Prediction of laminar flow over a back facing step using new formula of penalty parameter

Mahmud A. Mehemed Abughalia
Department of Mechanical Engineering, Al-Fateh University, Libya

Dedication: To the soul of my Fiancée, the architecture engineer Rima

Abstract

This paper introduces a numerical solution for flow over a back facing step. In this study, the continuity equation is replaced by a penalty function and a new formula of penalty parameter is derived and implemented. The new formula of penalty parameter is restricted only to the finite element method and the formula requires the existence of the second derivative of the trial function so that a rectangular element with exponential trial function is proposed and used.

A laminar flow over a back facing step is chosen as a test model to examine the numerical solution. The location of the reattachment point is captured accurately only if the penalty parameter is multiplied by a suitable factor and this factor depends on the Jacobian.

The current numerical results are validated through a comparison to available numerical and experimental results for the case of flow over a back facing step. The tested range of Reynolds number was 200, 400 and 1000 and the ratio between the step height to the duct height was 0.5. The reattachment point is tabulated against Reynolds number. The comparison shows that the finite element solution using the new penalty parameter is closer to the experimental results than the available numerical results.

Keywords: penalty method, back facing step, finite element, laminar flow, Navier Stokes equations, numerical solution.

1 Introduction

The Navier Stokes equations govern the physical behavior of most of the fluid flow applications. Thus they have attracted a great deal of attention by most engineering fields, chemical, civil, aeronautical, oil and mechanical engineering. These equations usually are solved numerically and the challenge of the

WIT Transactions on Engineering Sciences, Vol 59, © 2008 WIT Press
www.witpress.com, ISSN 1743-3533 (on-line)
doi:10.2495/AFM080031

numerical studies usually focuses on predicting more precise results for the case study.

The well known methods for solving the Navier Stokes equations are finite difference, finite volume and finite element method. The numerical solution by these methods faces three aspects of instability sources.

- *The direct solution of continuity equation.*

The direct solution of continuity equation is unstable and this equation is used to be replaced by pressure Poisson equation, pressure or velocity correction methods, artificial compressibility or penalty methods.

- *Advection dominant.*

At Reynolds number greater than 2, the advection term dominants the solution and observed that the solution is unstable where the variable oscillating from node to node. This problem studied early and many techniques are developed to handle the problem as Pertov Galerkin or upwinding streamline techniques.

- *Using equal order element.*

For Navier Stokes equations, the using of the same element order to the velocity and pressure leads to non physical oscillation in the pressure and in the velocity. The pressure field usually interpolated at lower order than the velocity field but equal order elements for the pressure and velocity can be used and that requires a stabilization technique.

The current work considers a numerical solution to laminar flow over a back facing step. The numerical solution adopts the penalty method instead of the continuity equation in order to reduce the computation time. The validation of the results is achieved by a comparison to numerical and experimental results.

2 Previous studies

There are a vast number of publications devoted to employ the finite element method to solve Navier Stokes equations. The applications go back to 1950's. The numerical studies usually consider the effect of the element type, the numerical scheme and the stabilization techniques. Among the early studies using finite element method, the study of Taylor and Frances 1981 in which they employed finite element method to very low Reynolds number flow over a back facing step using second order element for velocity field and less order for pressure field.

Back facing step model is preferred as test study and usually the developing of the numerical schemes is validated over this model due to the well experimental investigations.

Numerical and experimental results to flow over a back facing step is available from the work of Mateescu [3], and Barber and Fonty [2].

Mateescu presented efficient solutions of the steady and unsteady Navier-Stokes equations based on a finite difference formulation and using artificial compressibility. His numerical method is validated for steady incompressible flows past a downstream-facing step. The method is used to obtain efficient solutions for several 2D and 3D unsteady flow problems with oscillating walls.

The comparison with experimental results of Armaly and by Lee presented in [2] indicated a good agreement for Reynolds numbers between 400 and 700, with a deteriorating agreement between Re=800 and 1200; this agreement deterioration was attributed by Gartling [26], Kim and Moin presented in [2] and Armaly to the three-dimensional effects occurring in the experimental flows due to the side walls, as opposed to the rigorous two-dimensional numerical analysis.

Barber and Fonty [2] introduced a novel vortex element method for simulating incompressible laminar flow over a two-dimensional backward-facing step. Their model employs an operator-splitting technique to compute the evolution of the vorticity field downstream of abrupt changes in flow geometry. They validate the model by comparing the length of the recirculating eddy behind a confined backward-facing step against data from experimental and numerical investigations and commercial finite-volume computational fluid dynamics code, CFD-ACE+. The results show that the vortex element scheme over-predicts the length of the downstream re-circulating eddy.

The current work is devoted to employ the penalty method. This method is widely implemented to solve the Navier Stokes equations. The basic of the penalty method is relating the pressure explicitly to the velocity gradients as;

$$P = -\lambda\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right) \tag{1}$$

where λ is the penalty parameter. The magnitude of the penalty parameter can not determined easily and must be assigned a large value in order to approximate the incompressibility. Small value of penalty parameter increases the error and large value may lead to ill condition system of equations.

Recent works that conducted to the penalty method that have used different penalty parameter are surveyed next. Most of the penalty parameter are determined by numerical experiments. There are two trends in handling the penalty parameter. In the first one, penalty parameter is set as constant may depend on the fluid properties.

In the study of Vellando et al [20], they employed penalty function to study laminar flow over a back facing step, flow in square cavity and the flow past a cylinder. Numerically they used a SUPG type algorithm as a stabilization procedure, in order to eliminate the numerical oscillations. The results for the penalty algorithm for Reynolds numbers of 100, 1000, 5000 and 10000 considered, the penalty parameter as 10^4.

Another study presented in [21] employed penalty method to study the laminar flow over a back facing step at Re = 900. The penalty parameter was suggested to be in the range between 10^7 and 10^9. The study showed oscillating pressure field from one element to the next because of the discretization of the pressure.

Among the studies employed constant penalty parameter that related to the fluid density, Roylance [24], where he set the penalty parameter as,

$$\lambda = 10^7 \times \mu \tag{2}$$

The other trend in the expressing the penalty parameter is setting the parameter as function of Reynolds number as that presented by Baker [10] who suggested to use penalty parameter as;

$$\lambda = \max(\nu \mathrm{Re}, \mathrm{Re}) \tag{3}$$

and John et al [28], relates the penalty parameter to Reynolds number as follows;

$$\lambda = \frac{h.\mathrm{Re}}{\sqrt{\pi\delta}} \tag{4}$$

where h is the element width. Hozman [25] also employed the penalty method to compressible flow using penalty parameter as;

$$\lambda = \mathrm{Re} \tag{5}$$

In the manual of FIDAP, the penalty parameter is picked according to the relation,

$$\lambda = c \times D \tag{6}$$

where c depends on the machine accuracy and D is a measure of the dominant contribution in stiffness matrix. All versions of FIDAP use floating point word lengths of 64 bits and numerical studies reveal that for this range of lengths, an appropriate choice of c is 10^6 and D can be chosen as,

$$D = \max(\mu \mathrm{Re}, \mu) \tag{7}$$

when μ is the viscosity, Re is the Reynolds number, lie within a range of 10^{-3} to 10^3 it is not necessary to be very particular about the selection of λ as it may vary over several orders of magnitude with essentially insignificant effect on results; however, outside this range the penalty parameter should be decreased accordingly from its default value of 10^6.

Chan et al [23] employed a finite element and penalty method to single-phase viscous incompressible fluid, or single-phase elastic solid, as limiting cases of a biphasic material. Interface boundary conditions allow the solution of problems involving combinations of biphasic, fluid and solid regions. The results are compared to independent, analytic solution for the problem of Couette flow over rigid and deformable porous biphasic layers and show that the finite element code accurately predicts the viscous fluid flows and deformation in the porous biphasic region. The study use penalty parameter as function of fluid properties as

$$\lambda = Ct_O \frac{\beta + 2\mu^s}{N_i} + C\frac{\mu^f}{N_i} \tag{8}$$

where t_o is a reference time, C is a computer-dependent parameter related to the accuracy of the numerical calculations C is usually chosen in the range 10^7 to 10^9, β and μ^s are the solid phase Lamé parameters and μ^f mf is the viscosity of the fluid.

Generally, in practical computations, the selection of the value of the penalty parameter is of crucial importance. The following section considers a derivation of penalty parameter formula.

3 The determination of the penalty parameter

In the present paper, we assume a generalized formula to the penalty parameter based on mathematical ground where such a formula does not exist in the extensive survey. The penalty parameter is intended to be derived based on the momentum equations.

If the momentum equations are differentiated with respect to their corresponding velocity component, we obtain

$$\rho(u^* \frac{\partial N_J}{\partial x} + v^* \frac{\partial N_J}{\partial y}) - \lambda(\frac{\partial^2 N_J}{\partial x^2}) = \mu(\frac{\partial^2 N_J}{\partial x^2} + \frac{\partial^2 N_J}{\partial y^2}) \tag{9}$$

and

$$\rho(u^* \frac{\partial N_J}{\partial x} + v^* \frac{\partial N_J}{\partial y}) - \lambda(\frac{\partial^2 N_J}{\partial y^2}) = \mu(\frac{\partial^2 N_J}{\partial x^2} + \frac{\partial^2 N_J}{\partial y^2}) \tag{10}$$

Adding the two equations and obtain penalty parameter;

$$\lambda = \left| 2 \frac{\rho(u^* \frac{\partial N_J}{\partial x} + v^* \frac{\partial N_J}{\partial y})}{(\frac{\partial^2 N_J}{\partial x^2} + \frac{\partial^2 N_J}{\partial y^2})} - 2\mu \right| \tag{11}$$

The above formula gives a continuous distribution of penalty parameter over the element.

4 The element type

The usual strategy in the applications of the finite element method is using a polynomial trial functions. The pressure field is recommended to be of lower order than that used for the velocity field and should satisfy the inf-sub condition. The *usage* of such elements is summarized by Gresho et al [17]. They tabulated the advantages and disadvantages of the most known 2D-3D polynomial elements. Table 1, shows some elements that can be used when penalty method employed.

Patankar [27] used exponential trial function for triangle element as

$$N_i = A + B\exp(\frac{\rho U x}{\Gamma}) + Cy$$

where $A, B\, and\, C$ are evaluated as in the traditional way.

The approach in this paper is to use an element that simulates the mathematical solution of the governing equations. The penalty parameter in Eqn (11) requires an existence of the second derivative of the trial function, so polynomial elements are excluded from the study where higher order elements do not recommended in the Navier Stokes equations. According to the solution nature, an element with exponential distribution is suggested and used. The using of exponential distribution over triangular or rectangular elements works well with the penalty method. The pressure and velocity elements are of equal order and the trial functions of the element are,

Table 1: Working elements with penalty method [17].

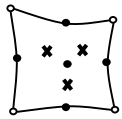
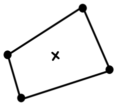
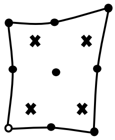
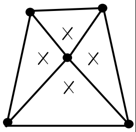
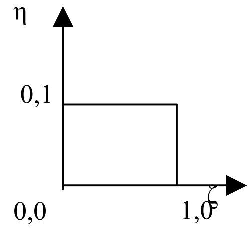
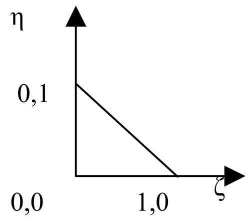
Q_2P_{-1}		Q_2Q_0	
Q_2Q_{-1}		P_1P_0	

Table 2: The trial functions of exponential elements using parent coordinates.

$N_1 = 1 - \frac{e^{\zeta}-e^{-1}}{e^1-e^{-1}} - \frac{e^{\eta}-e^{-1}}{e^1-e^{-1}} + \frac{e^{\zeta}-e^{-1}}{e^1-e^{-1}} \times \frac{e^{\eta}-e^{-1}}{e^1-e^{-1}}$ $N_2 = \frac{e^{\zeta}-e^{-1}}{e^1-e^{-1}} - \frac{e^{\zeta}-e^{-1}}{e^1-e^{-1}} \times \frac{e^{\eta}-e^{-1}}{e^1-e^{-1}}$ $N_3 = \frac{e^{\zeta}-e^{-1}}{e^1-e^{-1}} \times \frac{e^{\eta}-e^{-1}}{e^1-e^{-1}}$ $N_4 = \frac{e^{\eta}-e^{-1}}{e^1-e^{-1}} - \frac{e^{\zeta}-e^{-1}}{e^1-e^{-1}} \times \frac{e^{\eta}-e^{-1}}{e^1-e^{-1}}$	
$N_1 = 1 - \frac{e^{\zeta}-1}{e^1-1} - \frac{e^{\eta}-1}{e^1-1}$ $N_2 = \frac{e^{\zeta}-1}{e^1-1}$ $N_3 = \frac{e^{\eta}-1}{e^1-1}$	

5 Numerical scheme

It is observed from the current numerical experiments that the direct implementation and linear iterative solvers are inefficient to get precise results. In order to get a convergent solution, the following discretization is stabilized through the Pertove Galerkin method and the residual of the previous iterations is added to current iteration.

The current technique is based on varying inlet boundary condition from zero. For very small inlet boundary condition, the advection term can be neglected and the system looks like the diffusion problem. This solution can be considered as exact for the next iteration in which the inlet boundary condition is increased linearly.

$$\tau_{ps}^{n}\begin{bmatrix} \Theta_{ad}\dfrac{\rho u^{*}}{2}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial u}{\partial x}dA \\ -\Theta_{ad}\dfrac{\rho u^{*}}{2}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial v}{\partial y}dA+ \\ \Theta_{ad}\rho v^{*}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial u}{\partial y}dA+ \\ \Theta_{P}\lambda\int\dfrac{\partial N_i}{\partial x}(\dfrac{\partial u}{\partial x}+\dfrac{\partial v}{\partial y})dA+\mu\int(\dfrac{\partial N_i}{\partial x}\dfrac{\partial u}{\partial x}+\dfrac{\partial N_i}{\partial y}\dfrac{\partial u}{\partial y})dA \end{bmatrix}^{n} + \tag{12}$$

$$\sum_{jk=0}^{jk=n-1}\tau_{ps}^{jk}\begin{bmatrix} \Theta_{ad}\dfrac{\rho u^{*}}{2}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial u}{\partial x}dA \\ -\Theta_{ad}\dfrac{\rho u^{*}}{2}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial v}{\partial y}dA+ \\ \Theta_{ad}\rho v^{*}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial u}{\partial y}dA+ \\ \Theta_{P}\lambda\int\dfrac{\partial N_i}{\partial x}(\dfrac{\partial u}{\partial x}+\dfrac{\partial v}{\partial y})dA+\mu\int(\dfrac{\partial N_i}{\partial x}\dfrac{\partial u}{\partial x}+\dfrac{\partial N_i}{\partial y}\dfrac{\partial u}{\partial y})dA \end{bmatrix}^{jk} =0$$

$$\tau_{ps}^{n}\begin{bmatrix} \Theta_{ad}\rho u^{*}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial v}{\partial x}dA+ \\ -\Theta_{ad}\dfrac{\rho v^{*}}{2}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial u}{\partial x}dA \\ \Theta_{ad}\dfrac{\rho v^{*}}{2}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial v}{\partial y}dA+ \\ \Theta_{P}\lambda\int\dfrac{\partial N_i}{\partial y}(\dfrac{\partial u}{\partial x}+\dfrac{\partial v}{\partial y})dA+\mu\int(\dfrac{\partial N_i}{\partial x}\dfrac{\partial v}{\partial x}+\dfrac{\partial N_i}{\partial y}\dfrac{\partial v}{\partial y})dA \end{bmatrix}^{n} +$$

$$\sum_{jk=0}^{jk=n-1}\tau_{ps}^{jk}\begin{bmatrix} \Theta_{ad}\rho u^{*}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial v}{\partial x}dA+ \\ -\Theta_{ad}\dfrac{\rho v^{*}}{2}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial u}{\partial x}dA \\ \Theta_{ad}\dfrac{\rho v^{*}}{2}\int(N_i+\dfrac{\beta}{\|u\|}(u^{*}\dfrac{\partial N_i}{\partial x}+v^{*}\dfrac{\partial N_i}{\partial y}))\dfrac{\partial v}{\partial y}dA+ \\ \Theta_{P}\lambda\int\dfrac{\partial N_i}{\partial y}(\dfrac{\partial u}{\partial x}+\dfrac{\partial v}{\partial y})dA+\mu\int(\dfrac{\partial N_i}{\partial x}\dfrac{\partial v}{\partial x}+\dfrac{\partial N_i}{\partial y}\dfrac{\partial v}{\partial y})dA \end{bmatrix}^{jk} =0 \tag{13}$$

where n is the current iteration number and β is defined

$$\beta = \frac{\bar{\xi}.u.\Delta x + \bar{\eta}.v.\Delta y}{2}, \ \bar{\xi} = \coth(\alpha_\xi) - \frac{1}{\alpha_\xi},$$

$$\bar{\eta} = \coth(\alpha_\eta) - \frac{1}{\alpha_\eta}, \alpha_\xi = \frac{\rho.u.\Delta x}{2\mu}, \ \alpha_\eta = \frac{\rho.v.\Delta y}{2\mu}$$

and τ_{PS} is defined

$$\tau_{PS} = 1$$

where $\Theta_{\mathrm{ad}} = 2$ and this value also multiplied to the term includes fluid density in equation 11. . The factor Θ_{ad} depends on related parameters.

$$\Theta_P = \max(|J_{11}|,|J_{12}|,|J_{21}|,|J_{22}|)$$

Where J_{ij} is the element of Jacobian matrix and the relaxation factor is taken as 0.75 and the number of iterations is 4000. Another workable scheme is increasing the Θ_P linearly with the iterations.

6 Laminar flow over a back facing step

The behavior of the flow over a back facing step in laminar mode is sensitive to the Reynolds number and also sensitive to the ratio between the step height to the duct height. The main feature of such flow is the circulation that formed behind the step as a result of the abrupt change in geometry.

In flows at $ST = 0.33$ only one circulation behind the step observed while at $ST = 0.5$ another circulation occurs at the upper wall. For ST greater than 0.33, the upper circulation occurs as a result of the vacuum induced by the lower circulation and it occurs before the reattachment point of the lower circulation and extends afterwards.

Irrespective to dependency of the numerical solution on grid concentration and size, the current study is intended to be performed on uniform equal size rectangular elements to exhibit the efficiency of the developed numerical model.

The study is performed over a wide range of laminar flow and the Reynolds number ranged from 400 to 1200. The Reynolds number is based on the mean velocity and duct height as a characteristic length as in the study of Mateescu [3] but Barber and Fonty [2], calculated the Reynolds number based on the step height.

The locations of the separation and reattachment points are commonly used as a validation criteria for the computational and experimental results. Thus, the separation and the reattachment points for laminar flow is tabulated versus Reynolds number in table 1 and the table also shows a comparison to the data given in [2] and in [3].

Table 3 shows a change of lower reattachment point with change of Reynolds number that presented in [2] and [3].

Kim and Moin's study that presented in [2] used a computational domain that does not include an inlet region and they directly impose the inflow boundary condition at the step edge itself. In this study and the study of Mateescu [3], the domain contains an inlet region which has made the problem more physically realistic.

The linear iterative solvers are inefficient to solve the system of equations so that gauss elimination method is applied to compressed matrices in each iteration.

The numerical solution is very sensitive to Θ_p and relaxation factor. Their changes have great effect on the numerical prediction of the reattachment point. Although of that, the comparison of the current results shows the efficiency of the current numerical solution. The current results are more accurate than the numerical results presented in both [2] and [3]. The consistency of accuracy is hold over all tested range of Reynolds number. But for the study of [2] and [3] the accuracy is lost for Reynolds number greater than 400.

For real flow, a secondary bubble circulation is observed experimentally behind and closer to the step and the current numerical technique is capable to predict this bubble circulation.

7 Conclusions

In this study, a new formula of penalty method is derived from momentum equations and used with an element of exponential trial function to study a laminar flow over a back facing step. A special technique to handle the problem is employed hence linear iterative solvers are found inefficient. Validation of the code is achieved through a comparison over a wide range of Reynolds number of flow over a back facing step in laminar mode. The comparison shows that a precise results are obtainable if the multipliers to the advection and penalty parameter are well employed. The coefficients, θ_{ad} , θ_p needs a mathematical optimization and generally the present numerical technique shows efficiency over the numerical results presented in the references.

Table 3: The effect of Reynolds number and step height at ST =0.5.

Re_D	Lower Reattachment point: R1			upper separation point: R2			upper Reattachment point: R3		
	Curr	Ref [2]	EXP [3]	Curr	Ref [2]	EXP [3]	Curr	Ref [2]	EXP [3]
400	4.6	5.5	4.1-4.3	-	-	-	-	-	-
600	5.6	7	5.21-5.8	4.3	-	4	7	-	8
800	7	8.5	6.45-7.1	5.2	-	5-5.25	10	-	9.8-10
1200	9.1	11	8.4-8.9	4.7	-	7.8	9.8	-	11-11.5

Curr: Current results, **Exp [3]**: Experimental results presented in [3].
Re_D : Reynolds number based on the hydraulic diameter.

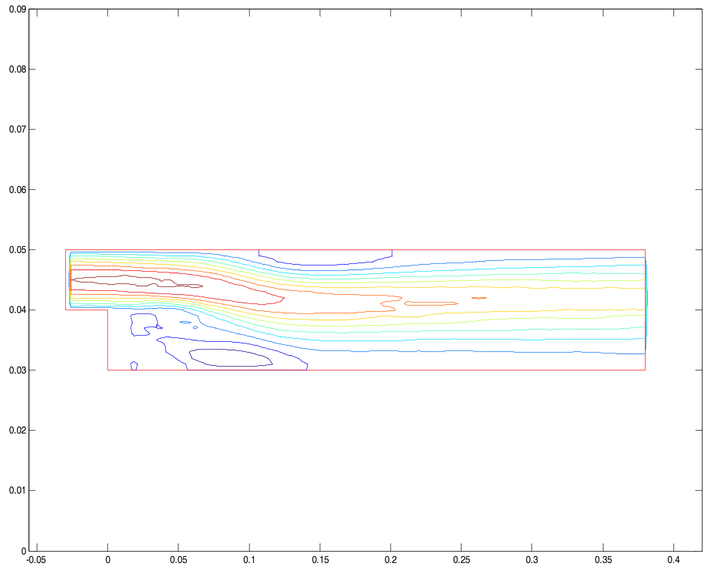


Figure 1: Backward facing step problem for Re_D =800.

Nomenclature

ρ : Fluid density.
μ : Fluid dynamic viscosity.
ν : Fluid kinematics viscosity.
λ : Penalty parameter.
Re : Reynolds number.
δ : Boundary layer thickness.
h : Element size.
N_I : Weighting function.
N_J : Trial function at node J .
ST : The ratio between the step height to the duct height.

References

[1] Mahmud A. Mehemed Abughalia, *Numerical study of reattachment point to flow over Back Facing Step*, M. Sc. Thesis, AL-Fateh University, Libya,2007.

[2] Robert W. Barber and Anne Fonty, *Comparison of vortex-element and finite-volume simulations of low reynolds umber flow over a confined backward-facing step*, 4th GRACM Congress on Computational Mechanics, GRACM 2002,Patra, 27-29 June, 2002.

[3] Dan Mateescu, *Efficient methods for 2D and 3D steady and unsteady confined flows based on the solution of the Navier-Stokes equations*, aerospace program director, Department Of Mechanical Engineering, Mcgill University, Montreal, Canada, email: Dan.Mateescu@mcgill.ca, 2004.

[4] J.G. Barbosa Saldana and N. K. Anand, *Forced convection over a three-dimensional horizontal backward facing step*, Department Of Mechanical Engineering, Department Of Computer Science, Texas University.

[5] H. M. Blackburn, D. Barkley and S. J. Sherwin, *Convective instability and transient growth in flow over a backward-facing step*, Department Of Mechanical Engineering, Monash University, australia, mathematics institute, University of warwick, UK and Department Of Aeronautics, Imperial College London, UK.

[6] E. Creus´e, a. Giovannini, and I. Mortazavi, *Active control strategies for transitional backward-facing step flows*, lamav, universit´e de valenciennes, france equipe simpaf, inria futurs, site de lille ecreuse@univ-valenciennes.fr 2 imft, 118, route de narbonne, 31062 toulouse, france giova@cict.fr 3 universit´e de bordeaux, 33405 talence cedex, france projet mc2, inria futurs, site de bordeaux mortaz@math.u-bordeaux1.fr.

[7] T. M. Shih, numerical heat transfer, hemisphere, 1984.

[8] Gianni Comini, Stefano Del Giudice, Carlo Nonino, *finite element analysis heat transfe*r, Taylor & Francis, 1994.

[9] Suhas V. Patanker, *Numerical heat transfer and fluid flow*, Taylor & Francis, 1980.

[10] A. J. Baker, *Finite element computational fluid mechanics*, Hemisphere, 1983.

[11] Klaus Jurgen Bathe, F*inite element procedure*, Prentice Hall, 1996.

[12] Roald Akberov, Darrell W. Pepper, Yitung Chen, *Turbulent flow over a backward facing step using penalty and equal order methods*, Nevada Center For Advanced Computational Methods.

[13] A. Kolesnikov and A. J. Baker, *Efficient implementation of high order methods for advection diffusion equation*, PROC of the third ASME-JSME Joint Fluids Engineering Conference, July 18-23 1999, San Francisco, California.

[14] T. Chen and L. S. Raju, *Coupling finite element and meshless local petrov- galerkin methods for two dimensional problems*, AIAA-2002 - 1659,2002.

[15] Jahrul Alam and Md. Ashaque Meah*, Finite element simulation of two dimensional fluid flow*.

[16] S. A. Mohsen Karimian, Anthony G. Straaman, *An improved scheme for pressure velocity coupling in control volume based finite element discretization*, the University of Western Ontario, London, Ontario, Canada n6a 5b9.

[17] P. M. Gresho, R. L. Sani and M. S. Engelman, *incompressible flow and the finite element method*, John Wiley & Sons, 1998.

[18] Rajiv Sampath and Nicholas Zabaras, D*esign and object-oriented implementation of a preconditioned-stabilized incompressible Navier Stokes solver using equal-order-interpolation velocity pressure elements*, Cornell University.

[19] B. S. Varaprasad Patnaik et al, *simulation of internal flows with heat transfer using velocity correction*.

[20] P. Vellando, J. Puertas, j. Bonillo, J. Fe. *On the resolution of the viscous incompressible flow for various supg finite element formulations*, European Congress on Computational Methods in applied sciences and engineering. Eccomas 2000 Barcelona, 11-14 September 2000. Ó ECCOMAS.

[21] Numerical Methods, *Finite element for modeling fluid flow*, section 410.4,fluid flow division, march 1999. Genium Publishing.

[22] R.D. Lazarov, J.E. Pasciak, J. Sch¨oberl, and P.S. Vassilevski, *Almost optimal interior penalty discontinuous approximations of symmetric elliptic problems on non-matching grids*, 1991 Mathematics Subject classification. 65f10, 65n20, 65n30. Science foundation under grant dms-9973328. 2000- 2001.

[23] B. Chan, P. S. Donzelli and R. L. Spilker, *A mixed-penalty biphasic finite element formulation incorporating viscous fluids and material interfaces*, Department Of Mechanical Engineering, Aeronautical Engineering, and Mechanics Department of biomedical engineering, Scientific computation Research center 9 Rensselaer Polytechnic Institute 10 Troy, New York 12180.

[24] David Roylance, *Finite element analysis of non-isothermal reactive flows*, Department Of Materials Science and Engineering, Massachusetts Institute of Technology, Cambridge, 2003.

[25] J. Hozman, *Numerical solution of the Navier–Stokes equations by semi–implicit schemes*, Charles University, Faculty of Mathematics and Physics, Prague, Czech Republic. Wds'06 PROC of contributed papers, part i, pp 59–64, 2006.

[26] Gartling D.K. *A test problem for outflow boundary conditions – Flow over a backward-facing step*. International Journal for Numerical Methods in Fluids 11: 953-967, 1990.

[27] Suhas V. Patanker, *Numerical heat transfer and fluid flow*, Hemisphere Publishing corporation, 1980.

[28] V. John, W. Layton and N. Sahin, *Derivation and analysis of near wall models for channel and re-circulating flows*, Faculty of Mathematics Ottovon-Guericke- University Magdeburg, Germany, 0898-1221/04/ - 2004 Elsevier ltd.