ISSN: 0067-2904

Numerical Investigation of Newtonian Flow of Conical Nozzle: Taylor Galerkin Pressure Correction Finite Element Method

Ahmed M. Shareef[1], Alaa H. Al-Muslimawi[1], Alaa A. Sharhan[2]*
[1]Department of Mathematics, College of Science, University of Basrah, Basrah, Iraq
[2]General Directorate of Education of Baghdad/First Rusafa, Baghdad, Iraq

Received: 18/3/2024 Accepted: 17/10/2024 Published: 30/12/2025

Abstract

In the current research, we dealt with the numerical method of Newtonian flow through the conical nozzle under incompressibility constraint by using a Taylor Galerkin Pressure Correction (TGPC) finite element algorithm. The governing equations that describe such problem consist of time-dependent equations for the conservation of momentum and continuity equation. The essential feature of this investigation is concerned with the effect of outlet dimeter of nozzle, length of the cylinder and conical angle on the component of solution. There, we found that the levels of velocity and pressure are increased as these parameters increase, it was found that the maximum velocity increased from 38.39 to 1254.67 when the ratio I_2/d_2 increased from 1 to 5 and the maximum pressure also increased from 753.83 to 1254.67. The effect of conical angle was also studied, increasing the β angel from 21.8 to 90 resulted an increasing in the maximum velocity from 13.53 to 30.79 and an increase in the maximum pressure from 214.26 to 615.7. While the maximum velocity decreased from 675.8 to 67.79 when the outlet diameter increased from 0.5 to 1.5 also the maximum pressure decreased from $15.79 * 10^4$ to 2268.21.

Keywords: Conical nozzle; Finite element method; Taylor Galerkin Pressure Correction; Newtonian fluids.

الاستقصاء العددي للتدفق النيوتوني في الفوهة المخروطية: طريقة تايلر جالركين لتصحيح الضغط للعناصر المحدودة

احمد محمد شريف[1], علاء حسن المسلماوي[1], علاء عبد الواحد شرهان[2]*
[1]قسم الرياضيات, كلية العلوم, جامعة البصرة
[2]المديرية العامة للتربية في بغداد الرصافة/الاولى

الخلاصة

تناولنا في هذا البحث الطريقة العددية للتدفق النيوتوني غير القابل للانضغاط عبر الفوهة المخروطية باستخدام طريقة تايلرجالركين لتصحيح الضغط للعناصر المحدودة. تتكون المعادلات الحاكمة التي تصف هذه

*Email: alaa11187@gmail.com

المشكلة من معادلة الاستمرارية لحفظ الكتلة والمعادلات المعتمدة على الزمن لحفظ الزخم. السمة الرئيسية لهذه الدراسة هي تأثير قطر مخرج الفوهة وطول الاسطوانة والزاوية المخروطية على مكونات الحل. وهناك، وجدنا أن مستويات السرعة والضغط تزداد بزيادة هذه المعلمات، ووجدنا أن السرعة القصوى زادت من 38.39 إلى 1254.67 عندما زادت النسبة I_2/d_2 من 1 إلى 5 وزاد الضغط الأقصى أيضًا من 753.83 إلى 1254.67. و تمت دراسة تأثير الزاوية المخروطية أيضًا، حيث أدى زيادة زاوية β من 21.8 إلى 90 إلى زيادة السرعة القصوى من 13.53 إلى 30.79 وزيادة الضغط الأقصى من 214.26 إلى 615.7. بينما انخفضت السرعة القصوى من 675.8 إلى 67.79 عندما زاد قطر المخرج من 0.5 إلى 1.5 كما انخفض الضغط الأقصى من $10^4 * 15.79$ إلى 2268.21 .

1. Introduction

Nozzles are utilized in a different of industries several reasons, including accelerating flow for the atomization of liquid phases, enhancing kinetic energy and accelerating gas in rocket engines, and accelerating gas velocity in natural gas production wells and ets. In addition, cone flow researches are often achieved in the industrial sector to address the relevant problems in this field. Basically, the nozzle shape represents crucial matter to produce choked flow conditions. Also, improvement of the nozzle design represents one of the majority applications to reduce pressure loss brought on by higher velocities at the nozzle throat. Moreover, conical nozzles and their structure play an important role in outflow problems over widely fields of industrial. In this context, the many designs have been proposed to treat different types of applications (see for example T. Jiang et al, J. Li et al and Y. Lu et al [1-3]). Therefore, and for these reasons we highlighted on this problem, especially on the nozzle geometry design and the effects of geometric parameters (for more detail see T. Jiang et al [1]). The last decade has seen considerable advance in the treatment of such problem. In this context, numerous numerical studies have been conducted and takes a wide area in literature. Accordingly, many studies have been conducted to see the impact of nozzles geometric characterise on the flow Z.-Y. Sun et al [4]. Schmidt et al. [5] used a two-phase, two-dimensional and transient model of cavitating nozzle flow to see the influence of the parameters of nozzle. Numerical study of compressible, cavitating flow is conducted by Jian et al. [6]. There the numerical results are compared with experiment data. Echouchene et al. [7] adopted a mixture model based on a single fluid to study the impact of wall roughness in the turbulent and cavitating flow developing within a Diesel injector numerically. Numerical analysis is applied to study the influences of cavitation upon a proportional, directly-operated, hydraulic and directional valve by Amirante et al. [8]. Chunyan et al. [9] studied the influence of geometric parameters on cavitation characteristics. These authors analyzed the effect of conical angle, inner-wall roughness and ratio of orifice length to diameter. Ahmed N. Abdulhasan and Alaa H. Al-Muslimawi [10] developed a numerical method to treat the flow through a conical channel of Newtonian laminar fluids based on Galerkin finite element method.

There are many numerical methods for solving flow problems for example, (see M. F. Ahmed et al and F. Ali & N. Summayya [11, 12]). But the current study, we used a time stepping Taylor Galerkin Pressure Correction (TGPC) finite element technique, which was proposed by Townsend and Webster (see [13]). In this algorithm, two methods, a Taylor Galerkin approach and a pressure correction approach are applied to develop the current method. Taylor Galerkin method is a two-step Lax-Wendroff time stepping procedure (predictor-corrector), extracted via a Taylor series expansion in time (see Donea [14]). In contrast, the pressure-correction method accommodates the incompressibility constraint to ensure second-order accuracy in time (see Ren & Liu. [15], Shen & Liu. [16]). Recently, this method has been employed widely to address various fluid problems in both Cartesian coordinates system and cylindrical coordinates system (for more details see N. Thongjub and R. B Khokhar et al [17, 18]). The present article expands upon the important numerical themes by using one of efficient algorithm to provide a detailed study of Newtonian flow through the conical nozzle. Here, we consider the performance of the TGPC-method to address conical nozzles problem for first time, which represents a novel idea did not address by researchers previously. Practically, the design of the conical nozzle directly affects the thrust or flow rate, making it a critical component for achieving desired operational goals. So, a special structure of conical is used in this study with specific dimensions to provide efficient benefit in the flow of jet propulsion systems. In order to highlight this issue, the influence of geometric parameters of conical nozzles, such as downstream length, outlet diameter and cone's half-angle is investigated widely in this article.

The following sections of the study are introduced as, mathematical model of the Newtonian fluid flow in the next section. The discretization method that utilized to treat the governing equations will be introduced in Section 3. The specification of problem and numerical findings will be presented in Sections 4 and 5, respectively.

2. Mathematical modelling

With ignoring body forces, the momentum and continuity equations of incompressible flow under isothermal circumstances are as follows (see A. Khayyer et al and E. DiBenedetto & U. Gianazza [19, 20]):

$$\nabla \cdot (u) = 0, \tag{1}$$

$$\rho\left((\frac{\partial u}{\partial t}) + u \cdot \nabla u\right) = -\nabla p + \nabla \cdot (2\mu\, g). \tag{2}$$

Here, u, p and ρ, indicate velocity, pressure, and density, respectively, while μ represents solvent viscosity, as well as, $g = \frac{1}{2}(\nabla u + \nabla u^{\tau})$ is the rate of deformation, and ∇ is the operator for derivatives.

In addition, the relavent equation can alternatively be defined by non-dimensional Reynolds number groups, by utilizing the relationship $Re = \rho\frac{ZL}{\mu}$, (see D. Chandran et al and G. Kefayati [21, 22]) as well as (Z), (L) and (ρ) characteristic velocity, density and length, respectively see A. Khayyer et al, E. DiBenedetto & U. Gianazza, D. Chandran et al and G. Kefayati [19-22]. Thus, in this instance, in general Newtonian, the non-dimensional formulation of the numerical solution may be stated as:

$$Re\left((\frac{\partial u}{\partial t}) + u \cdot \nabla u\right) = -\nabla p + \nabla \cdot (2\mu\, g). \tag{3}$$

3. Numerical method

To solve the governing equations numerically, (TG-PC)-method is employed in this study. Actually, this approach consists of three fractional stages. The first stage contains two steps, in the first step the initial velocity and pressure fields are used to compute $u^{n+\frac{1}{2}}$ and in second step u* components is evaluated using a two-step predictor-corrector process. During the second stage, differential pressure $[P^{n+1} - P^n]$ is computed utilizing $u^*$and the Choleski technique. In the third stage, $((P^{n+1} - P^n))$ and (u^*) are employed to compute the velocity u^{n+1} using "Jacobi iteration". These partial stages are exported as:

$$\text{Stage [1a]: } \frac{2Re}{\Delta t}\left[(u^{n+\frac{1}{2}} - u^n)\right] = L\,[(u^n, g^n)] - (\nabla p^n), \tag{4}$$

$$\text{Stage [1b]: } \frac{Re}{\Delta t}\,([\,u^* - u^n]) = L\left[\left(u^{n+\frac{1}{2}}, g^{n+\frac{1}{2}}\right)\right] - (\nabla p^n), \tag{5}$$

$$\text{Stage[2]: } \nabla^2[(p^{n+1} - p^n)] = \frac{Re}{\theta \Delta t}(\nabla \cdot u^*), \quad (6)$$

$$\text{Stage[3]: } u^{n+1} = (u^*) - \frac{\theta \Delta t}{Re}[\nabla([p^{n+1} - p^n])]. \quad (7)$$

Where,

$$L[u, g] = [\nabla \cdot (2\mu\ g) - Re\ \ u \cdot (\nabla u)]. \quad (8)$$

Also, $\theta \in [0,1]$, if you choose $\theta = 1/2$, the "Crank-Nicolson" scheme (a temporal approach of second order) is commonly employed and is known as the "Crank-Nicolson" parameter. The following formulae can be used to approximate velocity and pressure:

$$u(x, t) = \sum_{j=1}^{Ju} u_j(t)\phi_j(x), \quad (9)$$

$$p(x, t) = \sum_{j=1}^{Jp} p_j(t)\psi_j(x), \quad (10)$$

where $[J_u]$ signifies the all-nodes number and $[J_p]$ the number of triangular vertices. The vectors $[u_j(t)]$ and $[p_j(t)]$ indicate the vectors of velocity and pressure nodal values, respectively. $[\phi_j(x)]$ and $[\psi_j(x)]$ are the (Interpolation or shape) functions. (u^*) and pressure difference are represented by comparable shapes. The domain Ω is subdivided into elements, as well as the velocity at the nodes on the half and vertex, pressure is calculated using only the vertices nodes of a triangle. Among the several shape functions, $[\phi_j(x)]$ is the basis function has been chosen as a quadratic function and $[\psi_j(x)]$ is the linear basis function. Hence, the equivalent (TGPC) derived from (4), (5), (6), and (7) may be expressed in the matrices form as follows.

$$\text{Step [1a]: } \left[\frac{2Re}{\Delta t}M + \frac{1}{2}S\right]\left(Z^{n+\frac{1}{2}} - Z^n\right) = \left\{-[S + Re\ \ N(Z)]Z + \mathcal{P}^T P\right\}^n, \quad (11)$$

$$\text{Step [1b]: } \left[\frac{Re}{\Delta t}M + \frac{1}{2}S\right](Z^* - Z^n) = \left\{-SZ + \mathcal{P}^T P\right\}^n - Re([N(Z)Z])^{n+\frac{1}{2}}, \quad (12)$$

$$\text{Step [2]: } K(P^{n+1} - P^n) = -\frac{Re}{\theta \Delta t}\mathcal{P}Z^*, \quad (13)$$

$$\text{Step [3]: } \frac{Re}{\Delta t}M([Z^{n+1} - Z^*]) = \theta \mathcal{P}^T([P^{n+1} - P^n]). \quad (14)$$

Where, Z^n, Z^{n+1} and P^n, P^{n+1} are the velocity and pressure nodal vectors through the moment $[t^n]$ and $[t^{n+1}]$; and $[Z^*]$ is presented the intermediary velocity vector in Step 1b. *M, N, S,* $\mathcal{P}$*,* and *K* depict mass matrix, convective, momentum diffusion, pressure gradient/divergence, and matrix of pressure stiffness, respectively.

4. Specification of the problem and its boundaries:

In this current work, flow through benchmark 2D conical axisymmetric channel for Newtonian fluid flow is presented under isothermal conditions. Accordingly, the triangular finite element mesh, namely fine mesh (FM), is implemented in this study. This mesh and geometric configuration diagrams are shown in Figure.1, with mesh characteristic parameters per region are presented in Table 1. For this investigation, the equations were processed using a Matlab programme.

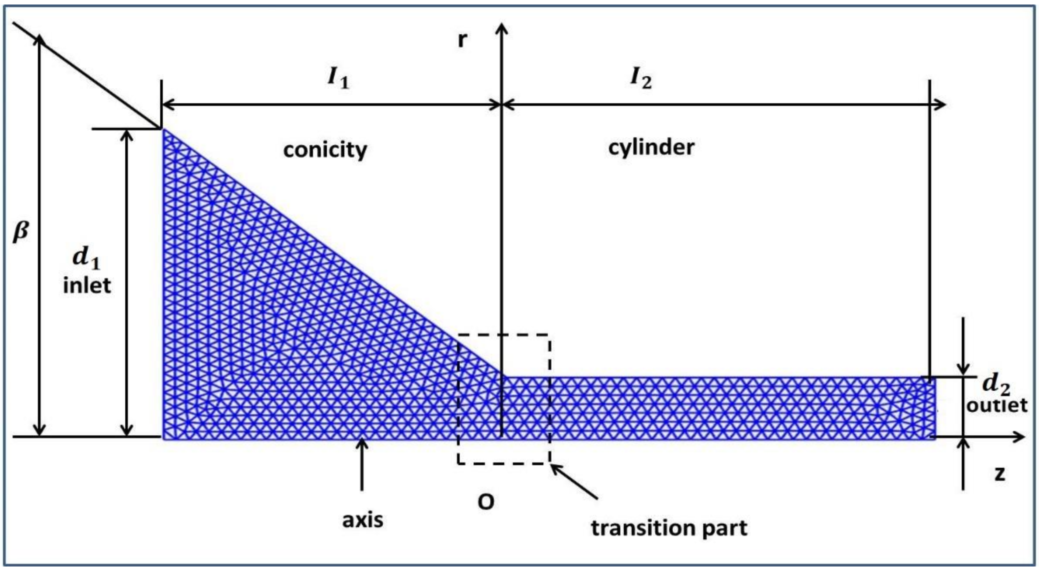


Figure 1: Conical nozzle geometry drawing

Boundary conditions (BCs): The boundary conditions of current canonical nozzle are set as:
(a) Zero radial velocity is imposed at the inlet with Poiseuille (Ps) flow for axial velocity.
(b) Free axial velocity (u_z) is assumed through axis of symmetry with ignoring radial velocity.
(c) No-slip BCs are imposed on the up walls of the channel.

Table 1: Conical nozzle's primary geometric characterizes.

Item	Value
Inlet Diameter [d_1]	*10*
Outlet Diameter [d_2]	*2*
Length of the conicity, [I_1]	*10*
Length of the cylinder,[I_2]	*8*
Conical angle, (β)	*21.8*

5. Results and discussion

In Figure 2 the contour distribution of pressure and velocity are presented with the outlet diameter (d_2= 2) and various value of ($I_2/\ d_2$=1, 2, 3, 4, 5), for that see (Table 2). The results reveal that, whenever the value of $I_2/\ d_2$ increases, it will be accompanied by an increase in speed of flow and pressure due to the increase in the length of the cylinder according to Bernoulli's equation. In addition, increase in the value of ($I_2/\ d_2$) had effect on the intensity of the cavity as shown by the contour distribution.

Table 2: Details about the conical nozzle's I_2/d_2.

Diameter	Length	Ratio
d_2	*2*	I_2/d_2
I_2	*2*	*1*
	4	*2*
	6	*3*
	8	*4*
	10	*5*

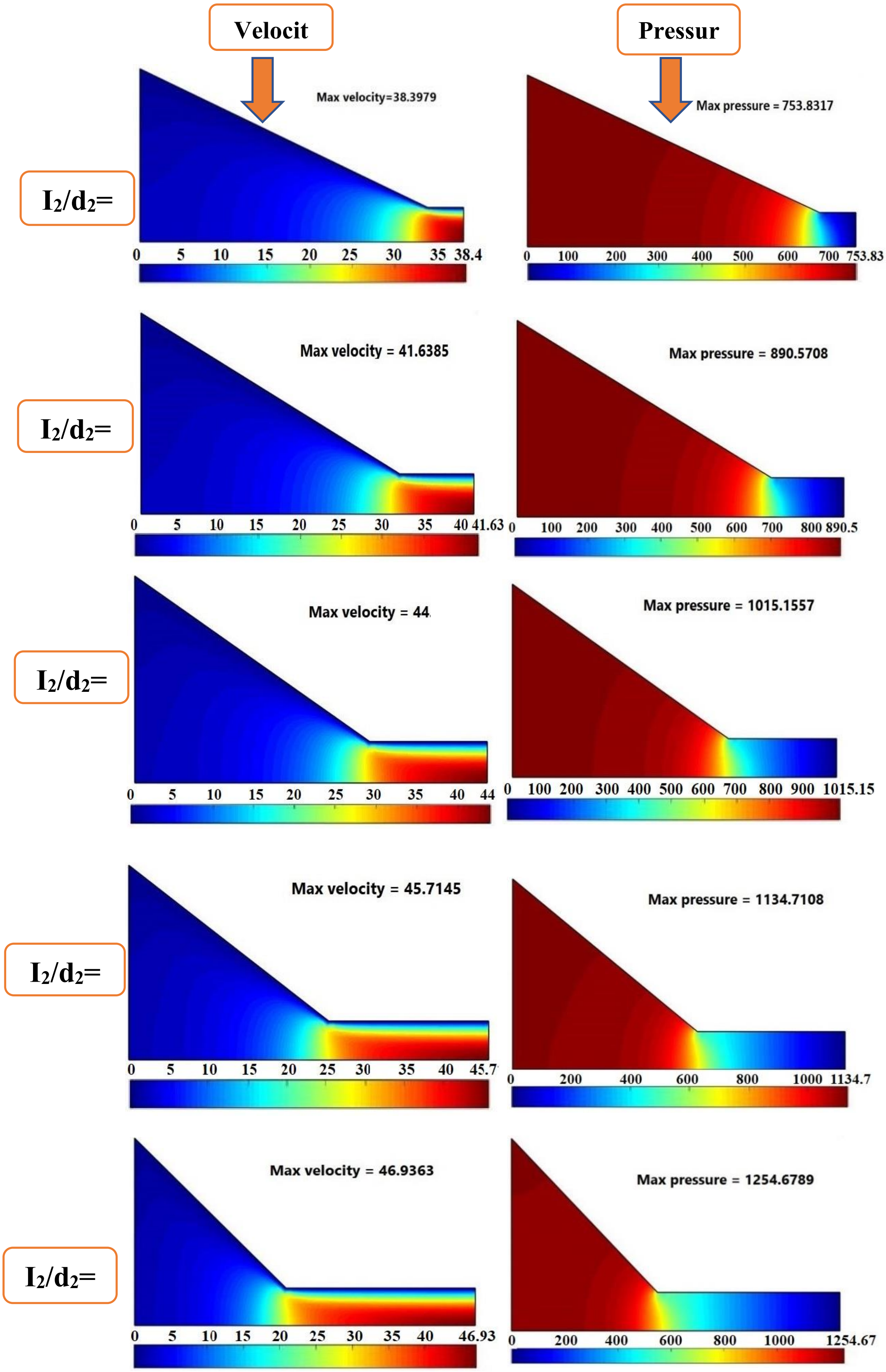


Figure 2: Velocity and pressure fields: effect of different $I_2/\ d_2$ $(d_2 = 2)$

For more detail, the profiles of velocity and pressure are presented as functions of $I_2/\ d_2$ in Figure 3, for same setting f conical nozzle parameters. From the results, we observed that increase in the ratio $I_2/\ d_2$ leads to rise in the level of velocity and pressure. In contrast, increase in the outlet diameter gives decrease in the pressure and velocity. For example, with d_2=1 the maximum velocity with I_2=8 is around 182 units, while for d_2=2 and I_2=8 the maximum value of velocity around 46 units almost O (74%) reduction. The present results are consistent with the results of previous studies [9]. To give more details we presented the fields of velocity and pressure under the effect of outlet diameter in Figure 4.

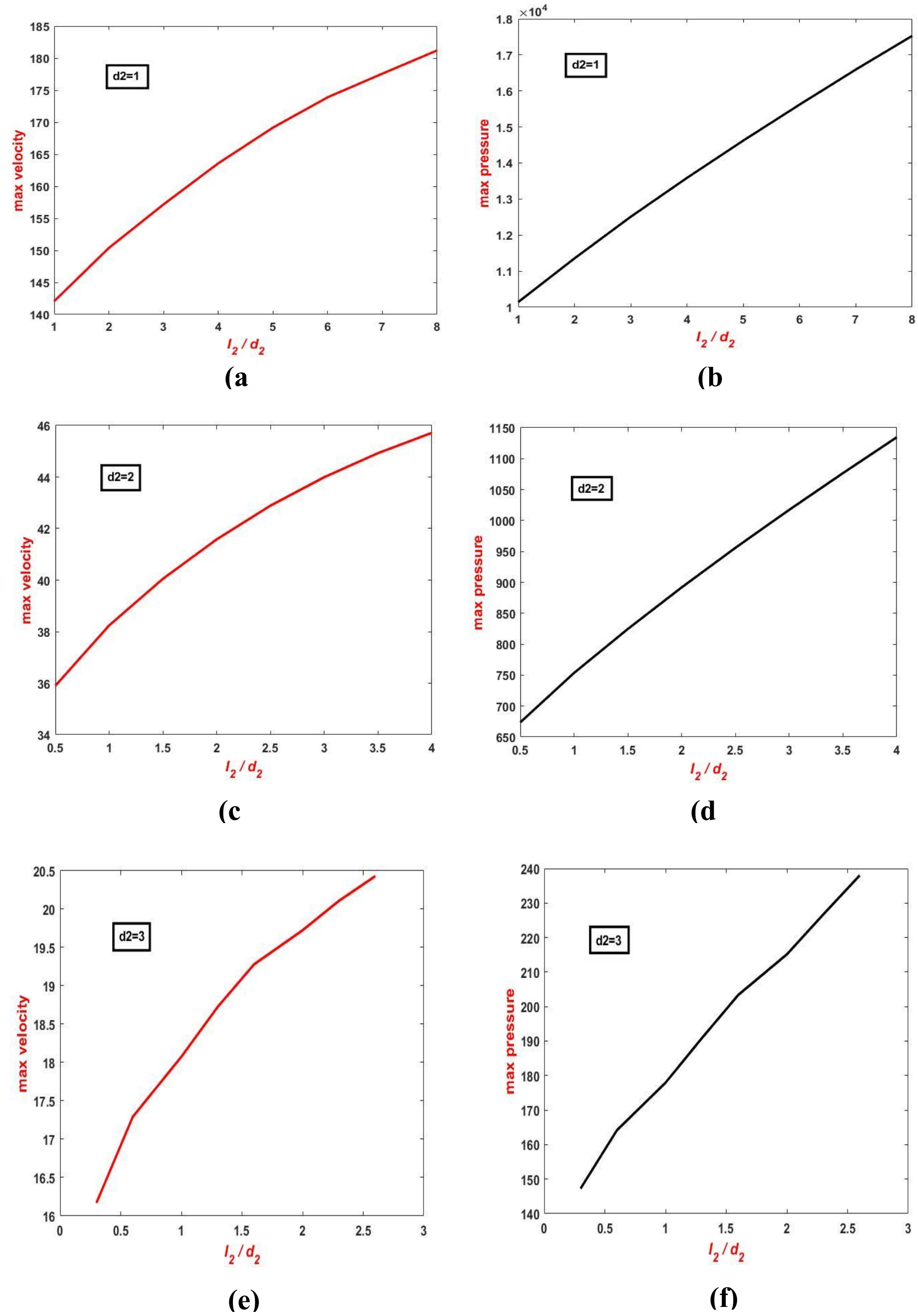


Figure 3: Maximum velocity and pressure under different (I_2/d_2)

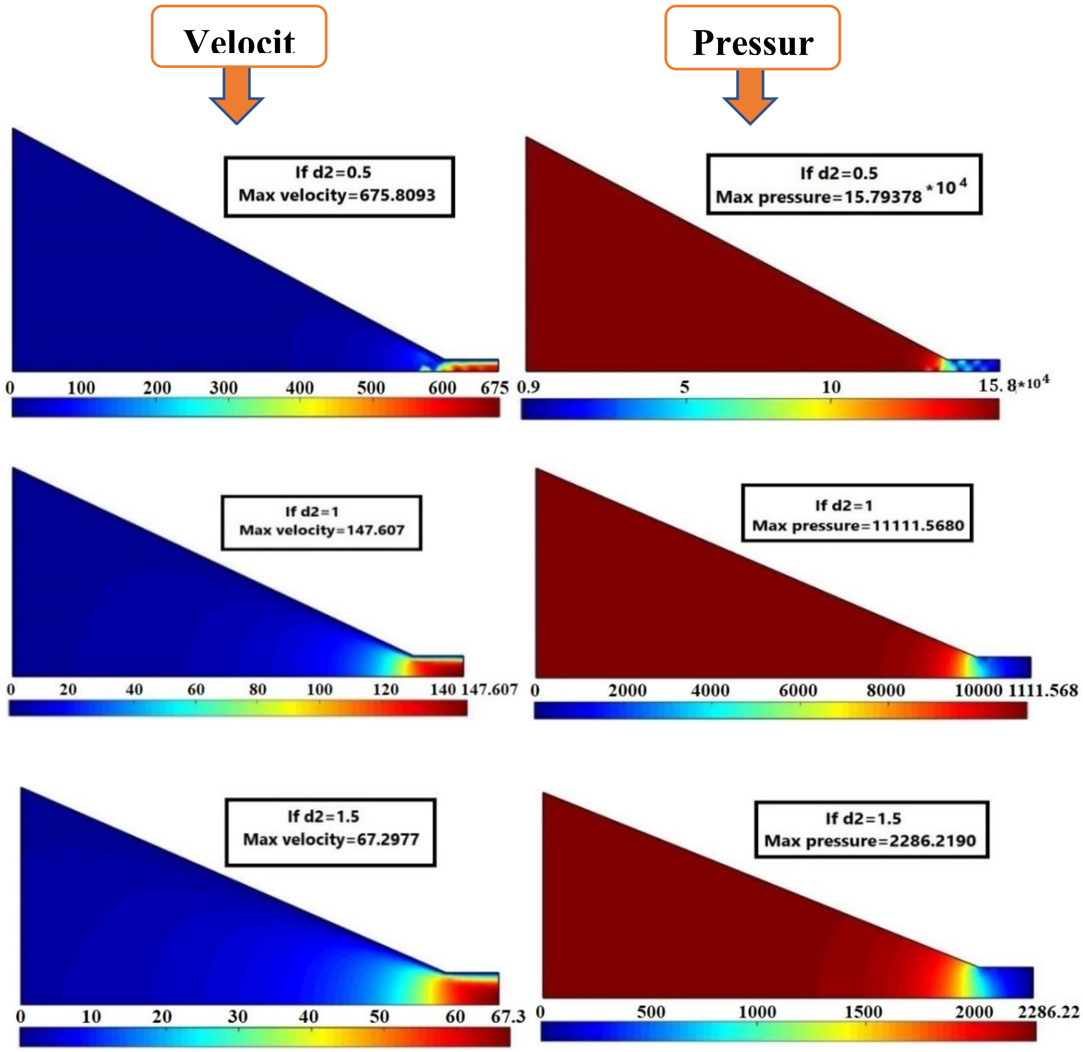


Figure 4: Velocity and pressure in [d_2=0.5, 1, 1.5]

Addition to above, the velocity and pressure profiles are illustrated in Figure 5 through centerline of the conical geometry for the various levels of d_2={1, 2, 3}. The findings give that, the velocity is increased until reach to the maximum level at the exit of the conical, while an opposite feature is appeared for the pressure.

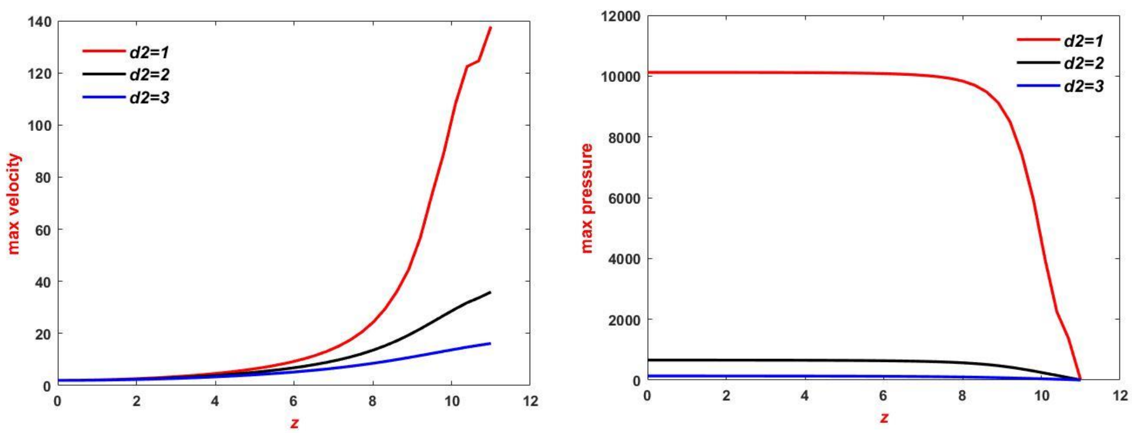
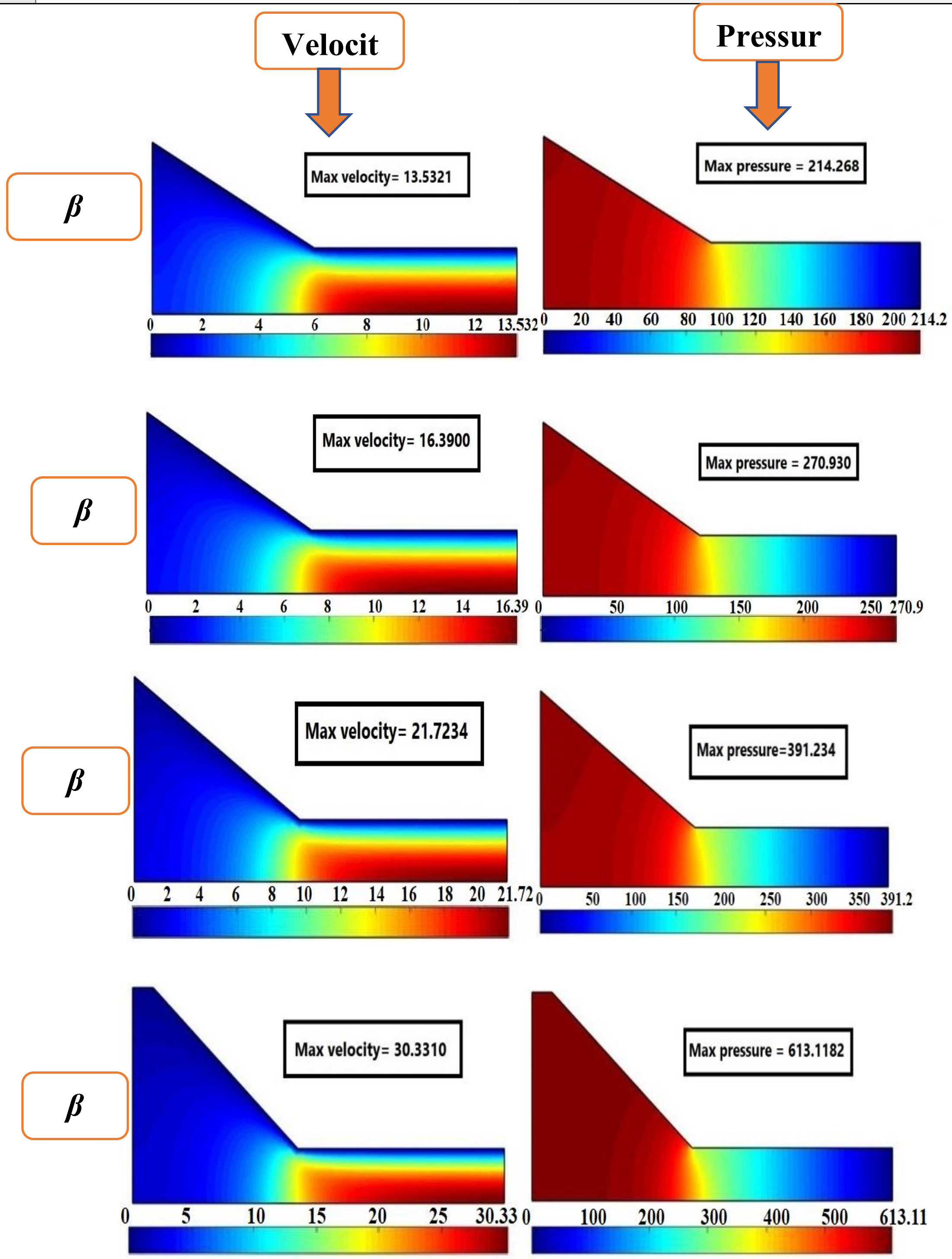


Figure 5: Velocity and pressure in [d_2=1, 2, 3]

Velocity has been presented in Figure 6 with d_2=2 and I_2=10 . From the contours we can see that, increase in the conical angles leads to reduce the pressure and change the direction of flow inside the nozzle. Moreover, insignificant reduction in the level of velocity is occurred as conical angles increases from 21.8 to 90, (see table 3). In addition, one can conclude that the effect of the ratio $I_2/\ d_2$ on the solutions is higher than the influence of conical angle (see Figure 2).

Table 3: Detailed detail about the conical nozzle

Item	1	2	3	4	5	6	7	8	9
β	*21.8*	*25*	*30*	*40*	*50*	*60*	*70*	*80*	*90*



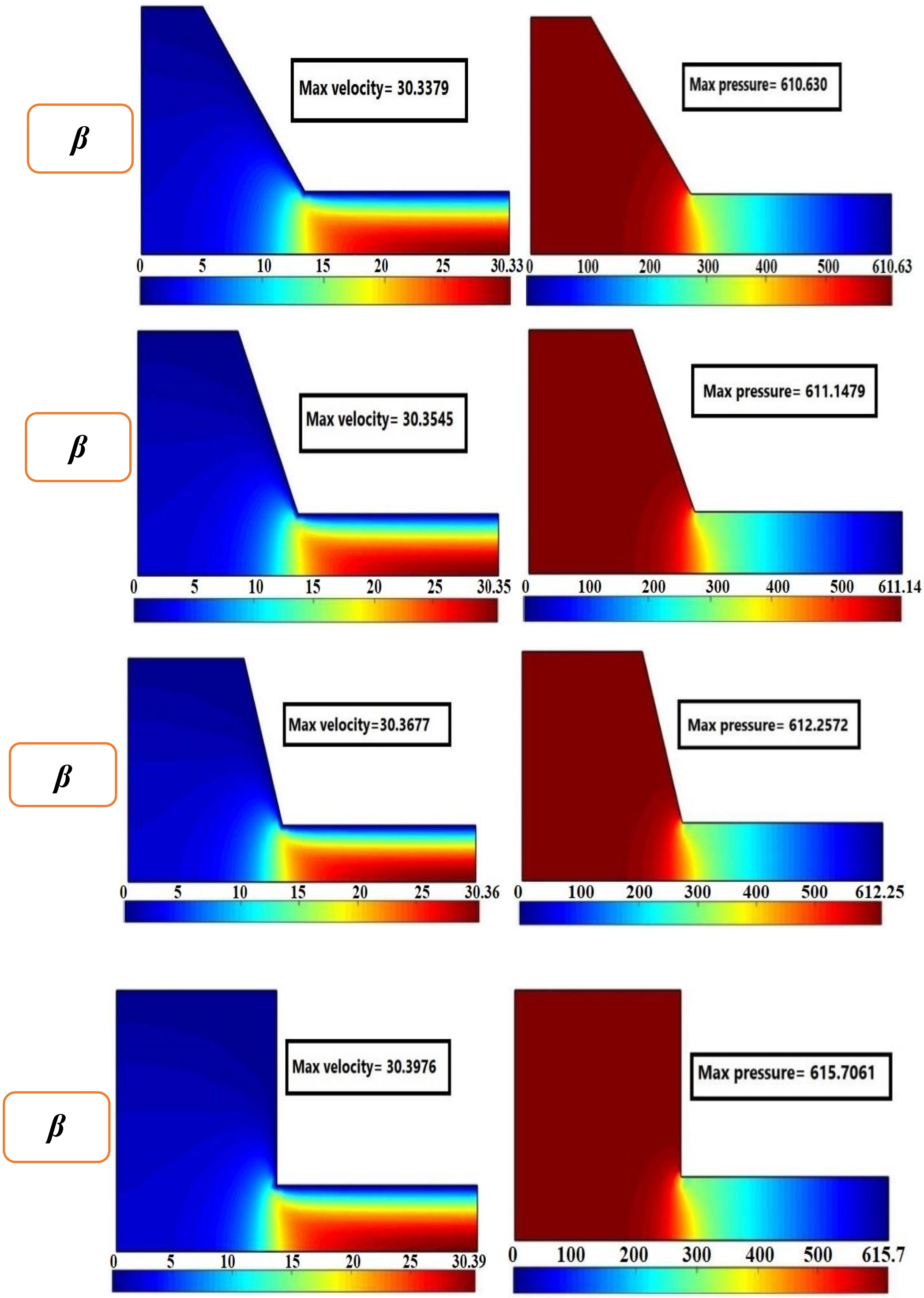


Figure 6: Maximum level of velocity and pressure under different (β)

In Figure 7, the maximum levels of velocity and pressure are plotted as functions of (β) with d_2=2 and I_2=10. The profiles in both cases reveal that, are is significant rise in the velocity and pressure as increased β from 21.8 to 40, while the increase is very modest for the rest of the β-values. That is reflect that the effect of β on the solution components is insignificant compared to that of d_2-variation and I_2-variation.

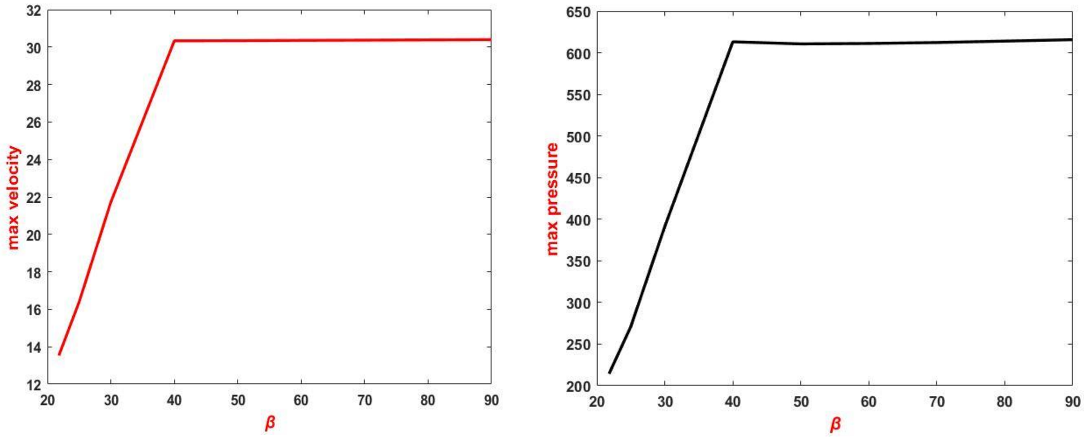


Figure 7 : Profiles of the level of velocity and pressure under different (β)

6. Conclusions

A numerical study for incompressible Newtonian fluid through a conical nozzle is achieved under the isothermal condition by using Taylor Galerkin Pressure Correction (TGPC) finite element. In this investigation, the effect of e geometric parameters such as the outlet diameter (d_2), Length of the cylinder (I_2) and conical angle (β) on the components of solution are presented. From the results, we deduced that there is a significant effect of the d_2 and I_2. There, we found that there is increasing in the levels of velocity and pressure as these parameters increase, which is in line with previous studies. In contrast, the effect of conical angle was also studied, where the impact from β-variation on the solution behaviour is modest.

Acknowledgements

The authors extend special and wide thanks to the department of mathematics in the collage of science, University of Basrah for their help.

Conflict of Interest

The authors declare that they have no conflicts of interest.

References

[1] T. Jiang, Z. Huang, J. Li, Y. Zhou, and C. Xiong, "Effect of nozzle geometry on the flow dynamics and resistance inside and outside the cone-straight nozzle," *American Chemical Society (ACS) Omega*, vol. 7, no. 11, pp. 9652-9665, 2022.

[2] J. Li, G. Li, Z. Huang, X. Song, R. Yang, and K. Peng, "The self-propelled force model of a multi-orifice nozzle for radial jet drilling," *Journal of Natural Gas Science and Engineering*, vol. 24, pp. 441-448, 2015.

[3] Y. Lu, S. Xiao, Z. Ge, Z. Zhou, Y. Ling, and L. Wang, "Experimental study on rock-breaking performance of water jets generated by self-rotatory bit and rock failure mechanism," *Powder Technol*, vol. 346, pp. 203-216, 2019.

[4] Z.-Y. Sun, G.-X. Li, C. Chen, Y. Yu, and G.-X. Gao, "Numerical investigation on effects of nozzle's geometric parameters on the flow and the cavitation characteristics within injector's nozzle for a high-pressure common-rail DI diesel engine," *Energy Convers. Manag*, vol. 89, pp. 843-861, 2015.

[5] Z. He, W. Guan, C. Wang, G. Guo, L. Zhang, and M. Gavaises, "Assessment of turbulence and cavitation models in prediction of vortex induced cavitating flow in fuel injector nozzles," *International Journal of Multiphase Flow*, vol. 157, pp. 104251, 2022.

[6] J. Wang, M. Petkovšek, H. Liu, B. Širok, and M. Dular, "Combined Numerical and Experimental Investigation of the Cavitation Erosion Process," *Journal of Fluids*

Engineering, *which is a publication of the American Society of Mechanical Engineers (ASME)*, vol. 137, no. 5, pp. 051302, 2015.

[7] F. Echouchene, H. Belmabrouk, L. Le Penven, and M. Buffat, "Numerical simulation of wall roughness effects in cavitating flow," *International Journal of Heat and Fluid Flow*, vol. 32, no. 5, pp. 1068–1075, 2011.

[8] R. Amirante, E. Distaso, and P. Tamburrano, "Experimental and numerical analysis of cavitation in hydraulic proportional directional valves," *Energy Convers. Manage*, vol. 87, pp. 208–219, 2014.

[9] C. Wang, L. Jiang, and H. Huo, "Numerical study of effects of geometric parameters on the flow and cavitation characteristics inside conical nozzle of autonomous underwater vehicles," *Advances in Mechanical Engineering*, vol. 11, no. 2, pp. 168781401982858, 2019.

[10] A. N. Abdulhasan and A. H. A.-M. Al-Muslimawi, "Numerical investigation of extensional flow through axisymmetric conical geometries: Finite element method," *Basrah Journal of Science*, vol. 38, pp. 399–421, 2020.

[11] M. F. Ahmed, A. Zaib, F. Ali, O. T. Bafakeeh, E. S. M. Tag-ElDin, K. Guedri, ... and M. I. Khan, "Numerical computation for gyrotactic microorganisms in MHD radiative Eyring–Powell nanomaterial flow by a static/moving wedge with Darcy–Forchheimer relation," *Micromachines*, vol. 13, no. 10, pp. 1768, 2022.

[12] F. Ali and N. Summayya, "Numerical simulation of Cattaneo–Christov double-diffusion theory with thermal radiation on MHD Eyring–Powell nanofluid towards a stagnation point," *International Journal of Ambient Energy*, vol. 43, no. 1, pp. 4939-4949, 2022.

[13] N. Thongjub and V. Ngamaramvaranggul, "Simulation of die-swell flow for Oldroyd-B model with feedback semi-implicit Taylor Galerkin finite element method," *Applied Science and Engineering Progress*, vol. 8, no. 1, pp. 55–63, 2015.

[14] I. Winnicki, J. Jasinski, and S. Pietrek, "New approach to the Lax–Wendroff modified differential equation for linear and nonlinear advection," *Numerical Methods for Partial Differential Equations*, vol. 35, no. 6, pp. 2275–2304, 2019.

[15] Y. Ren, Y. and D. Liu, "Pressure correction projection finite element method for the 2D/3D time-dependent thermomicropolar fluid problem," *Computers and Mathematics with Applications*, vol. 136, pp. 136–150, 2023.

[16] X. Li, J. Shen, and Z. Liu, "New SAV-pressure correction methods for the Navier-Stokes equations: stability and error analysis," *Mathematics of Computation*, vol. 91, no. 333, pp. 141–167,2022.

[17] N. Thongjub, "Feedback boundary of 4:1 rounded contraction slip flow for Oldroyd-B fluid by finite element," *Journal of Engineering Science and Technology*, vol. 13, no. 3, pp. 725–738, 2018.

[18] R. B. Khokhar, A. A. Bhutto, I. A. Bhutto, F. Sheikh, and M. A. Solangi, "Numerical analysis of Newtonian fluids in channel flows using semi-implicit time stepping Taylor-Galerkin/pressure method," *The Sciencetech*, vol. 4, no. 3, 2023.

[19] A. Khayyer, Y. Shimizu, T. Gotoh, and H. Gotoh, "Enhanced resolution of the continuity equation in explicit weakly compressible SPH simulations of incompressible free-surface fluid flows," *Applied Mathematical Modelling*, vol. 116, pp. 84-121, 2023.

[20] E. DiBenedetto and U. Gianazza, "Navier–Stokes equations," *In Partial Differential Equations*, pp. 591-656, Cham: Springer International Publishing, 2023.

[21] D. Chandran, A. Zampiron, A. Rouhi, M. K. Fu, D. Wine, B. Holloway, A.J, Smits, and I. Marusic, "Turbulent drag reduction by spanwise wall forcing. Part 2. High-Reynolds-number experiments," *Journal of Fluid Mechanics*, vol. 968, pp. A7, 2023.

[22] G. Kefayati, "Lattice Boltzmann simulation of cavity flows driven by shear and internal heat generation for both Newtonian and viscoplastic fluids," *Physics of Fluids*, vol. 35, no. 9, 2023.