II CONGRESSO NACIONAL DE ENGENHARIA MECÂNICA
II NATIONAL CONGRESS OF MECHANICAL ENGINEERING
12 a 16 de Agosto de 2002 - João Pessoa – PB

THE NO-SLIP BOUNDARY CONDITION IN THE STREAM FUNCTION-VORTICITY FORMULATION

Fábio Capelassi Gavazzi De Marco
demarco@mec.ita.br
Cláudia Regina de Andrade
claudia@mec.ita.br
Edson Luiz Zaparoli
zaparoli@mec.ita.br
Instituto Tecnológico de Aeronáutica – Departamento de Energia – 12228-900 – São José dos Campos, SP, Brasil.

Abstract. *This work reports an analysis of penalty method application to imposing no-slip boundary condition. The approach used herein consists to express the vorticity boundary condition through the natural boundary condition depending on the solid wall tangential velocity component. The penalty parameter is introduced in this boundary condition and its influence in the numerical solution is analyzed. Two numerical examples are considered, lid driven cavity flow and natural convection both in a square cavity. The results show the behavior of numerical solution taking into account the mesh density necessary to achieve an error limit in the mesh refinements context, the accuracy of solution when the penalty parameter is varied. The results shown that as long the penalty parameter tends to infinity the accuracy is improved but the solution process become slower.*

Key-words: *boundary condition, penalty term and no-slip.*

1. INTRODUCTION

One of the most popular form used in CFD for writing the Navier-Stokes equations is the stream- function vorticity formulation, examples are Manoj and Sengupta (1996), Anagnostopoulos et al (1996); Barragafy and Carey (1996). However, a computational difficulty, which arises in solving the coupled stream-function vorticity equations, is the vorticity conditions at the walls. The reason is that each equation involves a Laplacian operator and there are two boundary conditions associated with stream function while none associated with vorticity.

There are many works in finite difference and finite elements techniques to overcome the velocity no-slip boundary condition problem in the stream-function vorticity approach. A method that relates the vorticity at the boundary to the normal derivate of the stream function was presented by Glowinski and Pironneau (1979). Huang and Seymour (1996) had described an approach that involves decoupling the boundary vorticity from the computation of the interior flow field. Souli (1996) describes a vorticity boundary condition based on the Green's formula and Green's function for Laplace operator. Gresho and Shany (1998) suggest the application of natural boundary conditions to vorticity at all frontiers of computational domain. Napolitano et al (1999) had presented a review of vorticity boundary conditions in CFD, discussing coupled and uncoupled formulations of the problem as well as steady state and unsteady equations.

According to Layton (1999), there are at least three natural ways of imposing zero tangential velocities value along the boundary:

(i) Lagrange multiplier of tangential velocity component equal to zero as a constraint;
(ii) Penalty term imposing tangential velocity component equal to zero approximately;
(iii) Replacing "no-slip" with "slip with friction".

The purpose of this work is to present and analyses a technique to impose the tangential velocity component along the surface by penalization of the vorticity boundary condition. Such method consists to obtain the wall vorticity throughout the stream function values, similar approach was used in Chouikh et al (1999), Manoj and Sengupta (1996), Guo et al (1998) and shown to be adequate to simulate problems of flow around cylinders. This approach can be used not only to specify no-slip boundary condition but also to specify moving wall boundary condition.

The Poisson equation for stream function, the vorticity transport equation, and the energy equation were solved simultaneously to obtain a numeric solution of the mathematical model for steady state incompressible fluid flow, without the direct use of the continuity equation. The equations were solved using iterative-coupled algorithm. The discretization of stream function, vorticity and temperature equations was done independently with equal order interpolation functions (2^{th} order).

The resulting system of non-linear differential equations was solved by finite element method using adaptative refinement and unstructured mesh with six nodes triangular elements. The Newton-Raphson method (Heinrich and Pepper, 1999) is applied to linearize the algebraic systems of non-linear equations resultants of the discretization and solved in each iterative step through the conjugated gradient method (Axelsson, 1996).

2. MATHEMATICAL FORMULATION

2.1. Conservation Equations

The two-dimensional steady-state, laminar, incompressible constant properties fluid flow mathematical model consist of stream function (ψ)-vorticity (ζ) and energy equations. The vorticity transport equation is:

$$\vec{V}.\vec{\nabla}\zeta = \mu\vec{\nabla}^2\zeta + \vec{f} \tag{1}$$

where the stream function Poisson equation is:

$$\vec{\nabla}^2\psi = -\zeta \tag{2}$$

The energy equation is:

$$\alpha\nabla^2 T = (\vec{V}.\vec{\nabla})T \tag{3}$$

where the stream function and vorticity are given by:

$$\vec{V} = \vec{\nabla}\times\psi \tag{4a}$$

$$\zeta = \vec{\nabla}\times\vec{V} \tag{4b}$$

and T is the temperature, $\vec{f}$ is the body force term, $\vec{\nabla}\cdot(\ )$ is the divergent operator, $\vec{\nabla}(\ )$ is the gradient operator, $\vec{\nabla}\times(\ )$ is the curl operator, μ is the viscosity, α is the thermal diffusivity, and $\vec{V}$ is the velocity vector with u and v components.

The curl operator results a vector with potentially three components; the meaning of two dimensional curl operator must take into account a couple of possible situations:

(1) The curl operator has only one component perpendicular to the plane of computation ($\zeta = \vec{\nabla} \times \vec{V}$);

(2) The curl operator is a two-components vector in the plane of computation ($\vec{V} = \vec{\nabla} \times \psi$).

2.2. Boundary conditions

It was assumed the expression given by Eq. (5) to vorticity boundary condition at moving wall and the Eq. (6) to vorticity boundary condition in the wall where there is zero velocity in the wall. This approach consist to employ natural boundary to vorticity as:

$$\frac{\partial \zeta}{\partial n} = \phi \left[\vec{\nabla} \times \psi + \vec{V}_w\right] \cdot \vec{t} \tag{5}$$

$$\frac{\partial \zeta}{\partial n} = \phi \left[\vec{\nabla} \times \psi\right] \cdot \vec{t} \tag{6}$$

where: ϕ is the penalty parameter, $\vec{t}$ represents the oriented tangential unit vector, $\vec{V}_w$ is the velocity boundary condition and $\partial(\ )/\partial n$ is the boundary outward normal derivative.

According to the penalty method, when the penalty parameter increases the vorticity normal derivative should adjust to impose $\left[\vec{\nabla} \times \psi + \vec{V}_w\right] \cdot \vec{t} = 0$.

3. NUMERICAL EXAMPLES

In this section, two examples are presented to demonstrate and evaluate the application of penalty term in the vorticity boundary condition. The first example considers forced convection and the second one analyzes a natural convection problem both in a square cavity.

3.1. Lid Driven Cavity

The lid driven cavity problem has been used in the literature as an incompressible flow benchmark to test and evaluate numerical methods. This problem was included here to show the behavior of velocity field and vorticity values take into account the degree of boundary condition penalization. The Reynolds Number is defined as Re = $\rho . u_w . L / \mu$, where u_w is the velocity of the moving wall, L is the length of the side and ρ is the density.

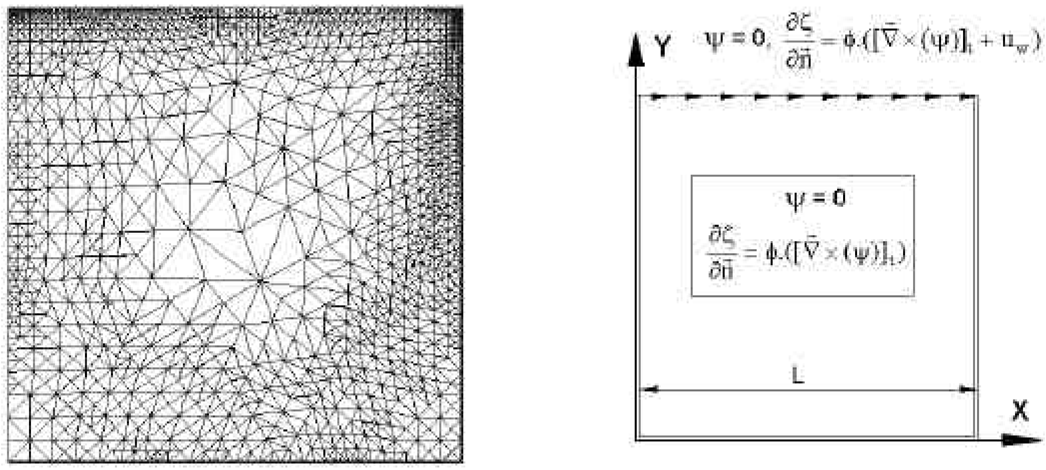

Figure 1. Mesh at Reynolds number 1000 and boundary conditions.

The simulations were performed at Reynolds number 1000 varying the penalty parameter (ϕ) from 1 to 5000. In the Fig. 1 there is an example of mesh (6572 nodes, 3153 cells) employed to achieve an error less than 1.e-5 with $\phi = 1000$.

Figure (2) gives an indication of the way that the velocities meet the correct value when the penalty parameter increases. In the $\overline{AB}$ boundary segment the velocity value must be equal unit and are null anywhere. The case that $\phi = 1$, the penalization is weak and it is clear that the no-slip condition is not satisfied. The progressive increase of velocity magnitude in the segment $\overline{AB}$ and decrease in $\overline{CD}$, $\overline{DA}$ and $\overline{BC}$ is achieved with increase of ϕ, although with high ϕ values there is still a residual in the left top corner ($\overline{DA}$ segment).

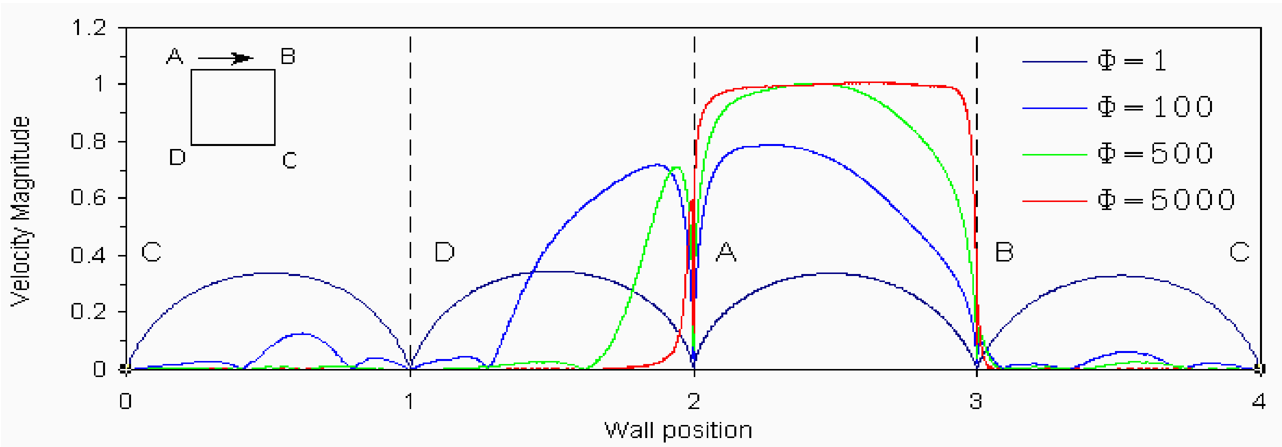

Figure 2. Influence of penalty parameter in velocity magnitude along wall at Reynolds 1000.

The velocity components u and v, in the horizontal and vertical cavity centerline, respectively, are shown in the Fig. (3). The results of Guia et al (1982) to lid driven cavity flow were assumed as benchmark solution to check the present results at Reynolds number equal to 1000. There is no visible differences between the velocity profiles when the penalty parameter is set to $\phi = 500$ or to $\phi = 5000$. Furthermore they are in good agreement with benchmark results, although in the Fig. 3 is clear that $\phi = 500$ does not provide corrects velocity values at the boundary.

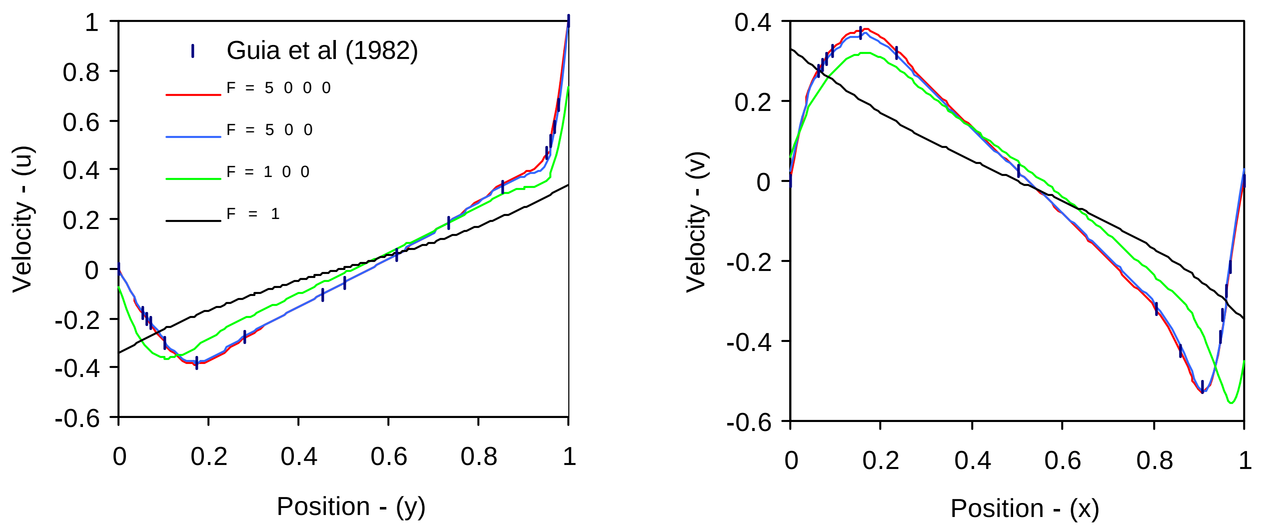

Figure 3. Velocity profile at the cavity center.

Figure (4) shows the comparison of present results to vorticity with literature benchmark results for the moving wall. This data confirm that, only to high values of ϕ, the solution to furnish accurate values. When ϕ = 1, practically there is no vorticity in the wall and the slipping at walls is visible.

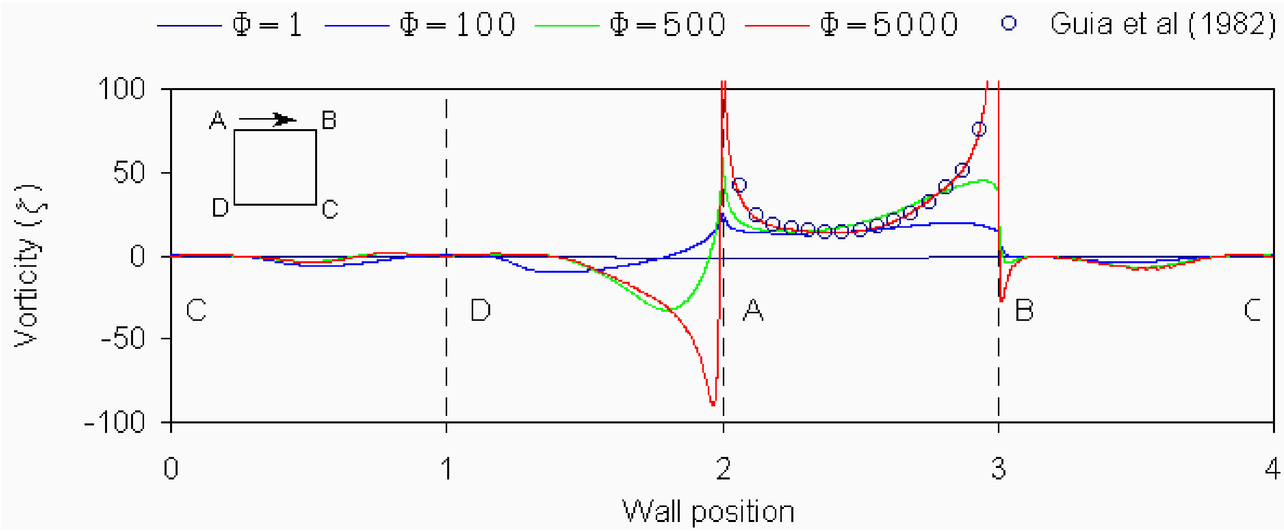

Figure 4. Influence of penalty parameter in vorticity value along wall at Reynolds 1000.

3.2. Natural Convection in a Square Cavity

A natural convection problem is considered in this section. The vertical walls are kept to be isothermal at T_h and T_c. The top and bottom are kept to be adiabatic. In Fig. (5) there is an example of the finest mesh (11,843 nodes, 6,584 cells) employed in this simulations to achieve an error less than 1.e-5 with ϕ = 1000.

It was assumed in this work the Boussinesq approach, them the density in the body force term of the vorticity transport equation varies linearly with temperature as $\rho = \rho_0[1-\beta(T-T_o)]$ with ρ_o equal to density at average temperature. The Boussinesq approach results only depends on the Rayleigh number and Prandtl number defined as:

$$Ra = \frac{\beta.g.(T_h - T_c).L^3}{\alpha.\nu} \tag{7}$$

and,

$$Pr = \frac{\mu.C_p}{k} \tag{8}$$

where: g is the gravity acceleration, L is the length of the cavity, the coefficient of volumetric expansion is $\beta = 1/T_o$ and T_o is the vertical walls average temperature, $\nu = \mu / \rho$ is the viscosity, ρ is the density, C_p is the specific heat and k is the thermal conductivity.

The simulations were carried out to air flow (Pr=0.71) in the range $1.10^3 \leq Ra \leq 1.10^6$. The penalty parameter in this problem varies from 1 to 1000.

In the present work the average Nusselt number is defined on the vertical mid-plane of the cavity and is given by:

$$Nu = \frac{-\int_0^L \frac{\partial T}{\partial x} dy}{(T_h - T_c)} \qquad (11)$$

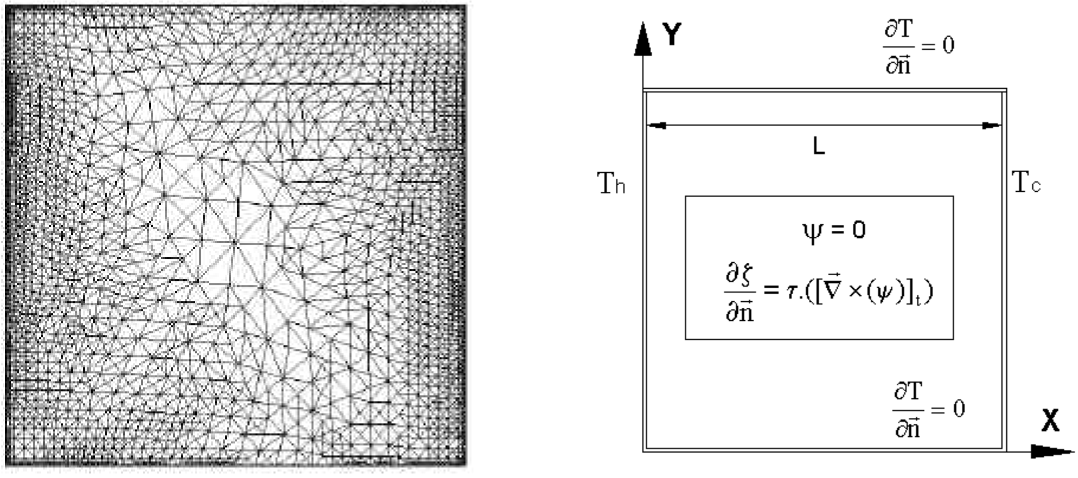

Figure 5. Mesh at Ra = 1.10^6 with ϕ = 1000 and boundary conditions.

Table (1) summarizes the present work results to natural convection and shows a comparison with numerical results provided by Davis (1983). The symbol $I(u,v)_D$ represents the magnitude of velocity integral along the cavity boundary. For each Rayleigh number and ϕ there is a correspondent indication of mesh refinement needful to obtain the numerical solution with error limit equal to 1.10^{-5}. It could be noted that the increase in the mesh nodes number come together with the increase in the ϕ parameter, fixing the Ra number. Such fact is related with the control error in the context of adaptive mesh refinement.

Table 1. Comparison of present results with literature for natural convection in a square cavity.

Ra		Present work				Davis, V.G., (1983)	Differences (%)
		ϕ = 1	ϕ = 10	ϕ = 100	ϕ = 1000		
1.10^3	Nu	1.213	1.119	1.116	1.117	1.117	0
	u_{max}	4.447	3.647	3.628	3.652	3.634	-0.50
	v_{max}	4.567	3.783	3.749	3.761	3.667	-2.56
	Nodes	313	461	696	1677	-	-
1.10^4	Nu	2.434	2.248	2.237	2.241	2.235	-0.27
	u_{max}	17.007	16.117	16.072	16.076	16.182	0.65
	v_{max}	20.698	19.480	19.439	19.495	19.509	0.07
	Nodes	1153	1403	1917	2912	-	-
1.10^5	Nu	4.780	4.516	4.508	4.513	4.512	-0.02
	u_{max}	36.019	34.626	34.557	34.626	34.81	0.53
	v_{max}	70.970	67.582	67.489	67.582	68.22	0.94
	Nodes	3191	3391	4455	5627	-	-
1.10^6	Nu	9.167	8.813	8.803	8.807	8.816	0.10
	u_{max}	66.6	64.4	64.37	64.41	65.33	1.41
	v_{max}	224.3	217.3	217.2	217.8	216.75	-0.48
	Nodes	7695	8233	10 335	11843	-	-
	$I(u,v)_D$	0.1	0.01	0.002	0.0005	-	-

As the ϕ parameter is increased, the relative error associated with vorticity near the boundary domain is amplified; consequently there is need for a finest mesh to comply with error limit requirements. This implies in a decrease in the convergence rate and more computational time effort, however the numerical solution accuracy is improved as shown in Tab. (1)

The maximum velocity values and the average Nusselt number were compared with Davis V.G. (1983). The differences column presented in Tab. (1) was obtained with $\phi = 1000$. Very small differences were found, mainly for the Nusselt number values.

4. CONCLUSIONS

At the present study the penalty technique was applied to impose vorticity boundary conditions at no-slip wall. This approach is an approximate method to establish the actual boundary condition but provides suitable results when the values at the boundary are checked.

Results obtained in this work showed that the boundary condition specified error decreases as the penalty parameter increases but this is accomplished with a computational effort penalization.

5. ACKNOWLEDGES

Authors are grateful to Capes, which supported this work.

6. REFERENCES

Anagnostopoulos, P., Iliadis, G. and Richardson, S., 1996, "Numerical Study of the Blockage Effects on Viscous Flow Past a Circular Cylinder", International Journal for Numerical Methods in Fluids, Vol. 22, pp. 1061-1074.

Axelsson, O., 1996, "*Iterative Solution Methods*", 654 p., Cambridge University Press, New York.

Barragy, E. and Carey, G.F., 1996, "Stream Function-Vorticity Driven Cavity Flow Solution Using *p* Finite Elements", Computer & Fluids, Vol. 26, No.5, pp. 453-468.

Chouikh, R., Guizani, A. and Maâlej, M., 1999, "Numerical Study of the Laminar Natural Convection Flow Around an Array of Two Horizontal Isothermal Cylinders", Int. Comm. Heat Mass Transfer, Vol. 26, No. 3, pp. 329-338.

Davis, G.V., 1983, "Natural Convection of Air in a Square Cavity: A Bench Mark Numerical Solution", International Journal for Numerical Methods in Fluids, Vol. 3, pp. 249-264.

Glowinski, R. and Pironneau, O., 1979, "Numerical Methods for the Biharmonic Equation and for the Two Dimensional Stokes Problem", SIAM Rev. 21(2).

Gresho, P.M. and Sani, R.L., 1999, "Incompressible Flow and the Finite Element Method, Advection-Diffusion and Isothermal Laminar Flow", John Wiley and Sons, England, 1021p.

Guia, U., Guia, K.N. and Shin, C.T., 1982, "High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method", Journal of Computational Physics, Vol. 48, pp. 387-411.

Guo, T., Chew Y.T., Luo, S.C. and Su, M.D., 1998, "A New Numerical Simulation Method of High Reynolds Number Flow Around a Cylinder" Comput. Methods Appl. Mech. Engrg., vol. 158, pp. 357-366.

Heinrich, J.C. and Pepper, D.W., 1999, "Intermediate Finite Element Method", 596 p., Taylor & Francis, Londres.

Huang, H., and Seymour, B.R., 1996, "The No-Slip Boundary Condition in the Finite Diference Approximations", International Journal for Numerical Methods in Fluids, Vol. 22, pp. 713-729.

Layton, W., 1999, "Weak Imposition of "No-Slip" Conditions in Finite Element Methods", Computer and Mathematics with Applications, Vol. 38, pp. 129-142.

Manoj, T.N. and Sengupta, T.K., 1996, "Onset of Asymmetry: Flow Past Circular and Elliptic Cylinders" Int. Journal for Numerical Methods in Fluids, Vol.23, pp. 1327-1345.

Napolitano, M., Pascazio, G., Quartapelle, L., 1999, "A Review of Vorticity Conditions in the

Numerical Solution of the ζ-ψ Equations”, Computer & Fluids, Vol. 28, pp. 139-185.
Souli, M., 1996, “Vorticity Boundary Conditions for the Navier-Stokes Equations”, Comput. Methods Appl. Mech. Engrg., Vol. 134, pp. 311-323.