results of sensitivity analysis of turbulence intensity for $R_e = 3270$ with SST Model

One of the difficulties in the numerical simulation of backward facing step is that the region of circulation is highly skewed with respect to the numerical mesh, causing what is known as "*numerical diffusion*". As a result, the prediction of such quantities as the reattachment length tends to compare poorly with experimental data. Therefore double precision solver was used in all computation to tackle the problem round-off error, which is imposed by the finer mesh in the near-wall regions. Segregated solver was used to handle this incompressible flow problem, which presents a special problem. It uses pressure as a guess to solve u,v,w momentum equation in turn. First we discretize the Partial differential equations using the finite volume method [9]. Take the equation of motion to discretize the ‘u’ component of velocity and get an algebraic equation, and then solve for ‘v’ component of velocity by taking ‘u’ and ‘v’ components from the previous iteration. After discretization we have a linear system to solve ‘u’ and one linear system to solve ‘v’ i.e. after discretization, one can get equation for each velocity components u, v and w. Momentum equations are used to solve for U and V in sequence with pressure gradients as the source terms as described earlier. However the continuity equation can not be used directly as an equation for pressure. So continuity equation is solved in form of pressure correction and update pressure. This restriction introduces the computational difficulty that the continuity equation contains only velocity components, and there is no obvious link with the pressure as there is for compressible flow through the density. This problem can be tackled by using SIMPLE [10]. This algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field. The convergence criteria were set equal to 10^{-6} by using Fluent 6.1and solution did not change from the solution obtained with this criteria.

II. TURBULENCE MODELING

Uptil now, numerous turbulence models from one equation model to higher order models have been implemented for the estimation of reattachment length. Present research focuses on generation of multiblock structured mesh and flow field model which can accurately resolve the flowfield features and to provide the quantitative as well as qualitative comparison of backward facing step at low Reynolds number. The turbulence models used in the present study include k- ε, RNG k- ε, SA and SST models. Transport equation/equations of these turbulence models are solved in combination with the Reynolds Averaged momentum and continuity equations, without relying on wall functions, including non-equilibrium wall function [11]. Brief descriptions of these turbulence models are given as follows:

Standard k-ε model is a two equation model in which two partial differential transport equations are solved for turbulent kinetic energy and turbulent dissipation rate. This model does not ensure that the turbulent normal stresses are positive, which is contradictory to the real physics of the flow. Although this model is famous due to its robustness and economy, however it performs poorly [12] when applied for the non-equilibrium boundary layers with adverse pressure gradients. RNG k-ε model [13] provides an option to cater the effects of swirl by modifying the turbulent viscosity. This model is more computationally expensive then standard k-ε model due to additional term in the ε equation. Spalart-Allmaras (SA) model [14] is a low-Reynolds number turbulence models and is specifically developed for wall bounded aeronautical applications with adverse pressure gradients. It is one equation model which solves a single partial differential equation for variable $\tilde{\nu}$, which is related to the turbulent viscosity. This model can directly be applied through out the boundary layer if the near wall mesh is fine enough to resolve the gradients. In SA model turbulent eddy viscosity is not specified with characteristic velocity and length scales, rather then solves by a transport equation which is given below:

$$\frac{\partial \tilde{\nu}}{\partial t}+\frac{\partial \overline{u}_j \tilde{\nu}}{\partial x_j}=Cb1\ \left[1-f_{t2}\right]\tilde{S}\tilde{\nu}+\frac{1}{\mathrm{Re}}\frac{1}{\sigma}\frac{\partial}{\partial x_j}\left[\left(\nu+\tilde{\nu}\right)\frac{\partial \tilde{\nu}}{\partial x_j}\right]$$

$$-\frac{1}{\mathrm{Re}}Cw1\,fw\left[\frac{\tilde{\nu}}{d}\right]^2+\frac{1}{\mathrm{Re}}\frac{Cb2}{\sigma}\frac{\partial \tilde{\nu}}{\partial x_j} \qquad (3)$$

Where $\tilde{S}$ is strain vorticity vector, $Cb1$ and $Cb2$ are constants and $Cw1$ and fw are auxiliary functions. Menter [15] combined the k-ε and k-ω models on a way that would allow them to be used in the regions where they attain the best advantage. This model is quite similar to the k-ω BSL model, except that the constants for the inner model are slightly changed and the definition of eddy viscosity was redefined as:

$$\mu_T=\mathrm{Re}.\min\left[\frac{\rho k}{\omega}\frac{a_1\rho k}{\Omega F_2}\right] \qquad (4)$$

SST model uses k-ω model near the wall but switches trough to k-ε model away from the wall. Furthermore, the model limits the shear stress according to experienced observation. Studies showed that this limitation much improved results in separated flows. Among eddy-viscosity models, this model performs fairly well in many applications. The switching function is defined as follows:

$$\mathrm{arg}_1=\min\left[\max\left(\frac{\sqrt{k}}{0.09\omega d},\frac{1}{\mathrm{Re}}\frac{500\nu}{d^2\omega}\right),\frac{4\overline{\rho}\sigma_{w^2}k}{CD_{k\omega}d^2}\right] \qquad (5)$$

$$CD_{k\omega}=\max\left(2\overline{\rho}\sigma_{w^2}\frac{1}{\omega}\frac{k}{x_j}\frac{\omega}{x_j},10^{-20}\right) \qquad (6)$$

$$F_1=\tanh\left(\mathrm{arg}_1{}^4\right),F_2=\tanh\left(\mathrm{arg}_2{}^2\right) \qquad (7)$$

$$\mathrm{arg}_2=\max\left(2\frac{\sqrt{k}}{0.09\omega d},\frac{1}{\mathrm{Re}}\frac{500\nu}{d^2\omega}\right) \qquad (8)$$

Where; a_1 is constant, d is the distance to the nearest wall, Ω is mean rate of rotation tensor, ω is vorticity, F_1 and F_2 are blending functions used for switching of k- ω model (inside the boundary layer) to k- ε model (away from surface).

III. RESULTS AND DISCUSSIONS

Re-attachment length is a commonly used parameter to determine the ability of a turbulence model to correctly simulate the flow over backward facing step. A key measure of the computational accuracy of any numerical scheme is the prediction of the reattachment point. This parameter is the distance from the step to the position on the wall, at the bottom of the channel, at which the velocity along the channel becomes positive. Different options available for the estimation of the reattachment length are by measuring it with the help of a scale from a simulation result of axial velocity, by defining a line in the numerical simulation, which is very close to the wall and then plotting the variation of axial velocity and by finding the zero streamwise wall shear stress location. The first technique will give a rough estimate of reattachment length and the second technique has given very precise results. Sample plot of x-velocity for a line located very close to the wall is shown in Fig. 5. Sensitivity of the re-attachment length was done by checking the difference in the values of axial velocity along two different lines lying close to the wall. Interaction between separation zone and circulation zone was also observed. Flow was separated below the step edge and free shear layer was developed behind the flow separation region. The velocity vector distribution over the step region is shown in Fig. 6.

Table. 2 show the comparison of the reattachment length estimated by different turbulence models with that of experiment. No separation vortex was captured with k-ε, RNG k-ε and SA turbulence models in 2D on opposite side wall except SST model which has shown very weak vortex. Use of k-ε and SA turbulence models does not allow capturing the development of primary vortex core correctly. However, pressure drop near the step edge was also observed in all simulations. Underprediction by use of k-ε model at supersonic flow over a small step of height 4mm was also observed by us [16]. However, the Shear Stress model (SST) has shown to provide a good qualitative description of the low Reynolds number flow and has predicted well the reattachment length. This model was also found capable of predicting the operational temperature of circuit board-mounted component within 5% difference [17] and was found suitable for generation of accurate temperature profiles for reliability assessment of electronic components.

Table. 2 Comparison of measured and predicted reattachment length at different Reynolds number in 2D on step side wall.

Re No	Reattachment Length Ratio				
	Exp	k-ε	RNG k-ε	SA	SST
3615	6.45	4.2	5.32	5.89	6.57
2976	7.6	5.1	5.98	6.93	7.89
2425	9.2	6.3	6.93	8.54	9.4

Overprediction in reattachment length in 2D incompressible flow [18] by using SST model was also observed by using WIND code [19]. Since actual engineering systems exhibit three dimensional behaviour. Therefore, 3D numerical experiments with different turbulence models were also carried out and the results are summarized in Table. 3. All results were taken by using multiblock fully structured grid of 3.6 million cells with $y^+ \approx 1$. All measurements were taken by taking 2D plan at the center of the step edge as the measurements in the experiment were taken by the micro flow sensors from 4 mm to 115 mm at the mid span of the test section of wind tunnel [7].

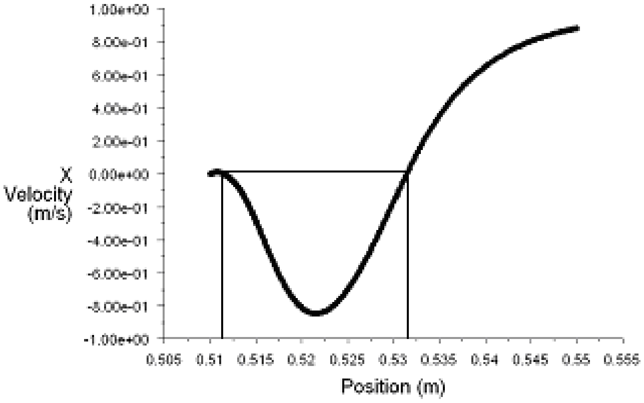


Fig. 5 Graphical representation of Re-attachment length

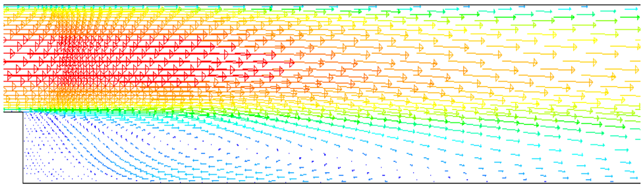
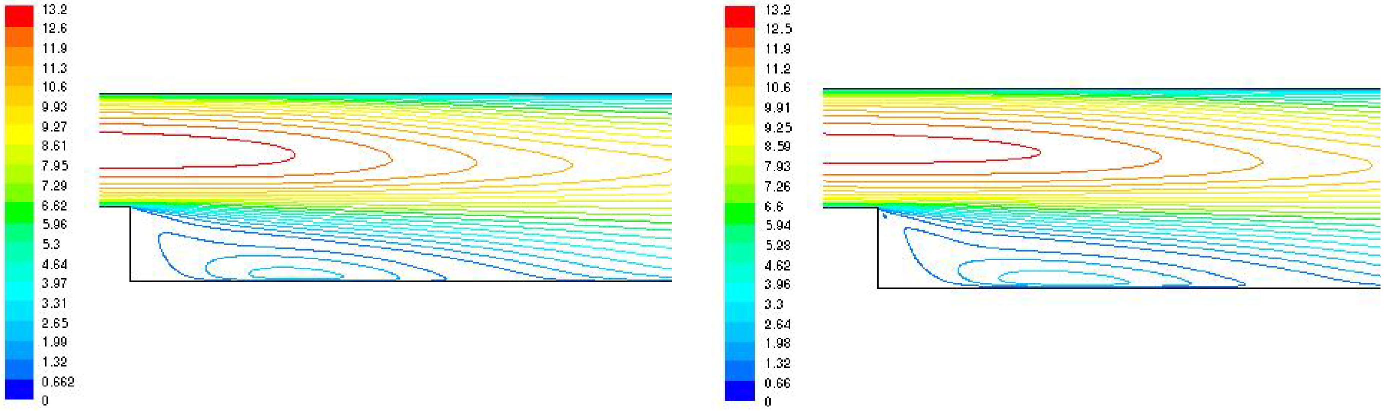


Fig. 6 Recirculation zone (velocity vector distribution)

Table. 3 Comparison of measured and predicted reattachment length at different Reynolds number in 3D on step side wall

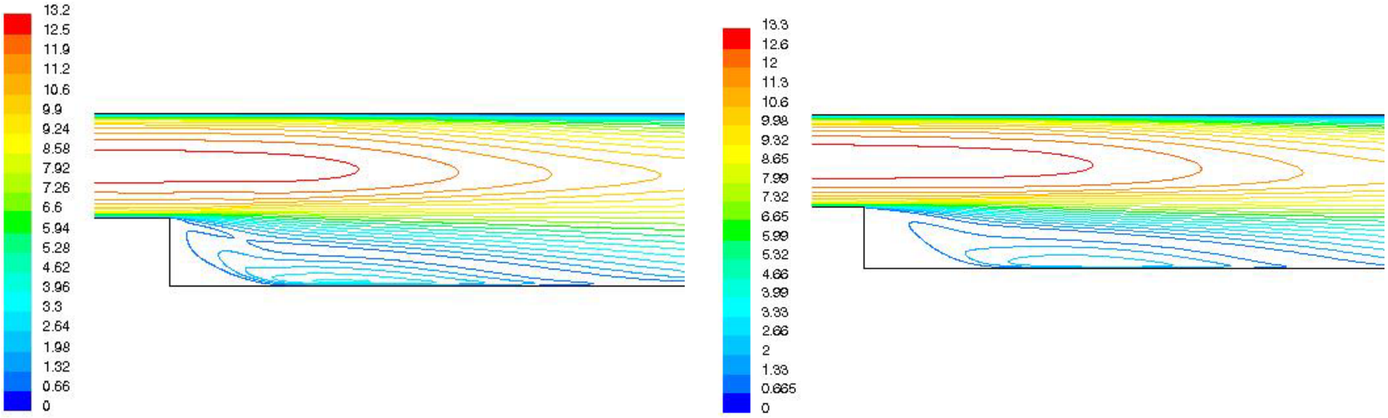
Re No	Reattachment Length Ratio				
	Exp	k-ε	RNG k-ε	SA	SST
3615	6.45	4.5	5.17	6.2	6.35
2976	7.6	5.3	5.87	7.23	7.48
2425	9.2	6.1	6.74	8.94	9.12

Almost all turbulence models have predicted the vortex behind the step in 2D and results are shown in Fig. 7. Moreover, the separation point is fixed in 2D as well as in 3D but the flow reattachment points are different in 3D. A plan was drawn close to the step side wall and x-velocity was plotted in the streamwise direction and is shown in Fig. 8. Difference in the values of reattachment lengths at different Reynolds number on step side wall is due to the fact that turbulence is 3D in nature. This figure shows that the reattachment points are different in 3D at different spanwise locations. Contours velocity magnitude is used for the visualization of vortex and a sample result is shown in Fig. 9. This flow topology also revealed that the strength of the vortex on step side wall can be decreased by increasing the Reynolds number. Unsteadiness in flow was observed by using Detached Eddy Simulation (DES) with SA model on a structured grid of 3.8 million size and is obvious from Fig. 10 and Fig. 11. Averaged value of reattachment length obtained at Re=3615 by using this technique is equal to 6.42. Fig. 12 shows that averaging process of RANS could not predict the unsteadiness in flow as compared with DES. However, DES solutions were not run for all cases as grid independent study is not possible in DES. Further investigations are required to compare the results computed with Large Eddy Simulation (LES) and Scale-Adoptive Simulation (SAS) approach [20].



(a) k-ε model

(b) RNG k-ε model



(c) SA model

(d) SST model

Fig .7 2D Contours of velocity magnitude with different turbulence models at $R_e = 3615$ with 3 % turbulent intensity at step edge

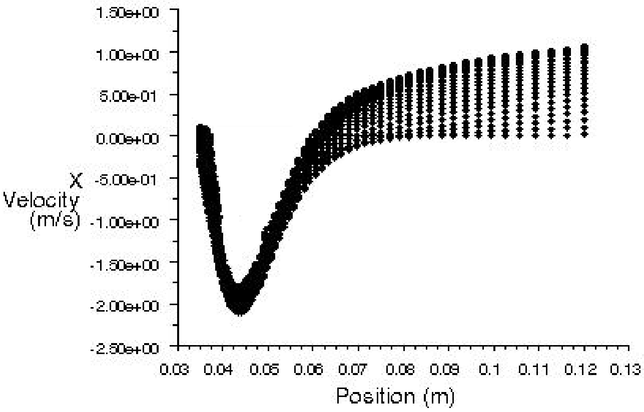


Fig. 8 x-velocity plots with SST model in 3D at $R_e = 3615$

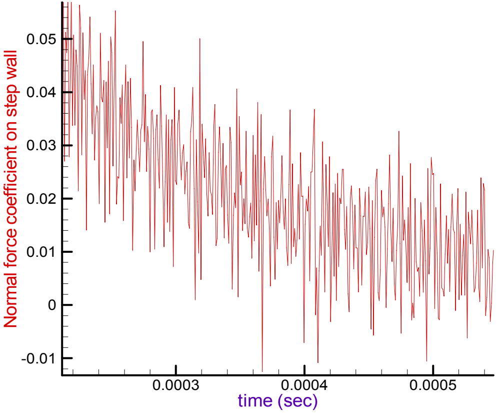


Fig. 10 Normal force co-efficient on step wall vs time (sec)

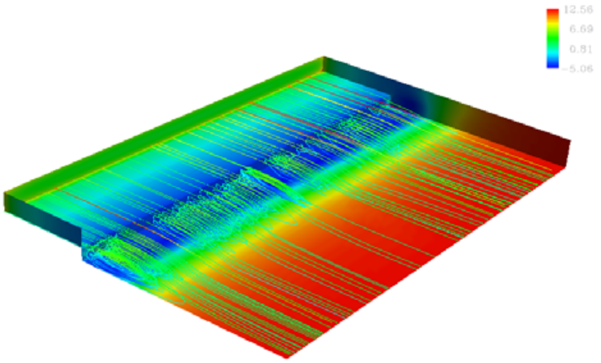


Fig. 9 Surface streamlines with surface contours of velocity magnitude at $R_e = 3615$

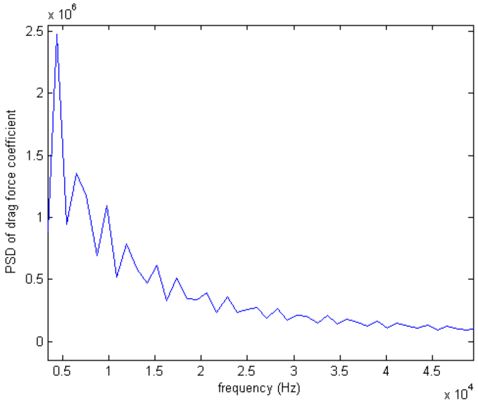
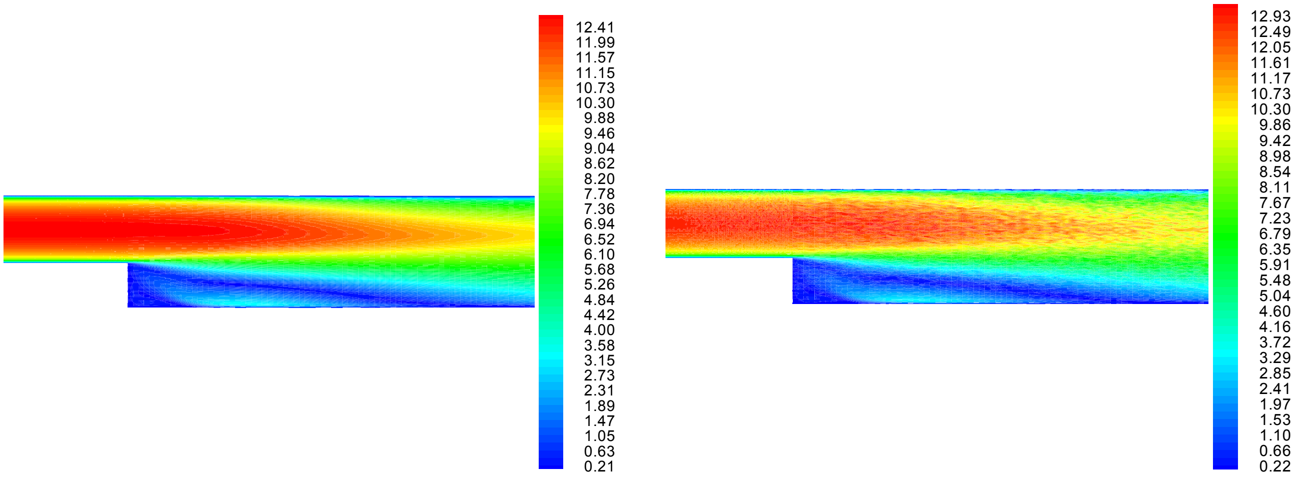


Fig. 11 FFT Results of drag force co-efficient



(a) RANS (b) DES

Fig.12 Contours of mean velocity (RANS Solution) and contours of instantaneous velocity (DES Solution) at 1.1 msec

IV. CONCLUSION

This paper concludes the fact that the difference in 2D and3D results of reattachment length is due to the difference of mass flow rate and the neglection of side-wall effects on the boundary layer development in 3D. The overall comparison indicates the quantitative adequacy of the SST turbulence model for low Reynolds number flow. Although the recirculation created by backward facing step (BFS) is predicted by all turbulence models but ε based turbulence models underpredict the reattachment length of flow where as Detached Eddy Simulation (DES), a hybrid RANS-LES method, accurately resolves the three dimensional vortical structure as compared with RANS for given free stream velocity and turbulence level.

ACKNOWLEDGMENT

Authors are thankful to to Mr. Adnan Asif, Country Manager of Nokia (Finland) in Pakistan for fruitful discussions on MEMS and PCB circuits.

REFERENCES

[1] B.F.Armaly, A. Li, J.H. Nie, "Measurements in three-dimensional separated flow", International Journal of Heat and Mass Transfer 46 (2003) 3573-3582.
[2] T. Lee and D.Mateescu, "Experimental and numerical investigation of 2-D backward-facing step flow", Journal of Fluids and Structures (1998) 12, 703-716.
[3] S. Aubrun, P.L. Kao, H.C. Boisson, "Experimental coherent structures extraction and numerical semi-deterministic modeling in the turbulent flow behind a backward-facing step", Experimental Thermal and Fluid Sciences 22 (2000) 93-101.
[4] Fessler J.R. and Eaton J.K., Particle Response in a Planar Sudden Expansion Flow, Experimental Thermal and Fluid Science, Vol.15(4), pp413-423,1997.
[5] J.H. Nie, B.F.Armaly, "Reverse flow regions in three-dimensional backward facing step flow", International Journal of Heat and Mass Transfer 47 (2004) 4713-4720.
[6] O. O. Badran and H. H. Brum, "The effect of inlet conditions on flow over backward facing step", Journal of Wind Engineering and Industrial Aerodynamics 74-76 (1998) 495-509
[7] Yamada S, Sagawa H, Okamoto S and Honami S, "A behavior of backward facing step flow in low Reynolds number", Proceeding on Int. Conf on jets, wakes and separated flows, ICJWF-2005.
[8] Muntean S., Buntic I., Ruprecht A. and Susan-Resga R., "A numerical investigation of the 3D swirling flow in a pipe with constant diameter. Part 2: Turbulent Computation", Workshop on Vortex Dominated Flows-Achievements and open problems, Timisoara, Romania, June 10-11, 2005
[9] Hirsch C., Numerical Computation of Internal and External Flows, Vol. I + II, John Wiley & Sons Ltd., Chichester, New York, Brisbane, Toronto, Singapore, 1988
[10] Vandoormaal J.P. and Raithby, "Enhancements of the SIMPLE method for predicting incompressible fluid flows, Nume Heat Transfer, 7:147-163, 1984
[11] Kim S.E. and Choudhry D, "A near wall treatment using wall functions sensitized to pressure gradients, In ASME FED vol. 217, Separated and Complex flows, ASME, 1995
[12] X.-D. Yang, H.-Y. Ma, Y.-N. Huang, Prediction of homogeneous shear flow and a backward-facing step flow with some linear and non-linear k- ε turbulence models, Communications in Nonlinear Science and Numerical Simulations 10 (2005) 315-328.
[13] F.S Lien and M.A. Leschziner, "Assessment of turbulence-transport models including non-linear RNG eddy-viscosity formulation and second-moment closure for flow over a backward-facing step", Computers and Fluids 23 (1994) 983-1004.
[14] P.Spalart and S.Allmaras, "A one-equation Model for Aerodynamic Flows," Technical Report AIAA 92-0439
[15] Menter, F.R., "Two-equation eddy viscosity turbulence model for aerodynamic flows", AIAA Journal 32(8), pp 269-289, 1994
[16] J. Khawar, A. ul-Haque and S. R. Chaudhry, "Validation of Turbulence Modeling in CFD2DV4 Code for wall bounded flows", 5th International Bhurban Conference on Applied Sciences and Technology, Bhurban, Pakistan, January 8-11, 2007. (submitted for publication)
[17] Peter Rodgers and Valerie Eveloy, " Application of low Reynolds number Turbulent Flow models to the prediction of electronic component heat transfer", 2004 Inter society Conference on Thermal Phenomenan, pp 495-503
[18] Driver D.M and Seegmiller H.L., "Features of a Reattaching Turbulent shear layer in divergent channel flow", AIAA Journal. Vol. 23, Feb.1985, pp 163-171
[19] "NPARC Validation Archive, study no. 1, Backward facing step", www.grc.nasa.gov/WWW/wind/valid/backstep01/backstep01.html
[20] Menter, F. R. Kuntz, M. and Bender, R., " A scale-adoptive simulation model for turbulent flow predictions, AIAA 2003-0797, 2003