NUMERICAL COMPUTATIONS OF THE CAVITY FLOWS USING THE POTENTIAL FLOW THEORY

Marcel ILIE[1], Augustin SEMENESCU[2], Gabriela Liliana STROE[3], Sorin BERBENTE[4]

Rezumat. *Metodele de calcul referitoare la dinamica fluidelor turbulente sunt costisitoare deoarece presupun folosirea unor calculatoare performante, care nu sunt totdeauna disponibile. De aceea, metode cum ar fi ecuaţiile Reynolds Navier-Stokes nu sunt practice pentru fluide cu un comportament variabil în timp; metoda simulărilor numerice directe (DNS) este ceea mai precisă însă nu este fezabilă pentru fluide cu numărul Reynolds mare. Metoda Large-Eddy Simulation (LES), de asemenea, nu este fezabilă datorită costului computaţional. De aceea, se caută metode computaţionale alternative. Această cercetare are ca scop dezvoltarea unei metode de calcul fezabilă pentru fluide incompresibile, în mod particular pentru fluide cavitaţionale, folosind metoda de potenţial al fluidelor. Metoda se bazează pe diferenţe finite. Discretizarea timpului şi spaţiului se face folosind scheme numerice de ordinul doi şi se poate realiza folosind un singur calculator. Analiza dinamicii fluidului identifică prezenţa vârtejului în centrul cavităţii şi la colţurile de jos.*

Abstract. *Computational fluid dynamics of turbulent flows requires large computational resources or are not suitable for the computations of transient flows. Therefore methods such as Reynolds-averaged Navier-Stokes equations are not suitable for the computation of transient flows. The direct numerical simulation provides the most accurate solution, but it is not suitable for high-Reynolds number flows. Large-eddy simulation (LES) approach is computationally less demanding than the DNS but still computationally expensive. Therefore, alternative computational methods must be sought. This research concerns the modelling of inviscid incompressible cavity flow using the potential flow. The numerical methods employed the finite differences approach. The time and space discretization is achieved using second-order schemes. The studies reveal that the finite differences approach is a computationally efficient approach and large computations can be performed on a single computer. The analysis of the flow physics reveals the presence of the recirculation region inside the cavity as well at the corners of the cavity.*

Keywords: numerical modeling, finite-differences, cavity flows

DOI https://doi.org/10.56082/annalsarscieng.2021.2.78

[1]PhD, Assistant Professor: Dept. of Mechanical Engineering, Georgia Southern University, Statesboro, GA 30458, USA, e-mail: milie@georgiasouthern.edu

[2]PhD, Professor, Dept. of Material Sciences, University Politehnica Bucharest, Bucharest, Romania, augustin.semenescu@upb.ro

[3] PhD, Assistant Professor, Department of Aeronautical Systems Engineering and Aeronautical Management ”Nicolae Tipei”, University Politehnica Bucharest, Bucharest, Romania, gabriela.mogos@upb.ro

[4] PhD, Assistant Professor, Department of Aeronautical Systems Engineering and Aeronautical Management ”Nicolae Tipei”, University Politehnica Bucharest, Bucharest, Romania, sorin.berbente@upb.ro

1. Introduction

The cavity flows has been of interest for decades and particularly nowadays for its use in the supersonic propulsion [1-15]. Cavity flow exhibits complex fluid dynamics such vortex dynamics and turbulent mixing [2-5]. The fluid dynamics inside the cavity is defined by the step and length of the cavity, specifically the length-to-depth (L/D) [7, 9, 11-14]. Usually, cavity flow is dominated by a large recirculation region which is also defined by the flow speed [9, 13]. Generally, the cavity flows are preferred for turbulent mixing and thus, nowadays the cavity it is used for supersonic combustion to enhance the turbulent mixing and hold the flame inside the combustion region [1]. A schematic of cavity flow used in supersonic combustion is shown in Figure 1.

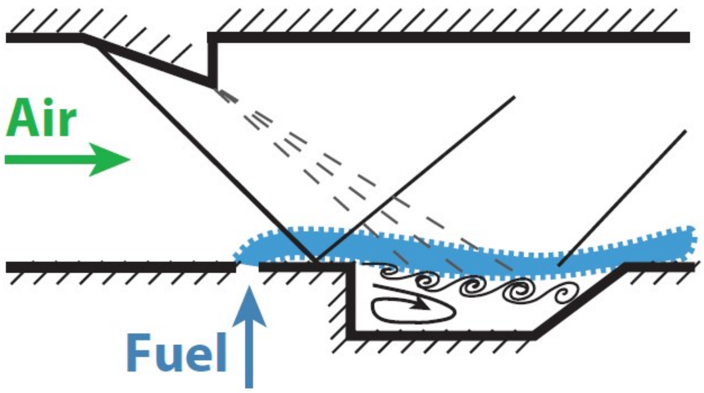

Figure 1. Cavity flow for supersonic combustion [1]

At moderate speeds the cavity flow exhibits a large recirculation region dominated by vortical flows [4-7]. Usually, at the boundary of the cavity a shear-layer is formed [15]. At high-speed, shocks and acoustic waves are generated [1]. The shear-layer impinges on the leeward side wall of the cavity and acoustic noise is generated [1].

2. Background

Generally, the numerical computations of cavity flows are challenging due to the high computational costs [5, 7]. Computational approaches such as Reynolds-averaged Navier-Stokes (RANS) are not suitable for time-dependent flow dynamics such as the cavity flow [5]. Thus, time-dependent numerical approaches must be sought. Direct Numerical Simulation (DNS) pose significant challenges due to the fact that the computational cost is proportional to $Re^{2.25}$ and thus, it is not suitable for the computation of the high-Reynolds number flows. On the other hand, the Large-Eddy Simulation (LES) still poses high computational costs. Thus, computationally efficient numerical approaches are sought. Numerical computations of incompressible, inviscid flows using the Laplace equations are a promising approach for the computation of cavity flows and it is employed in the present research.

3. Modeling and algorithms

This research concerns two-dimensional modeling of the cavity flow for the case of incompressible fluid. The governing equations for the incompressible flow are the Laplace equations defined as:

$$\frac{\partial^2\Psi}{\partial x^2}+\frac{\partial^2\Psi}{\partial y^2}=0$$

The Laplace equation is solved using the finite-differences approach and thus, the space and time-derivatives will be discretized using the second-order central-difference schemes. The space and time-discretization are presented in the following. Figure 2 shows the time and space-marching schemes with space on the horizontal and time on the vertical. For simplicity, an equally spaced computational domain will be used, with the space and time variables h and t. As shown in Figure 2, for the space marching of the solution, the time is fixed and solution is computed at the computational grid points $f(t,x-h)$, $f(t,x)$ and $f(t,x+h)$.

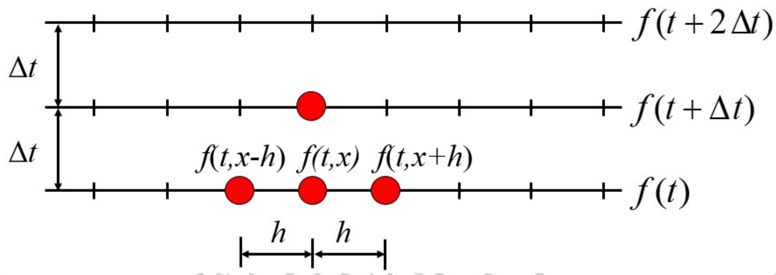

Figure 2. Space and time-discretization

Figure 3 shows the time-discretization of the Laplace equation. For the time-marching, the space is kept constant while the time increment is $t+\Delta t$.

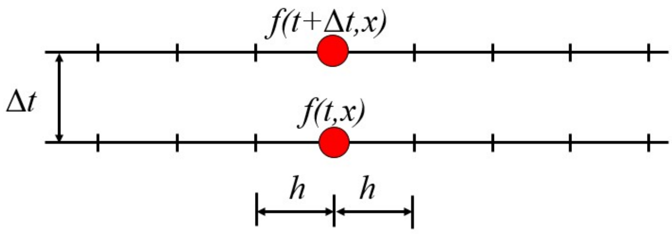

Figure 3. Time-discretization

The finite-difference approach makes use of the Taylor series. Thus, the Taylor series is sued for an arbitrary function f , where f can represent the flow variables such as pressure or velocity. The Taylor series expanded about the grid point $(x+h)$ becomes

$$f(x+h)=f(x)+\frac{\partial f(x)}{\partial x}h+\frac{\partial^2 f(x)}{\partial x^2}\frac{h^2}{2}+\frac{\partial^3 f(x)}{\partial x^3}\frac{h^3}{6}+\frac{\partial^4 f(x)}{\partial x^4}\frac{h^4}{24}+....$$

$$f(x-h)=f(x)-\frac{\partial f(x)}{\partial x}h+\frac{\partial^2 f(x)}{\partial x^2}\frac{h^2}{2}-\frac{\partial^3 f(x)}{\partial x^3}\frac{h^3}{6}+\frac{\partial^4 f(x)}{\partial x^4}\frac{h^4}{24}+....$$

Subtracting equation 3 from equation 2 and truncating the high-order terms, the difference $f(x+h)-f(x-h)$ is obtained such that

$$f(x+h)-f(x-h)=2\frac{\partial f(x)}{\partial x}h+2\frac{\partial^3 f(x)}{\partial x^3}\frac{h^3}{6}+....$$

Rearranging this equation to isolate the first derivative, we obtain the following equation:

$$\frac{\partial f(x)}{\partial x}=\frac{f(x+h)-f(x-h)}{2h}-\frac{\partial^3 f(x)}{\partial x^3}\frac{h^2}{6}$$

Next, we need to find the second derivative of function f . Making use again of the Taylor series

$$f(x+h)=f(x)+\frac{\partial f(x)}{\partial x}h+\frac{\partial^2 f(x)}{\partial x^2}\frac{h^2}{2}+\frac{\partial^3 f(x)}{\partial x^3}\frac{h^3}{6}+\frac{\partial^4 f(x)}{\partial x^4}\frac{h^4}{24}+....$$

$$f(x-h)=f(x)-\frac{\partial f(x)}{\partial x}h+\frac{\partial^2 f(x)}{\partial x^2}\frac{h^2}{2}-\frac{\partial^3 f(x)}{\partial x^3}\frac{h^3}{6}+\frac{\partial^4 f(x)}{\partial x^4}\frac{h^4}{24}+....$$

Adding equations, we obtain:

$$f(x+h)+f(x-h)=2f(x)+2\frac{\partial f^2(x)}{\partial x^2}\frac{h}{2}+2\frac{\partial^4 f(x)}{\partial x^4}\frac{h^4}{24}+....$$

Rearranging this equation to isolate the second derivative, we obtain the second derivative of function f in the form:

$$\frac{\partial^2 f(x)}{\partial x^2}=\frac{f(x+h)-2f(x)+f(x-h)}{h^2}-\frac{\partial^4 f(x)}{\partial x^4}\frac{h^2}{12}+...$$

Eliminating the high-order terms the second derivative becomes:

$$\frac{\partial^2 f(x)}{\partial x^2}=\frac{f(x+h)-2f(x)+f(x-h)}{h^2}$$

4. Results and discussion

Figure 4 presents the streamlines of the cavity flow. From the analysis of the streamlines, it is observed that the flow impinges on the sides of the cavity. As the flow impinges on the leeward side of the cavity, the flow field inside the cavity is disturbed and thus, a recirculation region is generated. The recirculating flow, inside the cavity, is impinging on the step and bounces back into the cavity and thus, the recirculation region is maintained.

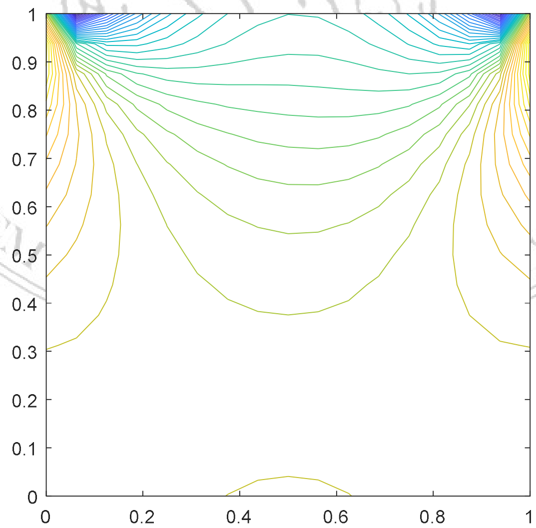

Figure 4. Streamlines of the cavity flow

The recirculation region is better illustrated in Figure 5. As shown in Figure 5, the recirculation region exhibits a very low velocity at the core of the vortex while the velocity presents larger values closer to the boundaries. From the analysis of the streamline, two small recirculation regions are observed at the bottom corners of the cavity.

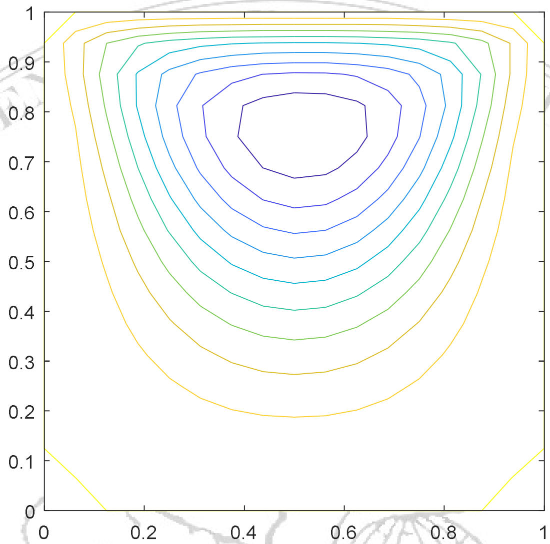

Figure 5. Streamlines inside the cavity flow

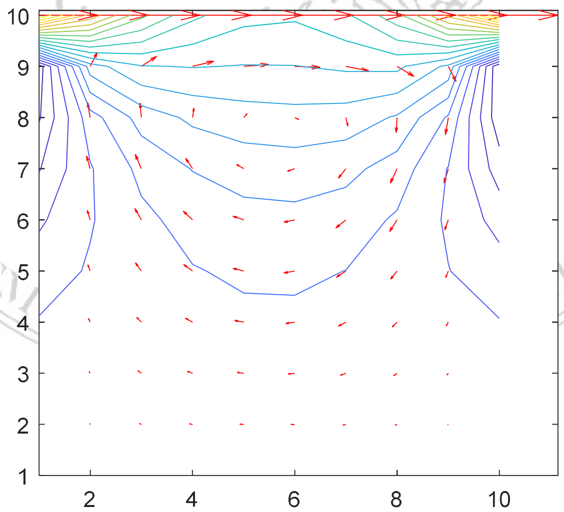

Figure 6. Velocity field and streamlines inside the cavity

Figure 6 presents the streamlines and superimposed velocity vector field. The analysis of the fluid flow in Figure 6 shows that the velocity reaches a zero value such that it satisfies the boundary conditions. The analysis also reveals that the velocity exhibits a minimum value at the core.

Figure 7, *a* presents the velocity vector field from two-dimensional computations. The analysis of the vector field reveals the presence of the recirculation region at the center of the cavity. Also, it can be observed that the velocity is zero near the wall to satisfy the no-slip boundary conditions. The velocity exhibits a minimum at the core of the cavity.

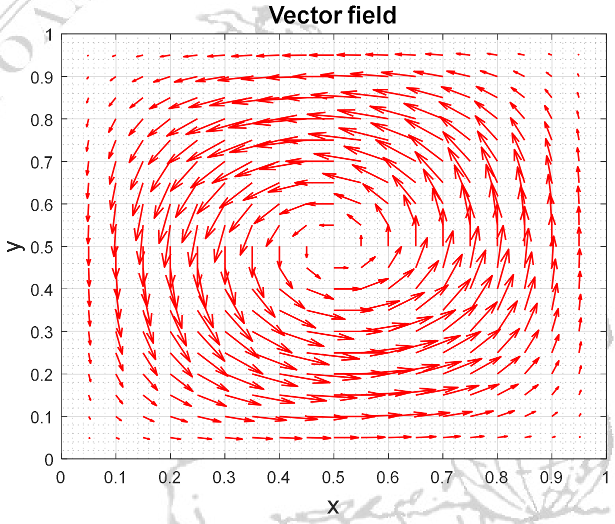

Figure 7, *a*. Velocity vector field

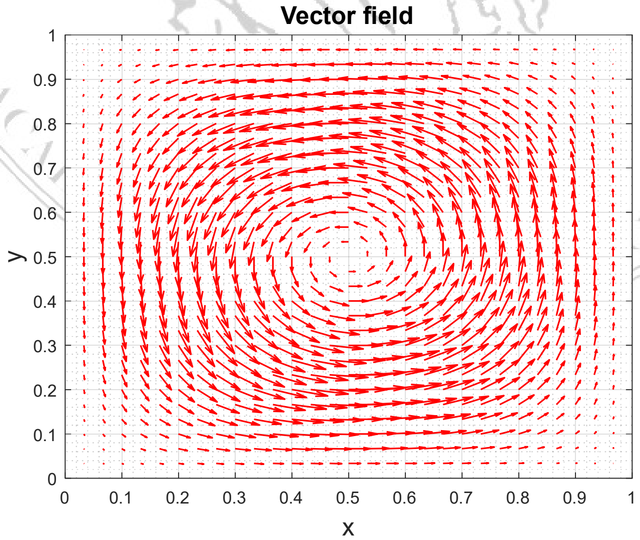

Figure 7, *b*. Velocity vector field

A grid convergence analysis was performed as well and the results are shown in Figures 7, *a* and 7, *b*. Therefore, increasing the number of grid points of the computational results in a more refined solution. However, it is worth mentioning that the increase of the computational grid points cause a negligible increase of the computational time, more specifically $t = 10^{-4} s$. The present research shows that the potential flow theory provides accurate solution of the cavity flow with a much lower computational cost compared with advanced CFD solutions such as LES or DNS.

Conclusions

A computational model for inviscid, incompressible flow was developed. The numerical algorithm is used for the computations of the cavity flow. The numerical algorithm uses the two-dimensional Laplace equations. The space and time-discretization of the computational domain is achieved using second-order numerical schemes. The analysis reveals that the computational domain capture very well the flow physics of the cavity flow, at a much lower computational cost when compared with direct numerical simulation (DNS) and large-eddy simulations (LES). The computations were carried out suing the MATLAB software.

R E F E R E N C E S

[1] Urzay, J., Supersonic Combustion in Air-Breathing Propulsion Systems for Hypersonic Flight, Annual Review of Fluid Mechanics, 50:1, 593-627, 2018

[2] E. Bilgen, H. Oztop, Natural convection heat transfer in partially open inclined square cavities, International Journal of Heat and Mass Transfer 48 (2005) 1470–1479, 2005

[3] K. Chang, G. Constantinescu, S.-O. Park, Journal of Fluid Mechanics 561, 113–145, 2006

[4] B. Hong, B.F. Armaly, T.S. Chen, Mixed convection in a laminar, vertical, backward- facing step flow: solution to a benchmark problem, Computational aspects of heat transfer-benchmark problems, HTD, vol. 258, 1993, pp. 57–62.

[5] I. Iglesias, J.A.C. Humphrey, F. Giralt, Numerical calculations of two-dimensional, buoyancy-assisted flow past a backward facing step in a vertical channel, Computational aspects of heat transfer — benchmark problems, HTD, vol. 258, pp. 57–62, 1993

[6] B.P. Leonard, A stable and accurate convective modeling procedure based on quadratic upstream interpolation, Computer Methods in Applied Mechanics and Engineering 19, 59–88, 1979

[7] J.C. Leong, N.M. Brown, F.C. Lai, Mixed convection from an open cavity in a horizontal channel, International Communications in Heat and Mass Transfer 32, 583–592, 2005

[8] O. Manca, S. Nardini, K. Khanafer, K. Vafai, Effect of heated wall position on mixed convection with an open cavity, Numerical Heat Transfer Part A 43, 259–282, 2003

[9] O. Manca, S. Nardini, K. Vafai, Experimental investigation of mixed convection in a channel with an open cavity, Experimental Heat Transfer 19, 53–68, 2006

[10] J. Pallares, I. Cuesta, F.X. Grau, F. Giralt, Natural convection in a cubical cavity heated from bellow at low Rayleigh numbers, International Journal of Heat and Mass Transfer 39, 3233–3247, 1996

[11] E. Papanicolaou, Y. Jaluria, Mixed convection from an isolated heat source in a rectangular enclosure, Numerical Heat Transfer, Part A: Applications 18, 427–461, 1990

[12] E. Papanicolaou, Y. Jaluria, Transition to a periodic regime in mixed convection in a square cavity, Journal of Fluid Mechanics 239, 489–509, 1992

[13] J.C.F. Pereira, J.M.M. Sousa, Experimental and numerical investigation of flow oscillations in a rectangular cavity, Journal of Fluids Engineering 115, 68–73, 1995

[14] Y. Stiriba, Analysis of the flow and heat transfer characteristics for assisting incompressible laminar flow past an open cavity, International Communications in Heat and Mass Transfer 35, 901–907,2008

[15] Y. Stiriba, F.X. Grau, J.A. Ferré, A. Vernet, A numerical study of three-dimensional laminar mixed convection past an open cavity, International Journal of Heat and Mass Transfer 53, 4797–4808, 2010.