25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-0650
THE INFLUENCE OF THE FLOW RATE ON THE JET IMPACT FORCE

Jonatas Emmanuel Borges
School of Food Engineering, Federal University of Mato Grosso, Barra do Garças, 78600-000, Brazil.
jonatasborges@ufmt.br

Sammy Cristopher Paredes Puelles
Elie Luis Martínez Padilla
School of Mechanical Engineering, Federal University of Uberlândia, Uberlândia, 38400-902, Brazil.
epadilla@ufu.br

Marcos Antonio de Souza Lourenço
School of Mechanical Engineering, Technologic Federal University of Parana, Cornélio Procópio, 86300-000, Brazil.
mlourenco@utfpr.edu.br

Abstract. *The present work aims to study a simplified case study of well drilling in the bottom hole, where the drill bit is represented by a sudden contraction, to determine the influence of the flow rate on the jet impact force which is an important factor in oil and gas well drilling operations. In the present case, the distance between the drill bit and the bottom hole, and the diameter of the drill bit was kept the same for all simulations. In order to solve this problem, the numerical platform in development has a parallel distributed-memory implementation to solve the Navier-Stokes equations. The Finite Volume Method is used in the spatial discretization where the diffusive terms approximated by the central differences method. The temporal discretization is accomplished using the Adams-Bashforth method. Both temporal and spatial discretizations are of second-order of accuracy. The Velocity-pressure coupling is done using the fractional-step method of two steps. The immersed boundary method (IB) is employed to represent the walls of the proposed problem while using a Cartesian mesh by adding a force term in the governing equations. The dynamic sub-grid scale model has been used to calculate the turbulent viscosity. The present results are compared with other published literature, where it was showed that the jet impact force and the peak pressure on the impacted surface increase with the increment of the flow rate.*

Keywords: *Computational fluid dynamics, immersed boundary method, jet impact force, oil well drilling.*

1. INTRODUCTION

The drilling of oil wells is one of the most expensive steps in oil exploration. It requires a high initial investment and it offers an uncertain economic viability. The need to reduce these costs has made drilling companies seek methods to achieve the aimed technological improvement. One of those ways is to increase the rate of penetration of the bit (Ifejaibeya, 2011).

One of the basic features of drilling fluid is to remove the drilled cutting from beneath the bit. The fluid pumped through the bit nozzles exist at high velocity and impacts the hole bottom. After impact, a crossflow develops that forces cuttings away from hole bottom and up the annulus. Penetration rate and overall drilling cost depend greatly on the cleaning action of the drilling fluid (Smalling and Key, 1979).

There is considerable uncertainty as to the best hydraulics parameter to use in characterizing the effect of hydraulics on penetration rate. Bit hydraulic horsepower, jet impact force and nozzle velocity all are used commonly (Bourgoyne *et al.*, 1986). Evaluation by hydraulic horsepower is based on the power expended as the fluid flows through the nozzles. The parameter optimized by the impact force method is the force produced by the momentum change after the fluid exists the nozzles and strikes the bottom. Jet velocity is based on the exit velocity from the nozzles.

The purpose of the jet nozzles is to improve the cleaning action of the drilling fluid at the bottom of the hole. While the cleaning action of the jet is not well-understood, several investigators have concluded that the cleaning action is maximized by maximizing the total hydraulic impact force of the jetted fluid against the hole bottom (Bourgoyne *et al.*, 1986). It has been shown that the peak impact pressure represents the hydraulic parameter that controls the effect that cutting removal has on rate of penetration (ROP). The peak impact pressure is proportional to the impact force on the hole bottom (Azar and Samuel, 2007).

Usually, parameters such as jet impact force, refer to properties of the fluid at the time it exits the jet nozzle. It is assumed that these parameters refer to properties also reflect the hydraulics energy at the bottom of the hole. The true impact force on the bottom of the hole is not calculated by the normal impact force equation used in most hydraulics model, because of the energy loss between the nozzle and the bottom of the hole. Empirical technique is used to estimate the true impact force for various jets and flow conditions (Azar and Samuel, 2007).

The exact determination of the jet impact force is an important factor in drilling operation, due to its influence on the rate of penetration (Figueiredo, 2014). Recently, numerical and experimental studies have been done focused on the understanding of the flow dynamics and the influence of process parameters on the impact force, such as, dynamic viscosity, volumetric flow rate, density, rotation and nozzle jet's diameter (Figueiredo *et al.*, 2013; Figueiredo, 2014; Santos, 2014; Daroz *et al.*, 2015).

The goal of the present work is to apply the immersed boundary method to simulate a Newtonian, incompressible and turbulent flow for a simplify case of well drilling in the bottom hole, where the drill bit is represented by a sudden contraction, solving the governing three-dimensional, partial differential equations in Cartersian meshes in order to obtain the true influence of the volumetric flow rate on the jet impact force.

2. MATHEMATICAL MODEL

The equations of mass conservation and momentum (also known as the Navier-Stokes equations) are used to model the cases simulated in this present work, considering an incompressible flow with a Newtonian fluid. These equations are given, in dimensional form and Cartesian coordinates, respectively as:

$$\frac{\partial \overline{u_j}}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial (\overline{u_i}\,\overline{u_j})}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}*}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu_\epsilon\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right)\right] + \frac{f_i}{\rho} \tag{2}$$

where, $\overline{u_i}$ is the filtered velocity component, $\overline{p}* = \overline{p} + \frac{2}{3}k\delta_{ij}$ is the modified pressure, k is the turbulence kinetic energy, ρ is the specific mass, f_i is the force component which represent any external force action on the fluid and $\nu_\epsilon = \nu + \nu_t$ is the effective viscosity that gather the terms of molecular viscosity and turbulent viscosity.

The dynamic sub-grid scale model has been used to calculate the turbulent viscosity (Germano *et al.*, 1991).

The immersed boundary method allows the specification of a particular boundary condition in the flow through the addition of a source term f_i in the momentum equations (Eq.2). This source term is calculate in the Lagrangian domain and transmitted to the Eulerian domain to account for the presence of the boundary walls (geometry). The term f_i is null in all Eulerian grid, except in volumes neighbor to the Lagrangian markers. Mathematically the source term can be represent as:

$$\vec{f}(\vec{x},t) = \int_\Gamma \vec{F}(\vec{x}_k,t)\delta(\vec{x}-\vec{x}_k)d\vec{x}_k \tag{3}$$

where $\delta(x)$ is the auxiliary Dirac delta function, k denotes a Lagrangian variable and $\vec{F}(\vec{x}_k,t)$ is the Lagrangian force, which is determined in the points of the object interface.

3. PROBLEM DESCRIPTION

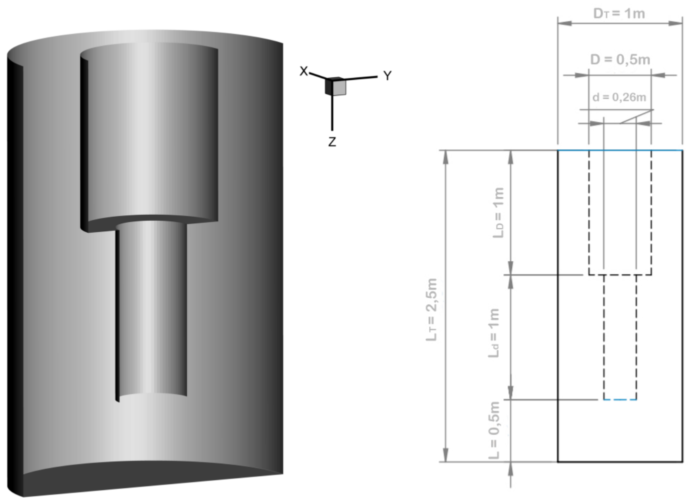
The problem consists in a numerically investigation of Newtonian turbulent flow in a three-dimensional domain. The geometry is composed of two domains, where the inner domain is a sudden contraction. The outer domain is composed of the hole bottom and the annulus, where the drilling fluid returns to the surface. The geometry configuration is illustrated in Fig. 1. The Cartesian coordinates system *(x,y,z)* is adopted for the Eulerian domain. To minimize the Eulerian domain size which is $1,1\times1,1\times2,86$m in x, y and z direction respectively, the inlet velocity profile for turbulent flow is approximated by the empirical *power-law* equation (Fox *et al.*, 2011), showing a fully developed behaviour in the upstream region with a bulk velocity U. No rotation was imposed in the inner domain.

In the inner domain, the upstream pipe has an inner diameter of $D = 0.50$m and a length of $L_D = 1.00$m, and the downstream section is a pipe with a diameter $d = 0.26$m, and a length of $L_d = 1,00$m. In the outer domain the pipe has an inner diameter of $D_T = 1.0$m and a length of $L_T = 2.5$m. The length between the nozzle and the bottom of the hole is $L = 0.5$m. The unsteady convective boundary condition is applied at the outlet boundary, where the global *convective* velocity out of the boundary surface, and its magnitude is determined, at any instant, by the requirement of the global mass conservation. The grid density of $100\times100\times260$ was chosen for all simulations.

Important parameters related to the fluid flow through a contraction are: the contraction ratio, $\beta = D/d$ and upstream Reynolds number, $Re_D = 4Q/\pi D\nu$ and downstream Reynolds number, $Re_d = 4Q/\pi d\nu$. The contraction ratio adopted in this present work is $\beta = 1.92$ based on the works of Figueiredo *et al.* (2013) and Figueiredo (2014) . The density of the fluid $\rho = 1200.0$kg/m^3, the kinematic viscosity $\nu = 5 \times 10^{-5}$ m^2/s and the flow rate was varied to set the Reynolds number.

Based on mass conservation, in control volume form, the relation between the Reynolds number upstream and downstream are defined as:

$$Re_d = Re_D \beta \quad (4)$$



(a) (b)

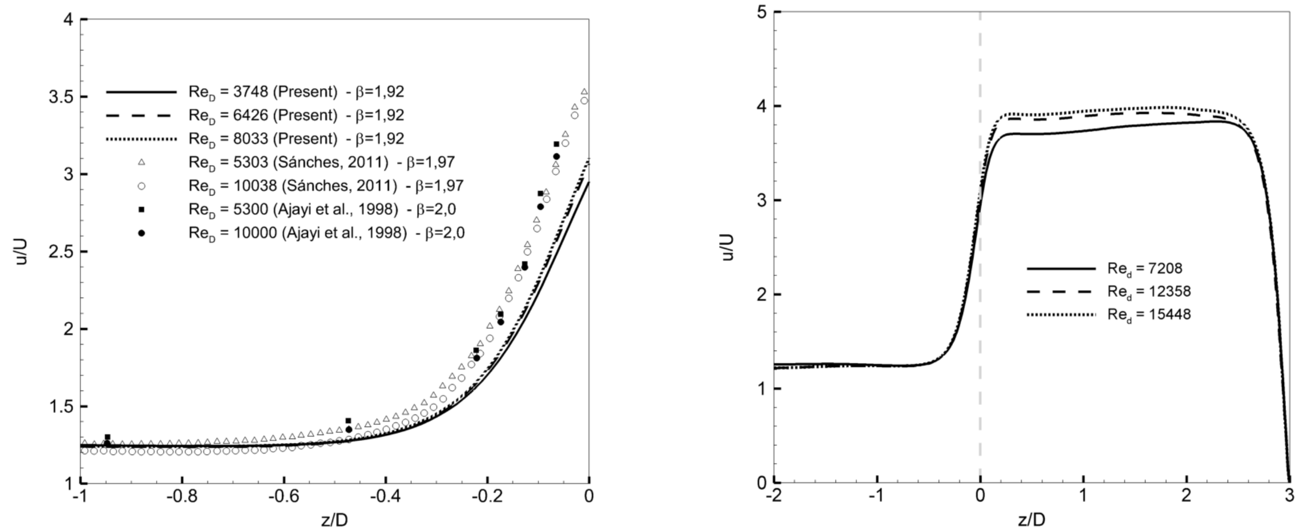
Figure 1: Schematic representation of the problem: (a) cross section view and (b) dimensions.

4. RESULTS AND DISCUSSION

In this work, the influence of the flow rate on jet impact force are analysed at various Reynolds number in a range in the range $3478 \leq Re_D \leq 8033$ for the upstream tube and $7208 \leq Re_d \leq 15448$ for the downstream tube.

In the inner domain, the present results of the mean axial velocity , u, adimensionalized with the upstream bulk velocity U are compared to other works which have higher contraction ratio and slightly different Reynolds number (upstream region). A fully developed flow region is achieved in the upstream region, far from the influence of the sudden contraction, then under the influence of the sudden contraction the flow adapt to pass through the contraction. A stationary flow vortex is present in the corner of the upstream tube just before the contraction plane. This reduces the available flow area forcing the fluid to adapt. The upstream influence of sudden contraction is limited to a region smaller than 0.6D. This is in agreement with the work by (Sánchez, 2011). The difference results between the present work and the references (Sánchez, 2011; Ajayi *et al.*, 1998) relies on the contraction ratio and Reynolds number difference, once the present work has a lower contraction ratio it is expected lower velocity (Fig. 2a). Same tendency is observed between the profiles.

After the contraction place, there is a region of minimum pressure and maximum velocity, this region is known as the *vena contracta*. The contracted flowing stream is surrounded by fluid that is in a state of turbulence but has very little forward motion. Downstrem of the *vena contracta* the flow stream expands, the velocity decreases and the pressure rises (Pienaar, 2004). As the flow rate increases the velocity increases as well, as can be seen on a plane containing the centreline (Fig. 2b). The velocity goes to zero when reaching the bottom hole at ($z/D = 3$).

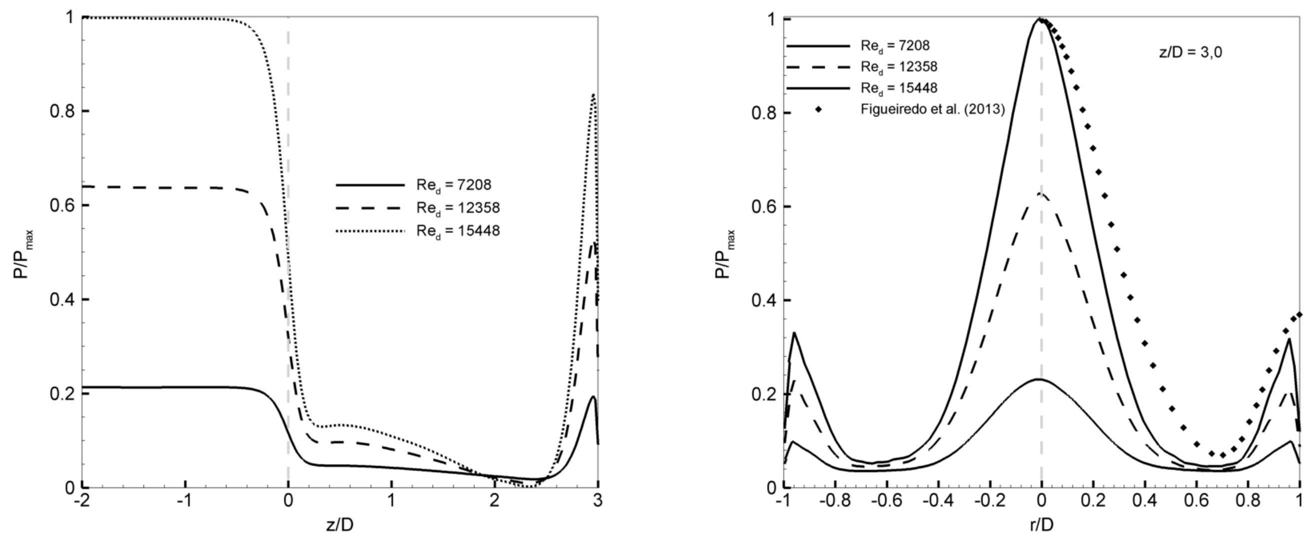


(a) (b)

Figure 2: Axial velocity through the centreline: (a) upstream and (b) whole domain.

Initially, the pressure drops linearly with z/D, as the fluid approaches the sudden contraction, there is a marked drop pressure. At the beginning of the downstream pipe, close to the wall, there is a region of low pressure, due to the *vena contracta* presence. Downstream of the *vena contracta* the pressure drops linearly with higher slope. In the outer domain, when the jet hits the bottom hole, a stagnation pressure profile forms at the surface (Fig. 3a). As can see observed, the peak pressure increases with the increment of the flow rate.

The pressure on the impacted surface has a higher pressure in the center of the bottom hole due to the jet impact, then the pressure decreases with the radius, finally, close to the wall the pressure increases again with low intensity. After the fluid impacts the bottom hole, it returns through the annulus where a recirculation takes place, between the nozzle exit and the bottom hole. Then, the fluid goes up to the surface by the annulus. The profiles on the impacted surface have the same tendency of Figueiredo *et al.* (2013) where the flow rate and the distance between the nozzle and the bottom hole are higher than the present study, it is expected to have higher pressure, as can be seen in Fig. 3b.



(a) (b)

Figure 3: Pressure through the centreline: (a) z/D at $r/D = 0$ and (b) r/D at $z/D = 3$.

Figure 4 show the pressure contour plot on a plane containing the centreline at various Reynolds numbers for the whole domain. From the plane, a elevation plot was performed enable to visualize a 3D view of the pressure field. As the flow rate increases, the pressure values increase as well for the whole domain, both positive and negative values. Similar pressure contour plot was obtained by Figueiredo *et al.* (2013).

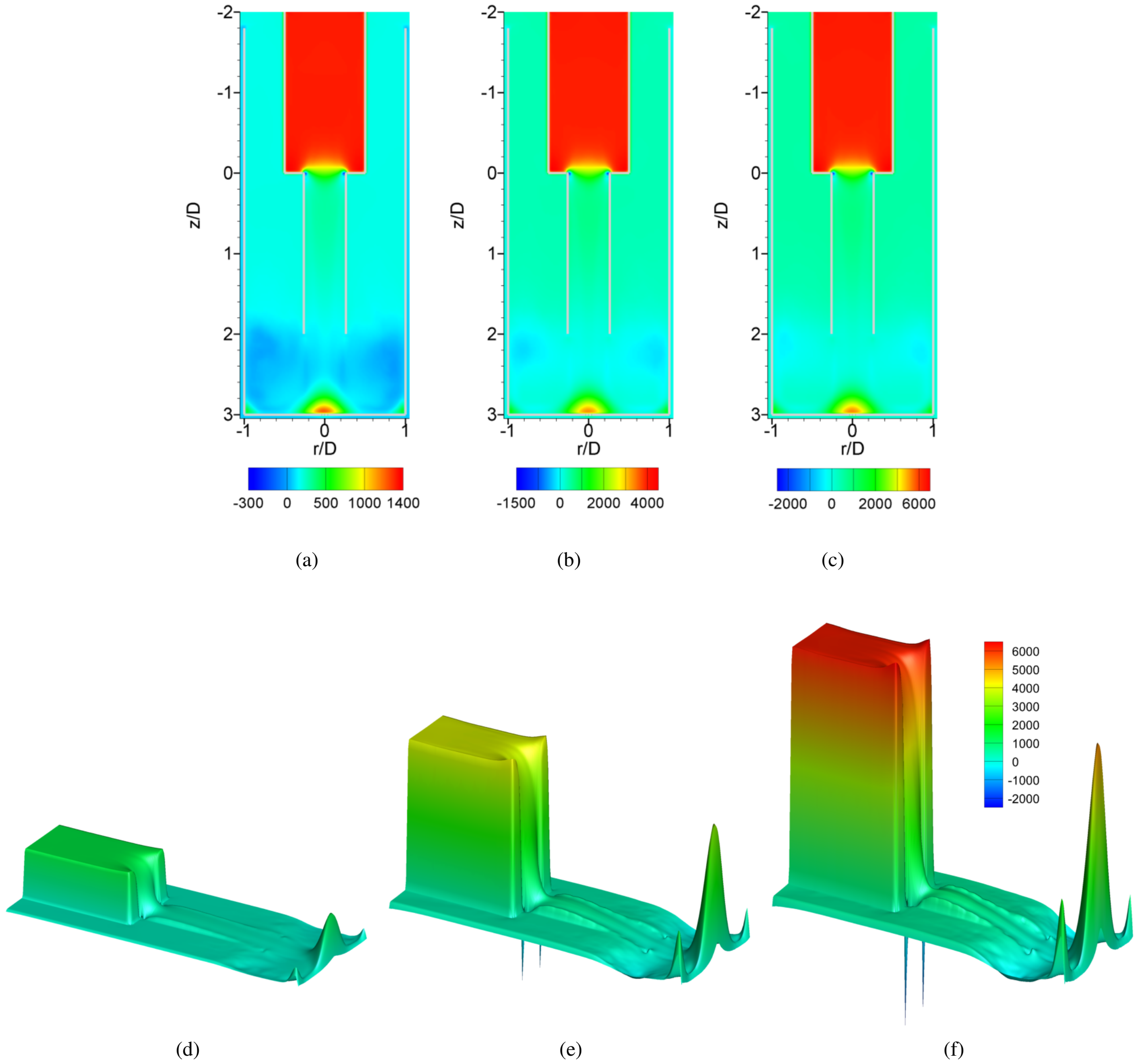


Figure 4: Pressure field at: (a) 2D-$Re_d = 7208$, (b) 2D-$Re_d = 12358$, (c) 2D-$Re_d = 15448$, (d) 3D-$Re_d = 7208$, (e) 3D-$Re_d = 12358$ and (f) 3D-$Re_d = 15448$.

4.1 Jet impact force validation

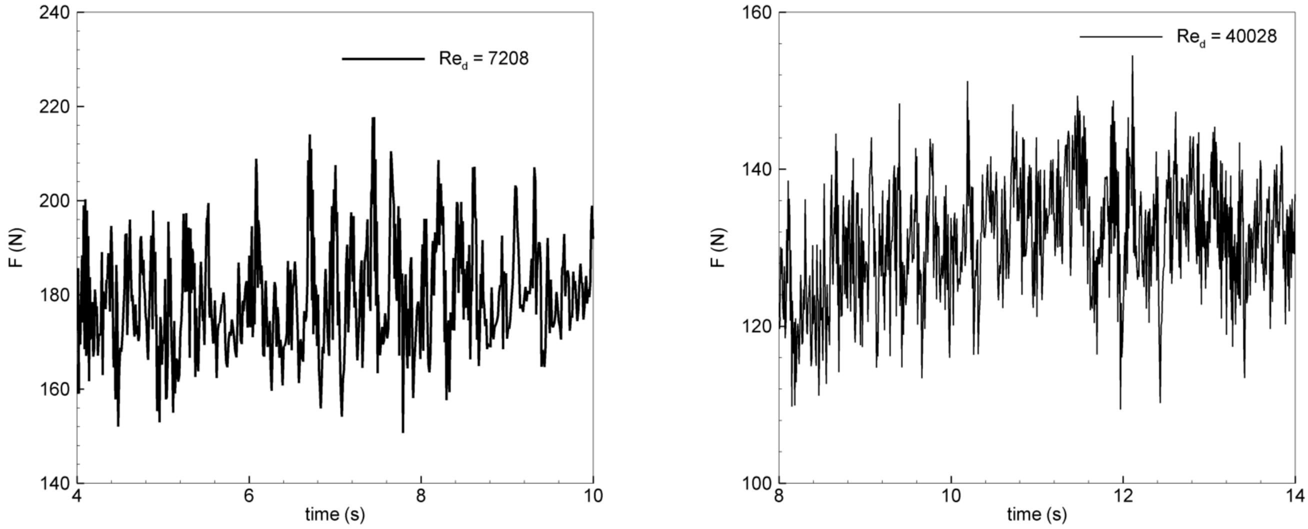
When the immersed boundary method is applied, the jet impact force can be easily obtained, by the sum of all Lagrangian forces which represent the hole bottom. This procedure allows to obtain the jet impact force every single time step (Uhlmann, 2005). The expression to obtain the Lagrangian force is written as:

$$F_{fp} = \sum_{ki=1}^{np} F_{ki} = \sum_{ki=1}^{np} \rho \left(\frac{u_{i\,IBM} - u_{ki}^*}{\Delta t} \right) \tag{5}$$

where F_{fp} is impact force, np is the number of Lagrangian markers which represents the hole bottom, $u_{i\,IBM}$ is the immersed body velocity, which is in this work, $u_{i\,IBM} = 0$, once the surface is static (no rotation of the inner domain).

The numerical result of Figueiredo *et al.* (2013) and experimental result of Santos (2014) were compared to the present results. Due to the turbulent flow, there is fluctuation in the jet impact force, which is not shown in those references.

The steady state jet impact force throughout the time is shown in Fig. 5. It is noteworthy that the physical properties between the reference works are different, influencing the magnitude of the jet impact force, directly.



(a) (b)

Figure 5: Jet impact force throughout the time: (a) $Re_d = 7208$ and (b)(a) $Re_d = 40028$.

Table 2 presents a comparison between present results and published literature of the jet impact force. There was good agreement when comparing the jet impact force with the result of Figueiredo *et al.* (2013) with has the same contraction rate and Reynolds Number. Santos (2014) has higher contraction ratio in the inner domain. As shown in Fig. 2a, as β increases the velocity increases as well, reflecting on the jet impact force. In other to deal with this slight difference and taking advantage of the expression proposed by Figueiredo (2014) to evaluate the relation between the jet diameter and the jet impact force, defined as:

$$F_{fp} = Cd^{-1.9} \tag{6}$$

where C is a constant and d is the jet diameter. Using the definition of contraction ratio, β, the Eq. 6 can be updated to take into account the contraction ratio. Thus a new expression is defined as:

$$F_{fp} = C\left(\frac{D}{\beta}\right)^{-1,9} \tag{7}$$

From Eq. 7, the result of Santos (2014) was modified to obtain the jet impact force for the present contraction ratio, $\beta = 1.92$. The percentage error of the jet impact force has maximum error within 1.4%

Table 1: Comparison of the jet impact force between the present results and literature.

ρ (kg/m^3)	Re_d	F_{fp} (N) Present	F_{fp} (N) Reference	$\beta = \frac{D}{d}$	F_{fp} (N) (Eq. 7) Reference modified	Error (%)
1200.00	7208	179.7 ± 11.6	177.2 Figueiredo (2014)	1.92	177.2	1.40
998.28	40028	132.1 ± 7.2	144.5 Santos (2014)	2.00	133.7	1.24

where the percentage error is defined as:

$$error(\%) = |\frac{(F_{fp})_{PRESENT} - (f_{fp})_{REF}}{(F_{fp})_{PRESENT}}| \times 100 \tag{8}$$

4.2 Influence of the flow rate on the jet impact force

The flow rate was varied to set the Reynolds number, keeping the geometry and properties constants. As Re increases, the force increases, as well. Figueiredo *et al.* (2013) reported a quadratic tendency between the volumetric flow rate and the impact force, which may be confirmed with more simulations to be carried out.

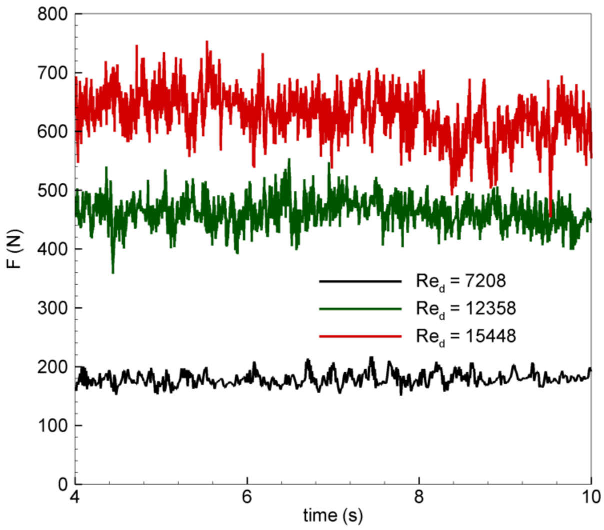


Figure 6: Jet impact force throughout the time at various Reynolds numbers.

The average and standard deviation of the jet impact force at steady state for the three flow rates evaluated is presented in Tab. 2.

Table 2: Jet impact force at various Reynolds numbers.

Re_d	F_{fp} (N)
7208	179.7 ± 11.6
12358	463.3 ± 23.7
15448	629.1 ± 39

5. CONCLUSIONS

In this paper, a numerical investigation of Newtonian turbulent flow in a simplified problem of well drilling in the bottom hole where the drill bit is represented by a sudden contraction. A structured grid in Cartesian coordinate was employed for the Eulerian domain. The IB method was able to represent the geometry of the problem, where the jet impact force was obtained by the Lagrangian force, make it easily calculated. The results of the simulations were compared with other publishing literature, showing good agreement between them. The jet impact force and the peak pressure increase with the increment of the flow rate.

6. ACKNOWLEDGEMENTS

The authors wish to thank the Federal University of Mato Grosso (UFMT), Federal University of Uberlândia (UFU), Technologic Federal Univeristy of Parana (UFTPR), Petrobras, FAPEMIG and CAPES for their support.

7. REFERENCES

Ajayi, K., Papadopoulos, G. and Durst, F., 1998. “Turbulent flow past a sudden contraction in a pipe”. *ASME Fluids Engineering Division Summer Meeting*.

Azar, J.J. and Samuel, R., 2007. *Drilling Engineering*. PennWell Books, 1st edition.

Bourgoyne, A.T., Chenevert, M.E., Millheim, K.K. and Young, F.S., 1986. *Applied drilling engineering*. Society of Petroleum Engineers, 1st edition.

Daroz, V., Maneira, E.L. and Franco, A.T., 2015. "Investigação numérica da circulação direta e reversa no processo de perfuração de poços de petróleo". *VI Encontro Nacional de Hidráulica de Poços de Petróleo e Gás*.

Figueiredo, L.M., 2014. "Investigação numérica da força de impacto e do coeficiente de descarga em bocais ejetores de brocas de perfuração." Monografia (Engenharia Mecânica), Technologic Federal University of Parana, Curitiba, Brasil.

Figueiredo, L.M., das Neves, D.S., Silva, L.C.K., Franco, A.T., NegrÃčo, C.O.R., Morales, R.E.M., Waldmann, A.T. and Martins, A.L., 2013. "Investigação da força de impacto e do coeficiente de descarga em bocais ejetores de brocas de perfuração". *V Encontro Nacional de Hidráulica de Poços de Petróleo e Gás*.

Fox, R.W., McDonald, A.T. and Pritchard, P.J., 2011. *Introduction to Fluid Mechanics*. 8th edition. ISBN 9780470547557.

Germano, M., Piomelli, U., Moin, P. and Cabot, W.H., 1991. "A dynamic subgrid-scale eddy viscosity model". *Physics of Fluids A: Fluid Dynamics*, Vol. 3, No. 7, pp. 1760–1765.

Ifejaibeya, W.N., 2011. *Estudo da influência da taxa de penetraçâo de broca de perfuraçâo nos custos da construçâo de poços de petróleo*. Master's thesis, Faculdade de Engenharia Mecânica,, Campinas.

Pienaar, V., 2004. *Viscous flow through sudden contractions*. Ph.D. thesis, Faculty of Engineering Cape Technikon.

Sánchez, F.M.P., 2011. *Estudo Experimental do escoamento de fluido newtoniano em contração abrupta axissimétrica com a técnica de velocimetria por imagem de partícula*. Ph.D. thesis, UTFPR, Universidade Tecnológica Federal do Paraná, Curitiba, PR, Brasil.

Santos, V.T.S., 2014. "Estudo experimental da força de impacto e do coeficiente de descarga de bocais ejetores utilizados na perfuração de poços de petróleo." Monografia (Engenharia Mecânica),Technologic Federal University of Parana, Curitiba, Brasil.

Smalling, D.A. and Key, T.A., 1979. "Optimization of jet-bit hydraulics using impact pressure". *Society of Petroleum Engineers*, pp. 23–26.

Uhlmann, M., 2005. "An immersed boundary method with direct forcing for the simulation of particulate flows". *J. Comput. Phys.*, Vol. 209, pp. 448–476. ISSN 0021-9991. doi:http://dx.doi.org/10.1016/j.jcp.2005.03.017. URL `http://dx.doi.org/10.1016/j.jcp.2005.03.017`.

8. RESPONSIBILITY NOTICE

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