24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-2592
NUMERICAL BEHAVIOR OF THE DISCONTINUOUS GALERKIN METHOD IN INCOMPRESSIBLE FLUID FLOW SIMULATIONS FOR LOW REYNOLDS NUMBER

Fernando José da Silva[1]
[1]Universidade Tecnológica Federal do Paraná, Graduate Program in Civil Engineering, Pato Branco, Brazil
[1]fernando_j.s@hotmail.com;
Francisco Augusto Aparecido Gomes[2]
[2]Universidade Tecnológica Federal do Paraná, Department of Mechanical Engineering, Pato Branco, Brazil
[2]francisco@utfpr.edu.br;
Paulo Rogério Novak[3]
[3]Universidade Tecnológica Federal do Paraná, Department of Mechanical Engineering, Pato Branco, Brazil
[3]novak@utfpr.edu.br
Áureo Quincas Garcia[4]
[4]Universidade Tecnológica Federal do Paraná, Department of Mechanical Engineering, Pato Branco, Brazil
[4]aureo@utfpr.edu.br
Mildred Balin Hecke[5]
[5]Universidade Federal do Paraná, Graduate Program in Numerical Methods in Engineering, Curitiba, Brazil
[5]mildredhecke@gmail.com

Abstract. *In this work, we demonstrate the numerical solution using the Galerking high order discontinued method to simulate the flow around a cylindrical channel and a Naca 0012 section using a low Reynolds number. These problems consist in a laminar viscous flow of incompressible fluid through a rectangular channel. The basic equations that represent the physical model are characterized by Navier-Stokes equation for viscous and incompressible fluids. These equations are very useful to represent numerical solutions for predominantly convective problems in engineering applications. For the flow around de cylinder, it was implemented an adequate output limit when the flow properties are detailed, this directional condition is known as do-nothing. This limit condition class can be applied with truncated domains, reducing the computational cost of the simulation, which is very compelling when high order methods are applied because their degree of freedom are high when compared to other discretization method. To study if this type of limit condition affects the numerical behavior of the solution, the convergence order and the accuracy of the solution must be investigated. To do that, some test cases will be described and both the numerical and physical aspects will be discussed. The comparisons and discussions will be made based on classical literature references.*

Key Words: High-order method, Discontinuous Galerkin, Incompressible fluid, Truncated domain, Low Reynolds number.

1. INTRODUCTION

Engineering problems are usually associated to solutions inside the complex domain that represent the geometry where the physical behavior needs to be analyzed. Since the experimental solution is often time consuming, the approximation methods are very useful to describe a solution. The most common approximation methods applied to engineering problems are: Finite difference method (FDM), Finite Volume Method (FVM) and Finite elements method (FEM). The broadest and more flexible method, among the three methods mentioned, to study complex geometries is the FEM. However, in fluid dynamic problems the flow may result in severe gradients ,where the classic FEM approach may fail. One of these disadvantages example can be noticed in problems where convection is predominant (J. N. Reddy and D. K. Gartling, 2010). To overcome this disadvantage of FEM, a new Galerkin method approach comes to the scene: The discontinued Galerkin Method (DGM) (J. S. Hesthaven and T. Warburton, 2008).

This paper aims to present the results regarding to the DGM numerical behavior when high order polynemes are applied to achieve better simulation performance related to flow around both a cylinder and a NACA 0012 section. The cylinder as well as the NACA 0012 are inserted in rectangular sections.

2. COMPUTATIONAL PROCEDURE

This paper's case is based on fluids flow simulations around an aerodynamic section NACA 0012 and a cylindrical channel. It was considered that Ω is two-dimensional, where ∂Ω = ∂ΩD U ∂ΩN is a disjunctive union, ∂ΩD represents the limit where the primitive variables are prescribed and ∂ΩN is the complementary limit where the primitive variables are known. The equations that describe the problem are the Navier-Stokes for incompressible fluids, considering continuity an also, Newtonian fluid with no external forces. The full equation set in [x, t] × Ω is represented by Eq.(1) e Eq.(2).

$$\frac{\partial u}{\partial t} + (u.\nabla)u = -\nabla p + v\nabla^2 u, \qquad \text{in } \Omega, \tag{1}$$

$$\nabla.u = 0, \qquad \text{in } \Omega, \tag{2}$$

where x is the position vector in Cartesians coordinates, u is the velocity vector, p is pressure, t is time e v cinematic viscosity, where $v = u/p$.

3. RESULTS AND DISCUSSION

3.1 Channel-cylinder flow

The first result analyzed in this paper is the cylindrical channel flow. The objective of this simulation is to show the results accuracy obtained by the DGM with the directional boundary condition. The computational domain with its limit conditions for this simulation is shown in Fig. 1, D= 0.1m, H= 0.41m and homogeneous boundary Dirichlet conditions.

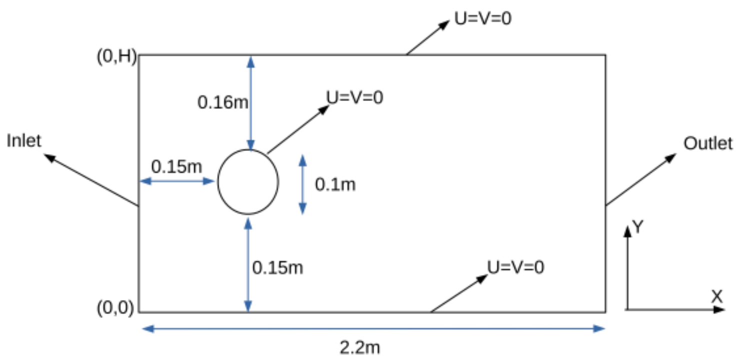

Figure 1. Computational domain.

In the input boundary, there is the following equation Eq. (3).

$$u(0; y; t) = \frac{4U_m y(H\ y)}{H^2} \quad ; \qquad v(0; y; t) = 0; \tag{3}$$

Where $Um = 1.5m/s$. In that case Re=100. The output limit condition is developed by Dong et al. (2014) and explained by Garcia et al (2016). It can be observed the computational grids applied to this test, emphasizing the reduced amount of elements in the grid with the truncated domain presented in Fig.3 comparing to the extended grid, Fig. 2 .

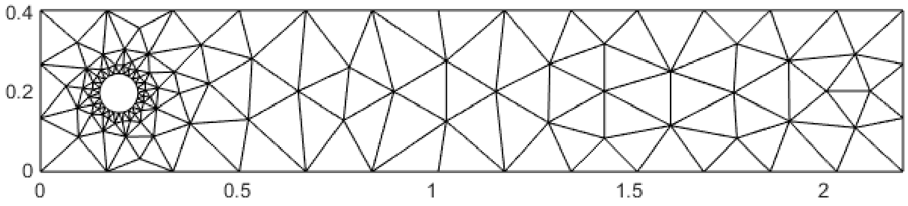

Figure 2. Discretized Domain in 206 elements, xmax=2.2.

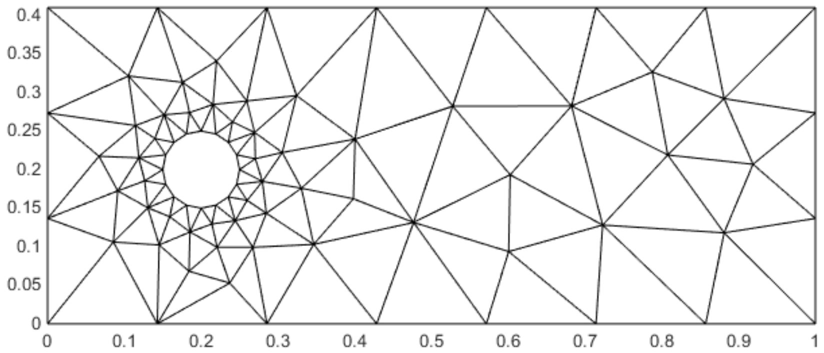

Figure 3: Truncated Domain with 130 elements, xmax=1.0.

It is shown in Fig.4 and Fig.5 the velocity isolines in x direction to the complete and truncated domain, respectively. Considering Reynolds=100.

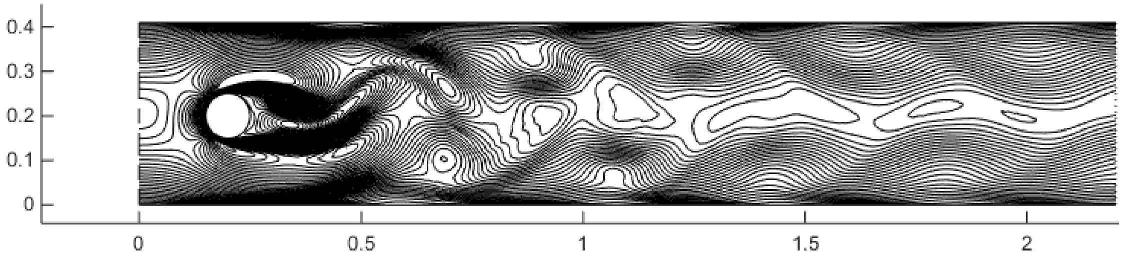

Figure 4. Isolines for the complete domain.

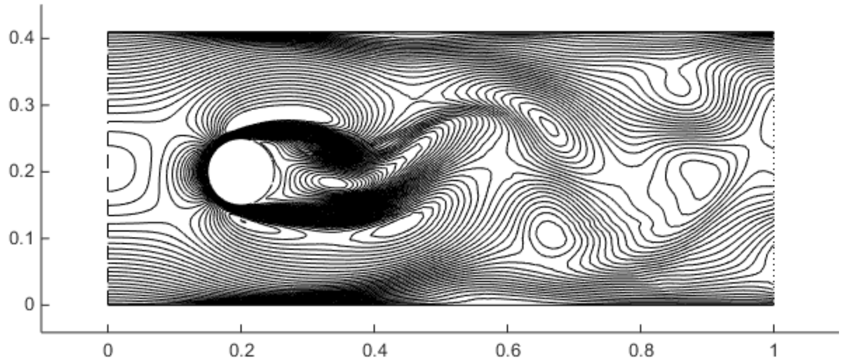

Figure 5. Isolines for the truncated domain.

The velocity isolines as shown in the figures presented above, indicate that the condition of the direction provides a very accurate result in the truncated domain solution compares to the solution of the extended domain. This limit condition can replicate accurate results by using a reduced dimensions domain.

Shafer e Turek (1996), based on several numerical experiments performed by other authors, established maximum values for the drag and lift coefficients. Tab. 1 shows the results obtained by DGM and Tab. 2 presents a comparison with the results obtained by Shafer and Turek (1996).

Table 1: N: element interpolation order; Nel: total number of elements; Tf: final time of time loop (*) our best results for xmax=2.2.

N	Nel	Xmax	Tf	High order boundary	Cd max	Cl max	St
7	130	1	8	yes	3,2320	1,0142	0,3000
7*	206	2.2	8	yes	3,2345	1,0087	0,3000
8	130	1	8	yes	3,2235	1,0300	0,3000
8	206	2.2	8	yes	3,2200	1,0200	0,3000

Table 2: Comparison between the computed maximum values intervals, from Schäfer and Turek (1996), and the numerical maximum values obtained by DG code in the present work.

Compared	Higher Maximum			Lower Maximum		
Methods	Cl	Cd	St	Cl	Cd	St
Shäfer & Turek	1.0100	3.2400	0.3050	0.9900	3.2200	0.2950
(DG)	Cl=1.0087		Cd=3.2345		St=0.3000	

The results are very close to the ones obtained by Schäfer e Turek (1996), Tab. 2. The most accurate result was obtained for Xmax = 2.2 and seventh order polynome (N=7). For Xmax = 1.0 the best result was achieved for N=7 and 130 elements. It can be noticed that the results obtained for the truncated domain are pretty close to the ones obtained with the extended domain.

3.2 Naca 0012

Given the widely usage of airfoils, in the aeronautics and automotive industries for example, and considering that its performance affects the machines associated with it, it is intended to evaluate the DGM potential to simulate the behavior of parameters such as velocity and pressure through the airfoil. A NACA 0012 airfoil was adopted and the pressure coefficient (Cp) was calculated for $\alpha = 0°$ e *Re*=100. The results were compared to the literature values. The Naca 0012 section can be verified in Fig. 6.

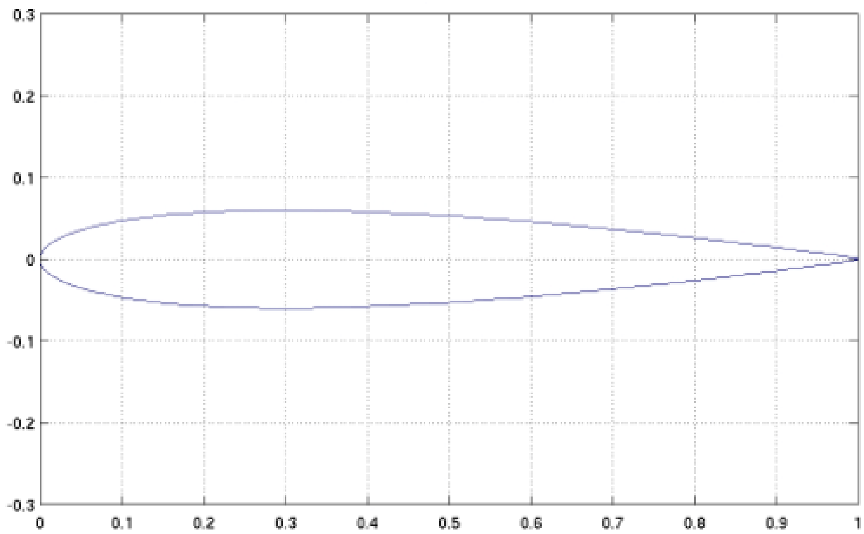

Figure 6. Perfil Naca 0012.

The computational domain is demonstrated in Fig. 7 among the simulation boundary conditions. The input velocity, Ux is constant and the computational domain is big enough to eliminate the wall effects over the results.

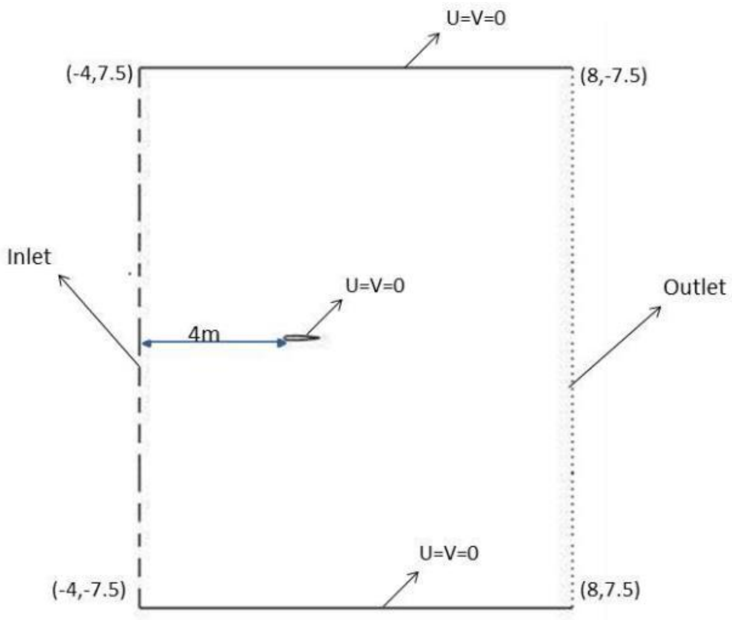

Figure 7. Dominio computacional.

Fig.8 show the discretized domain.

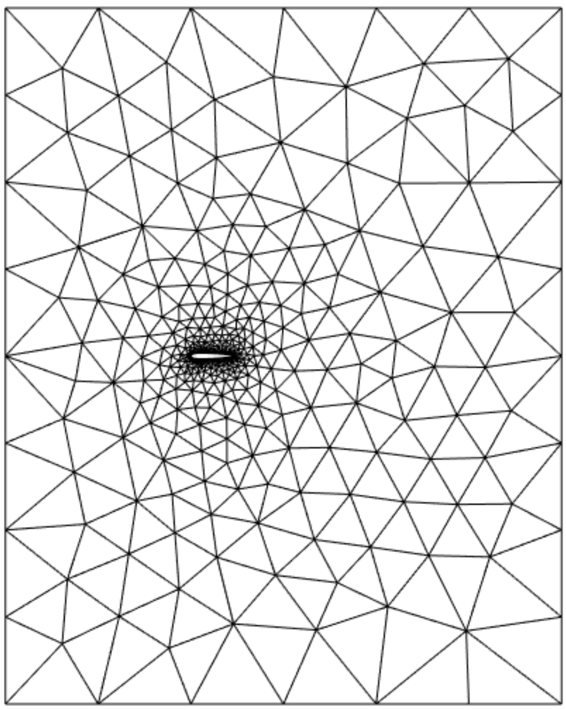

Figure 8. Discretized domain, 824 elements.

Fig.9 shows the pressure isolines obtained from numerical simulation applying DGM to N=9 and Fig. 10 presents the velocity isolines for this polynomial order.

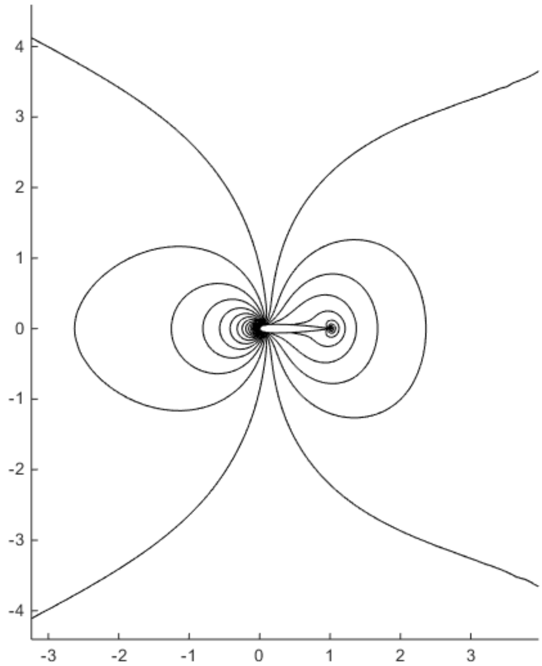

Figure 9. Isolines of pressure to N=9.

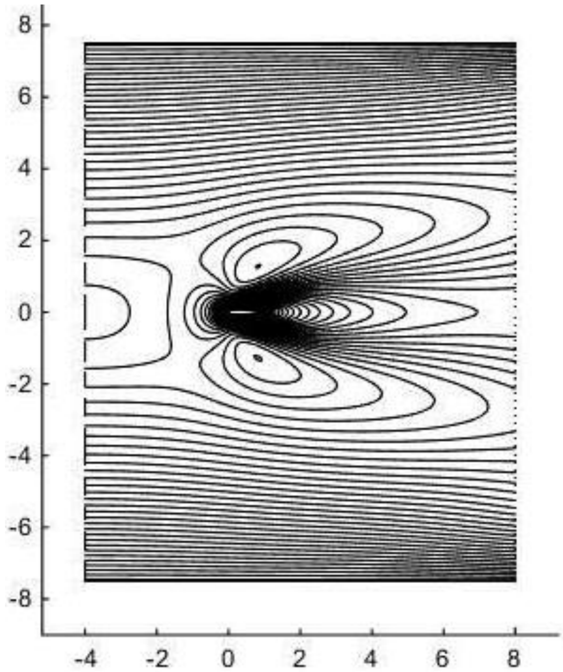

Figure 10. Isolines of velocity U_x to N=9.

The two-dimensional flow analysis over the aerodynamic NACA 0012 section is validated by NASA Langley Research Center NASA (2017). There are experimental data available which allows the validation, however, it should be recognized that the accuracy in two-dimensional is hard to achieve. The experimental curves used by the research center to compare Cp results consider values obtained from various publications with distinct Reynolds numbers. Among these publications Ladson (1987), Ladson (1988) e Gregory and O'Reilly (1970) can be mentioned.

The pressure coefficient (Cp), Eq.4, is calculated along the upper surface of the airfoil using the grid for α= 0° and *Re*=100 with distinctive polynomial degrees (N).

$$Cp = \frac{(P - P_\infty)}{\frac{1}{2}\rho U_\infty^2} \tag{4}$$

Where ρ is the static pressure at the point considered, P_∞ is the static pressure of the flow not disturbed and U_∞ is the velocity of the not disturbed flow.

The Fig. 11 shows the results obtained by the DGM code together with the experimental values obtained by other researchers.

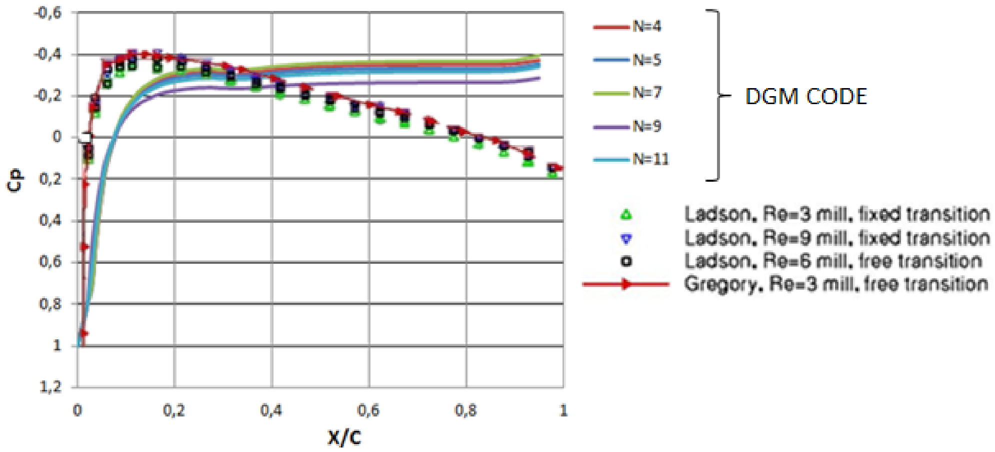

Figure 11. Coefficient pressure, α= 0°.

4. CONCLUSIONS

The numerical results obtained demonstrated that the DGM provides an accurate and physically consistent solution. The Cd, Cl and St were very close to the results found in the literature review for the Cylinder simulations. By observing the velocity isolines (Ux) one can detect that the solution is stable for all domains, primarily in the output limit. It is clear that both the DGM and the boundary condition applied are suitable to simulate engineering problems such as the ones which this paper is attempted to study.

5. ACKNOWLEDGEMENTS

The authors would like to acknowledge Fundação Araucária de Apoio ao Desenvolvimento Científico e Tecnológico, which provide a financial support under Grant 376/2014 and 230/2015.

6. REFERENCES

Dong, S., Karniadakis, G.E. and Chryssostomidis, C., 2014. "A robust and accurate outflow boundary condition for incompressible flow simulations on severely-truncated unbounded domains". *J. Comput. Phys*, Vol. 261, p. 83–105.

Garcia.A.Q., Gomes.F.A.A., Novak.P.R., 2016. "Numerical Analysis of the Incompressible Fluid Flow Around a Two-Dimensional Cylinder by High-Order Discontinuous Galerkin Method". In *16th Brazilian Congress of Thermal Sciences and Engineering, 2016.* Vitória, ES, Brazil.

Gregory, N. and O'Reilly, C. L., 1970 *"Low-Speed Aerodynamic Characteristics of NACA 0012 Aerofoil Sections, including the Effects of Upper-Surface Roughness Simulation Hoar Frost,"* NASA R&M 3726.

J. N. Reddy and D. K. Gartling., 2010. *The Finite Element Method in Heat Transfer and Fluid Dynamics*. CRC Press; 3rd edition.

J. S. Hesthaven and T. Warburton., 2008. *Nodal Discontinuous Galerkin Methods: Algorithms, Analysis, and Applications*.

Ladson, C. L., Hill, A. S., and Johnson, Jr., W. G., 1987 "Pressure Distributions from High Reynolds Number Transonic Tests of an NACA 0012 Airfoil in the Langley 0.3-Meter Transonic Cryogenic Tunnel," *NASA TM 100526*.

Ladson, C. L., 1988 "Effects of Independent Variation of Mach and Reynolds Numbers on the Low-Speed Aerodynamic Characteristics of the NACA 0012 Airfoil Section," *NASA TM 4074*.

M. Braack, P. B., 2014. Mucha, Directional do-nothing condition for the Navier-Stokes equations, "*Journal of Computational Mathematics*", 32 (No.5), p. 507–521.

NASA., 2017. "2D NACA 0012 Airfoil Validation Case". 15 Set. 2017. <https://turbmodels.larc.nasa.gov/naca0012_val.html>.

Schäfer, M. and Turek, S., 1996. "The benchmark problem flow around cylinder". In *H. EH, ed., Flow Simulation with High-Performance Computers II - Notes on Numerical Fluid Mechanics*. Vol. 52, p. 547–566.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.