ALGERIAN PEOPLE'S DEMOCRATIC REPUBLIC

Ministry of Higher Education and Scientific Research

Serial Number: …/2023

Kasdi Merbah Ouargla University

Faculty of Hydrocarbons, Renewable Energies, Earth, and Universe Sciences.
Department of Hydrocarbon Drilling Oil Mechanics.

THESIS

To obtain the Master's Degree

Option: Drilling

Presented by: Boussaha Mohamed Lamine and Bettayeb Abdelaziz

-Title-

Study of forced turbulent flow of a non-Newtonian fluid in convergent channels

Publicly defended on: 20/06/2023

Before the Jury:

President:	Bouhadda Mebarka	- MCB	- Kasdi Merbah Ouargla University
Examiner:	Chaich Zineb	- MCB	Kasdi Merbah Ouargla University
Supervisor:	Hadjadj Souad	- MCB	Kasdi Merbah Ouargla University

The academic Year 2022/2023

الملخص

تحلل هذه الدراسة الرقمية سلوك التدفق الدائم، غير القابل للضغط، القسري والمضطرب في القنوات المتقاربة، في الوضعيات الأفقية والعمودية. السائل المستعمل غير نيوتوني يخضع لنموذج قانون الاستطاعة. تم حل هذه الاشكالية ثنائية الأبعاد بواسطة برنامج انزيس فلونت 19.2 . الهدف من هذه الدراسة هو فهم السلوك الديناميكي والحراري مع تغيرعدد رينولدز وتقديم رؤى للتطبيقات العملية في هندسة البترول.

كشف تحليل منحنى السرعة عن التناسب بين عدد رينولدز والسرعة عند مخرج القنوات. أما التبادل الحراري بين السائل والجدار يمكن اهمالها تحت تأثير الحمل القسري. أين يحافظ السائل على درجة حرارته من المدخل إلى مخرج القناة

الكلمات المفتاحية: القناة المتقاربة ، سائل غير النيوتوني ، نموذج قانون الاستطاعة ، الحمل القسري ، التدفق المضطرب.

Abstract

this study analyzes numerically the behavior of permanent, incompressible, forced, and turbulent flow in convergent ducts, in horizontal and vertical configurations. The fluid uses is non-Newtonian and obeys to power law model. the resolution of this bi-dimensional problem was carried out by Ansys Fluent 19.2. The objective was to understand the dynamic and thermal behavior at different Reynolds numbers and provide insights for practical applications in petroleum engineering.

The analysis of velocity profiles revealed proportionality among the Reynolds number and velocity at the outlet of the ducts. The heat exchange between the fluid and the wall is neglected, influenced by forced convection. The fluid keeps its temperature from the inlet to the outlet of the channel

Keywords: convergent channel, non-Newtonian fluid, power law model, forced convection, turbulent flow

ACKNOWLEDGEMENT

First and foremost, we thank God for the strength and patience He has given us to overcome all difficulties.

We express our utmost gratitude to our dear professor Hadjadj Souad for her dedication to the success of this work

Next, we extend our thanks to the members of the examination jury, as well as the administrative staff, for their kindness. We also thank the engineers, technicians,. We must not forget to express our gratitude to the director who provided us with the opportunity to undertake the internship, bridging the gap between the theoretical knowledge acquired at university and the practical application in the company, as well as working on the subject of the final thesis.

Once again, a big thank you to everyone.

Dedication:

I have the great honor to dedicate this modest work:

To myself, as a culmination of all the sleepless nights and sacrifices throughout my journey.

To the remarkable woman who has endured so much to shape me into who I am, who gives me hope to live, and the strength to pursue my dreams and succeed. To the one woman in my life, my beloved mother

To my dear father for his sacrifices in making my wishes come true, and for all his prayers that have provided me with support and encouragement.

To my dear sisters and my dear brother, who have always supported me financially and emotionally at every moment, without expecting anything in return.

To all those who have wished me success and happiness

Bettayeb abdelaziz

Dedication:

I dedicate this modest work to:

To my dear parents, as a testament to my gratitude for their patience, sacrifices, and support throughout my studies. Thank you, dear Mother, thank you, Father.

To my dear sister and brother, of whom had been a source of inspiration and courage for me.

To all my friends, with whom I have shared my best moments.

To all my teachers since my early years of study.

To everyone who has known me, whether closely or from afar.

MOHAMED LAMINE BOUSSAHA

V	Velocity	[ms^{-1}]
C_p	Specific heat at constant pressure	[J/Kg. K]
D	Diameter	[m]
F	FORCE	[N]
G	Earth acceleration	[ms^{-2}]
H	Height	[m]
n	Behavior Index	
u,v	Velocity field components	[ms^{-1}]
V	Fluid velocity	[ms^{-1}]
X,y	Cartesian Coordinates	[m]
$\dot{\gamma}$	Shear rate	[kg.s.$n^{-1}m^{-1}$]
T_0	Minimum yield point	[kg.s.$n^{-1}m^{-1}$]
Re	Reynolds	
RMS	Reynolds stress model	
dV	Velocity difference between two adjacent	[m.s^{-1}]
dr	Distence between the two layers	[m]
μ_a	shear stress	[kg.s.$n^{-1}m^{-1}$]
K	consistency index	
F	friction factor	
L	length of section	[m]
$\bar{U}$	statistical average	[m]
$\bar{v}$	Fluctuations or deviations from these averages.	[m]
Γ_T	Turbulent diffusivity.	
ρ	Density	[kg/m3]
λ	Thermal conductivity	[W/m.k]
BHA	Bottom Hole Assembly	
CFD	Computational dynamic fluids	

A convergent channel is a particular duct; view its variable diameter. The narrowing of this geometric parameter accelerates the flow and settles many anomalies when using pipes with a constant diameter, like cylinders. This pipe plays a crucial role in various applications within the oil drilling rig industry. like nozzles, valves, and mudding guns. its existence in several lines as kill and chock lines, cementing lines, sampling lines, ...

This work investigates, numerically, permanent, incompressible, forced and turbulent flow of drilling mud, considered as pseudoplastic fluid obeys the power law, in a convergent pipe.

The objective of this study is to understand the hydrodynamic and thermal behaviour of this non-Newtonian fluid when it crosses a narrowing channel.

This bidimensional problem is managed by six differential equations. solved by the Ansys Fluent 19.2 code. using the finite volume method.

This study began with a comprehensive literature review to establish the theoretical background and previous studies related to flow in convergent ducts. the geometric, mathematical, and numerical modeling carried out in the second chapter. Last chapter contains the analysis of the results, considering different Reynolds numbers, provided valuable information regarding the velocity distribution and temperature profiles.

CHAPTER I: GENERALITY

I.1. Rheology:

Rheology is the study of the deformation of fluids. Its importance is recognized in the analysis of fluid-flow velocity profiles, fluid viscosity, friction pressure losses, ECD, and annular hole cleaning. It is the basis for all analyses of wellbore hydraulics

I.2. Fluid-flow velocity:

Figure I.1 shows the velocity profile of a fluid between two parallel plates, the lower of which is stationary while the upper is moving at a velocity.

The layer of fluid adjacent to the bottom plate has zero velocity, while the top layer has a velocity equal to that of the moving upper plate. The rate at which each layer moves past the other is termed the shear rate[1]

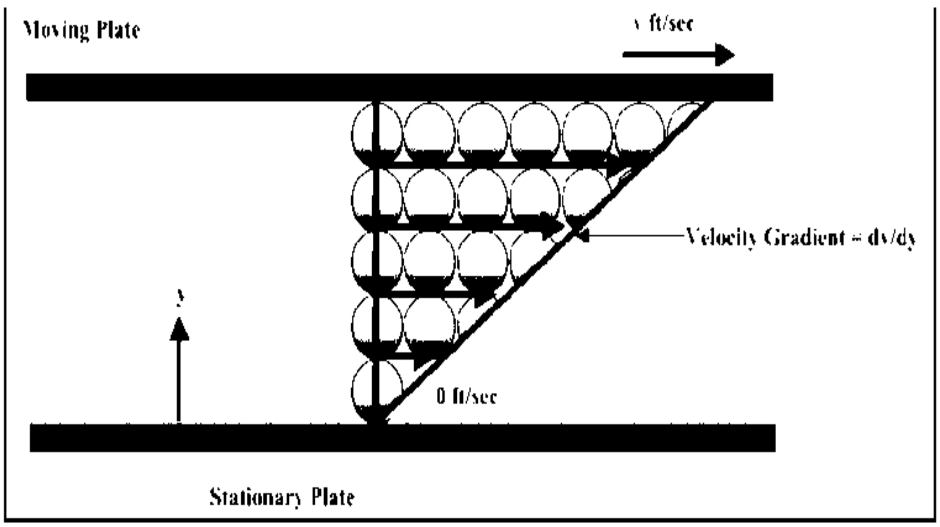


Fig.I.1: Velocity gradient between two parallel plates[1]

I.3. Rheological characteristics:

At a given temperature and pressure, fluids are characterized by:

- Their behavior under transient conditions, as manifested by their response time to changed conditions of flow.

- Their behavior in laminar flow, characterized by their experimental flow curve, or rheogram. The constant coefficients of the equation of flow represented by this curve are rheological parameters, specific to the particular fluid. [1]
- If the flow is laminar, the equation of flow relates the shear stress: with the shear rate y. For a given fluid, it varies with temperature and pressure. We have said that in laminar flow the fluid is sheared into laminar layers, parallel to the direction of flow, each layer moving at its specific velocity.[1] We may accordingly define: A shear rate such that:

$$\dot{\gamma} = \frac{dV}{dr} = \frac{Velocity\ difference\ between\ two\ adjacent\ layers}{distance\ between\ the\ two\ layers} \tag{I.1}$$

The dimensional equation of y is L:

$$\frac{LT^{-1}}{L} = T^{-1} \tag{I.2}$$

The dimension $\dot{\gamma}$ is an inverse time (s^{-1} or l/s). 1 (b) A shear stress, which is the force per unit area of the laminar layer inducing the shear. The dimensional equation of τ is:

$$\frac{MLT^{-2}}{L^{-2}} = ML^{-1}T^{-2} \tag{I.3}$$

τ has the dimensions of pressure. It is often expressed in Ib/100 ftz (lb force/100 ft2) or, in the International System of Units (SI) in pascal (Pa). For a given shear rate, the apparent viscosity μ_a is defined by the equation:

$$\mu_a = \frac{\tau}{\dot{\gamma}} \tag{I.4}$$

Where τ is the shear stress leading to $\dot{\gamma}$

The dimensional equation of μ_a is:

$$\frac{ML^{-1}T^{-2}}{T^{-1}} = ML^{-1}T^{-1} \quad (I.5)$$

μ_a has the dimensions of viscosity. (2) In the SI system, μ_a is expressed in pascal-second (Pa . s). The unit which is usually employed is its sub-multiple - the millipascal-second (mPa. s). It is equal to the centipoise (cp). It is often necessary to consider the shear stress, shear rate and apparent viscosity at the wall of the channel where the fluid is moving. C. Their behavior at rest, as manifested by gel formation after a certain period of time, for thixotropic fluids. A fluid is thixotropic if: (a) It forms a gel after being shaken and left to stand. (b) It returns to its original condition after it has been shaken again. At constant temperature and pressure, thixotropic behavior is reversible.[2]

I.4. Pseudo plastic or power law fluids:

I.4.1. Definition and typical curve:

Pseudo-plastic fluids, like Newtonian fluids, will flow under any applied stress, however small. But, as distinct from Newtonian fluids, the shear stress is not proportional to the shear rate, but to its n^{th} power; hence the name power-law fluids.[2] The equation of flow is:

$$\tau = K\dot{\gamma}^{n} \quad (I.6)$$

Where K is the consistency index in Pa • s^n or in lb. $s^n/100ft^2$, and n is the dimensionless flow behavior index, which is unity or smaller than unity. If n = 1, the equation becomes identical with the equation off low of a Newtonian fluid having the viscosity K. The following graphs shown in Figure I.2 are flow curves of power-law fluid in Cartesian and logarithmic coordinates respectively

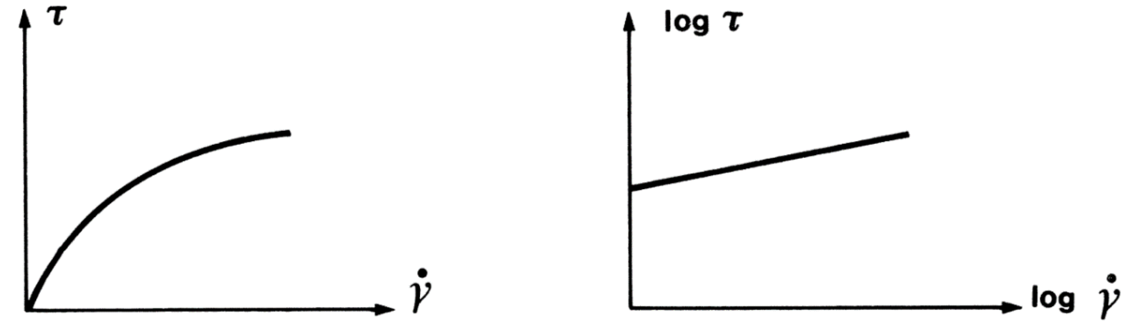


Fig.I.2: **Power-law fluid flow curve.[2]**

I.4.2 Fluid consistency index and flow behavior index:

In logarithmic coordinates, the flow curve is a straight line, the equation of which is

$$y = \log K + nx \quad \text{(I.7)}$$

Where:

$$y = \log \tau$$

$$x = \log \dot{\gamma}$$

Thus, the flow behavior index n represents the slope of this line, while the fluid consistency index K is given by the intersection of the flow curve with the axis at $\dot{\gamma}$ = 1:

$$n = \frac{\log \tau - log\tau'}{\log \dot{\gamma} - \log \dot{\gamma}\prime} = \frac{\log \tau/\tau\prime}{\log \dot{\gamma}/\dot{\gamma}\prime} \quad \text{(I.8)}$$

I.4.3 Determination of flow behavior index n and consistency index K in a Fann viscometer

The determinations made in a six-speed Fann viscometer (or, if this instrument is not available, in a two-speed Fann viscometer, using also g_0, which is considered to represent a determination at 3 rpm) are plotted, as a rheogram, on log-log paper, shear rates (in s^{-1}) being plotted on the abscissa, shear stresses (in Ib/l00 ft^2) on the ordinate

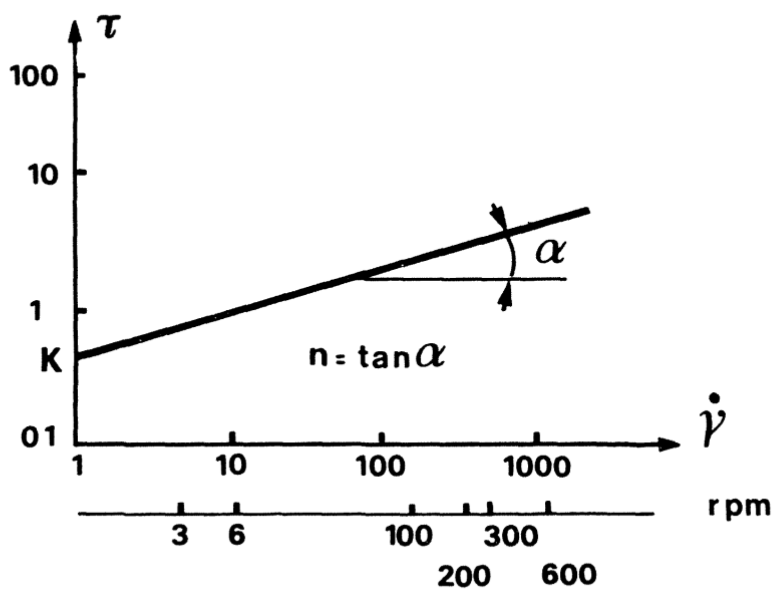


Fig.I.3: Determination of power-law fluid rheological parameters.[2]

- ✓ **Determination of n.**

We have seen that (view fig.I.3)

$$n = \frac{\log \tau/\tau\prime}{\log \dot{\gamma}/\dot{\gamma}\prime} \tag{I.9}$$

If $\dot{\gamma} = 2\gamma'$, we have

$$n = \frac{\log(\frac{\tau}{\tau'})}{\log 2} = \frac{\log(\frac{\emptyset}{\emptyset'})}{\log 2} = 3.32\log(\frac{\emptyset}{\emptyset'}) \tag{I.10}$$

- ✓ **Determination of K :**

$$K = \frac{\tau}{\dot{\gamma}^n} \tag{I.11}$$

If $\dot{\gamma}$=1, K=τ_1

If □ is given in Ib/100ft^2 andγ in s^{-1}, the unit of K will be Ib • s^n/100ft^2. If τ is given in Pascalthe unit of K will be Pa .S^n. It will be recalled that 1 Ib$_{force}$/100ft^2 = 0.478964 Pa.[2]

I.5. Turbulent flow:

Unlike laminar flow, the analysis of fluid-flow pressure losses in a pipe in the turbulent regime is largely empirical. The random shearing and intermixing motion of the fluid particles makes orderly mathematical analysis nearly impossible. Pressure losses in turbulent flow are calculated from the Fanning equation, defined for any fluid model by

$$\Delta P = \frac{2f\ .L\rho\ .\overline{V}^2}{D} \tag{1.12}$$

ΔP = pressure loss

f = friction factor

L = length of section

ρ = fluid density

V = average velocity

D = pipe diameter

The Fanning equation is empirically derived, and, like the Reynolds number, it attempts to quantify the basic factors affecting flow. The parameter f, called the Fanning friction factor, is a function of the Reynolds number and of the surface conditions of the pipe wall. The surface condition of a pipe wall is given by the relative roughness parameter ~. E, or the absolute roughness, is the average depth of the pipe wall irregularities. D is the inside pipe diameter. The parameters are illustrated in Figure I.4. Notice that the smoother the pipe, the lower the value ~. As one would expect, lower values of relative roughness are reflected by lower friction factor f which, in turn, results in lower pressure losses.[2]

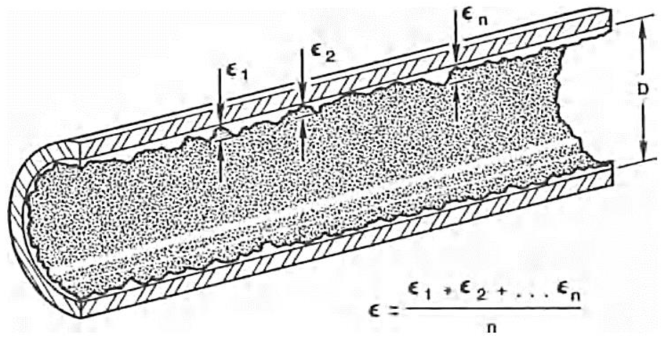


Fig.I.4: Relative Roughness of a pipe [3]

The Blasius relationship is of the form

$$F = y\,.\,Re^{-Z}$$

Which is a power function, giving a straight line on a log-log plot, For Power Law fluids, y and z show some dependence on the flow behavior index n.

Yield expressions for y and z:

$$y = \frac{\log(n)+3.93}{50} \tag{I.14}$$

$$z = \frac{(1.75-\log(n))}{7} \tag{I.15}$$

The von Karman relationship is more complex of the form:

$$\frac{1}{\sqrt{f}} = y\,.\log Re\sqrt{f} + z \tag{I.16}$$

The following modification for Power Law fluids:

$$\frac{1}{\sqrt{f}} = \frac{4}{n^{.75}}\,.\log\left(Re.f^{1-\left(\frac{n}{2}\right)}\right) - \frac{4}{n^{1.2}} \tag{I.17}$$

The mathematical expressions given above are widely used for computerized analysis of turbulent flow. Mathematical expressions for other non-Newtonian fluids besides the Power Law fluid are sparse because the preparation of fluids with the necessary properties is difficult.[3]

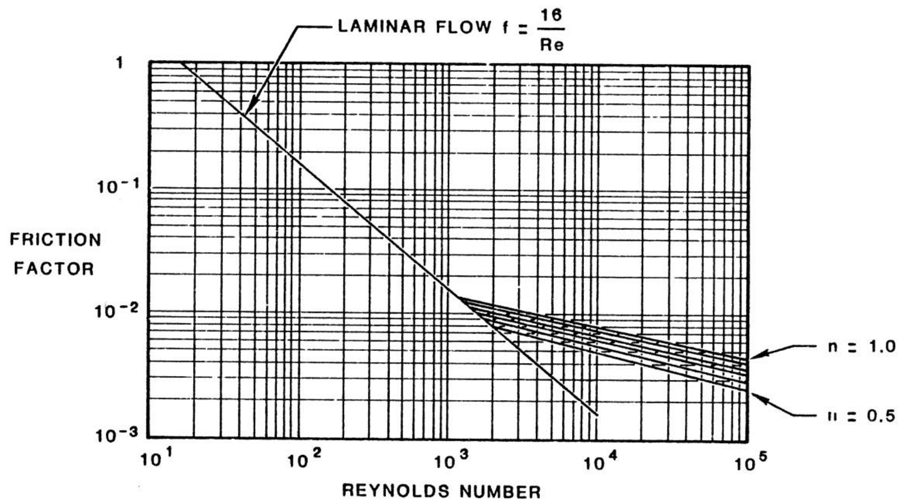


Fig.I.5: Von Karman Correlation for Various Power Law Fluids.[3]

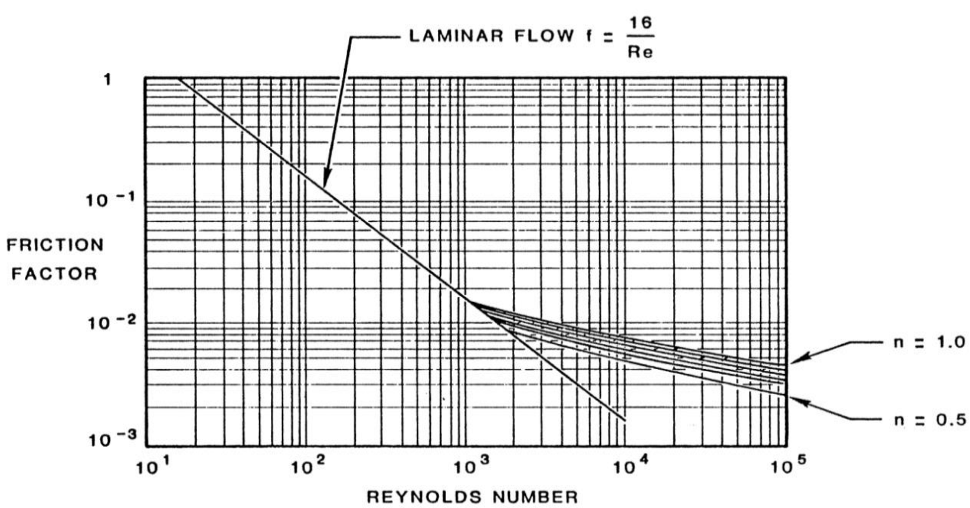


Fig.I.6:Blasius Correlation for Various Power Law Fluids.[3]

I.6. Reynolds numbers :

The Reynolds number is a dimensionless, empirically-deduced parameter. significance of the Reynolds number is in its use as a correlation parameter: different fluids with different properties exhibit similar flow characteristics at the same Reynolds number.

The major use of the Reynolds number is the determination of flow regime. Generally, the flow regime of a liquid changes from laminar to turbulent at a fairly well-defined Reynolds number value.

The Reynolds number was the result of the experiments in 1883 of Osborne Reynolds. Using water as the subject fluid, Reynolds related the various factors affecting flow (fluid density, fluid viscosity, the average velocity, and pipe diameter) and defined the Reynolds number as :[3]

$$\mathrm{Re} = \frac{\rho D_h V}{\mu} \tag{I.16}$$

D_h represent the hydraulic diameter

V is the fluid velocity

μ dynamic viscosity

It is apparent that Equation (I-16) is not valid for a non-Newtonian fluid because non-Newtonian fluids do not have an absolute viscosity; the viscosity varies with shear rate. The apparent viscosity equal to:

$$\mu_a = \frac{\tau}{\dot{\gamma}} = K\left(\frac{8V}{D}\frac{3n+1}{4n}\right)^{n-1} \tag{I.17}$$

The Reynolds number of Pseudoplastic fluid can be calculated from the two equations (I.16) and (I.17) : [2]

$$Re = \frac{V^{2-n} D^{n} \rho}{K 8^{n-1}\left(\frac{3n+1}{4n}\right)^{n}} \tag{I.18}$$

Another value used to determine flow regime is the critical velocity. The critical velocity is calculated by solving the Reynolds-number equation for velocity at the Reynolds-number value at which turbulence begins, called the critical Reynolds number. The critical Reynolds number for psedoplastic fluid obeys to power law model: [3]

$$Re_c = 3470 - 1370n \tag{I.19}$$

The critical velocity is obtained from: : [2]

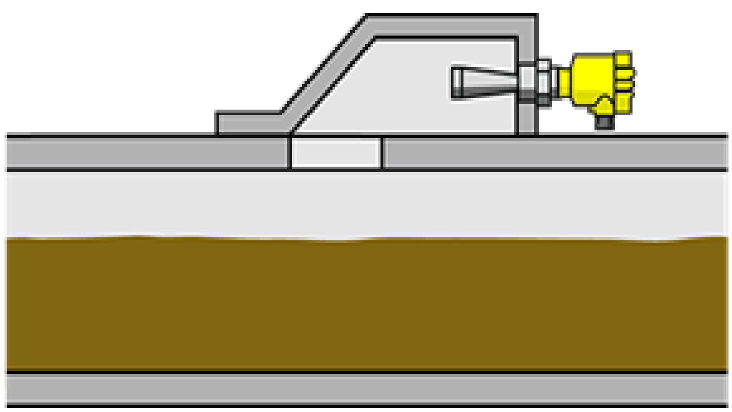
$$V_c = \left(\frac{(3470-1370n)K8^{n-1}\left(\frac{3n+1}{4n}\right)^n}{D^n\rho}\right)^{\frac{1}{2-n}} \tag{I.20}$$

I.7. Application of convergent pipes:

Convergent pipes, along with their specific component names, play a crucial role in various applications within the oil drilling rig industry. Here are some common applications supported by relevant references:

I.7.1. Mud circulation system:

Convergent pipes, including mud return lines, mud standpipe, and mud manifold, are essential components of the mud circulation system in drilling rigs. They efficiently direct the flow of drilling fluid, known as mud, throughout the system, serving multiple functions such as cooling the drill bit, removing rock cuttings, maintaining hydrostatic pressure, and providing lubrication (view fig.I.7) [4]



FigI.7: Drilling mud pipe flow measurement [6]

I.7.2. Jetting and cleaning:

Convergent pipes, such as high-pressure jetting lines, are employed for jetting and cleaning operations in drilling rigs. High-pressure jets of water or other fluids are directed through these pipes to clean the wellbore, remove debris, and dislodge obstructions that might impede the drilling process [5]

I.7.3. Well control and blowout preventer (BOP) systems:

Convergent pipes, including kill lines and choke lines, are integral to well control and blowout preventer systems used during drilling operations. These systems are crucial for maintaining control over the well and preventing blowouts, which are uncontrolled releases of oil or gas. Convergent pipes aid in directing and controlling fluid flow during well control operations (view fig.I.8) [5]

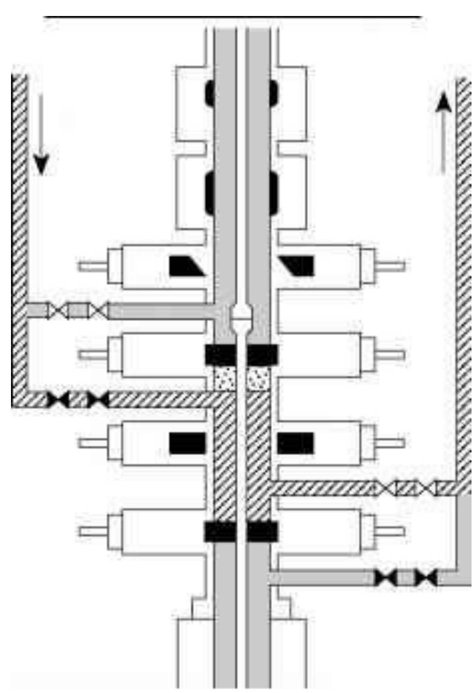


Fig.I.8:Removal of gas trapped in BOP[9]

I.7.4. Cementing operations:

Convergent pipes, such as cementing lines, are utilized during cementing operations in oil drilling. They facilitate the precise placement of cement into the wellbore to create a cement sheath around the casing. This ensures proper zonal isolation and well integrity, preventing fluid migration and maintaining well stability (view fig.I.9) [4]

I.7.5. Sampling and logging:

Convergent pipes, including formation sampling lines and logging tool conduits, are employed for sampling and logging operations in drilling rigs. They enable the extraction of formation samples for analysis and serve as conduits for logging tools to measure various well properties, including formation pressures, fluid composition, and rock properties [5]

I.7.6. Hydraulic fracturing:

Convergent pipes, often referred to as fracturing lines, are utilized in hydraulic fracturing operations, commonly known as fracking. They are responsible for delivering fracturing fluids under high pressure into the well. The fluids create fractures in the rock formation, allowing for the release of oil or gas. Convergent pipes ensure efficient fluid delivery and control throughout the fracturing process [5]

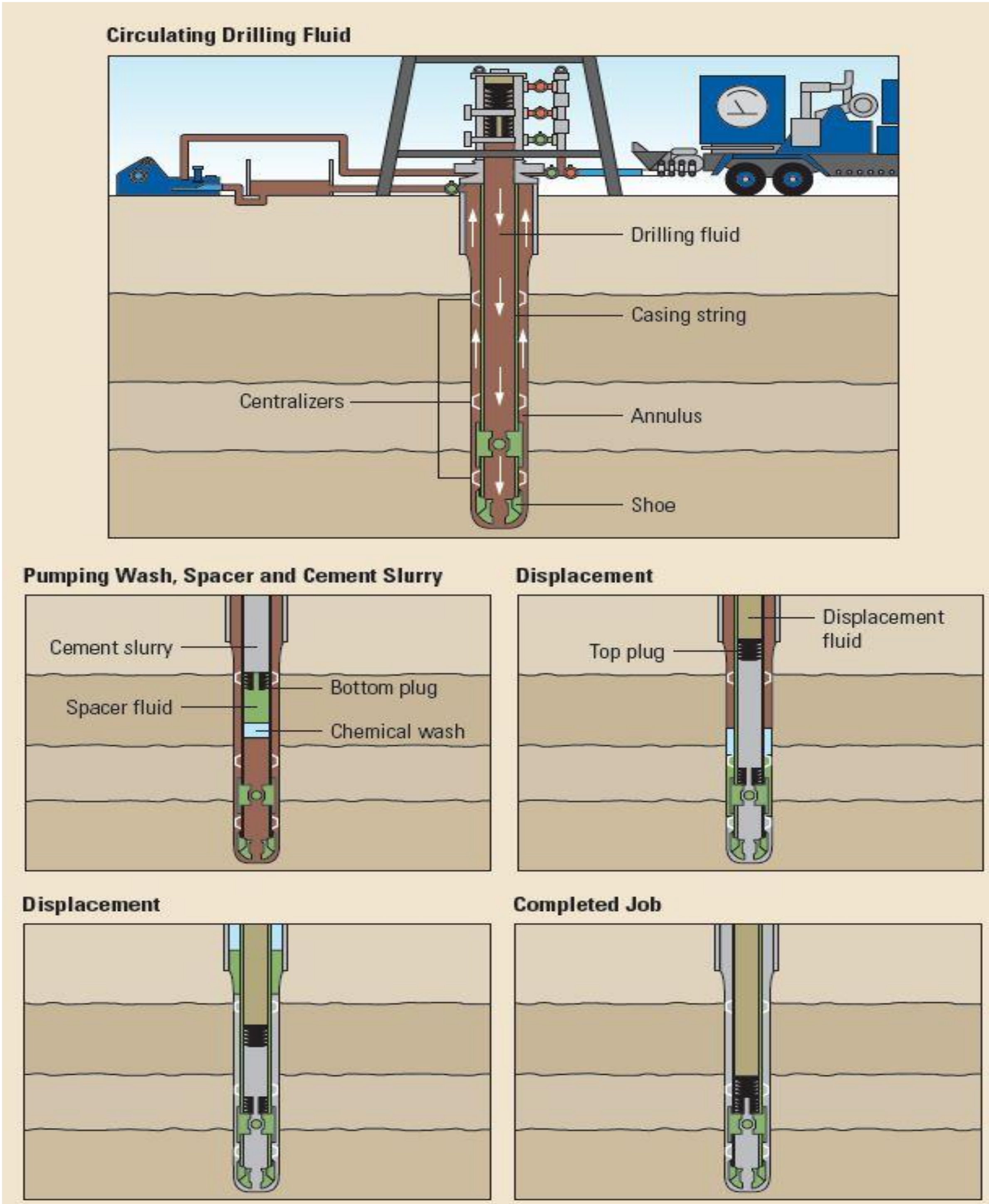


Fig.I.9:Primary cementing operation[8]

I.7.7. Jet nozzles:

The part of the bit that includes a hole or opening for drilling fluid to exit. The hole is usually small (around 0.25 in in diameter) and the pressure of the fluid inside the bit is usually high, leading to a high exit velocity through the nozzles that creates a high-velocity jet below the nozzles. This high-velocity jet of fluid cleans both the bit teeth and the bottom of the hole. The sizes of the nozzles are usually measured in 1/32-in increments (although some are recorded in millimeters), are always reported in "thirty-seconds" of size (i.e., fractional denominators are not reduced), and usually range from 6/32 to 32/32. (view fig.I.10) [5]

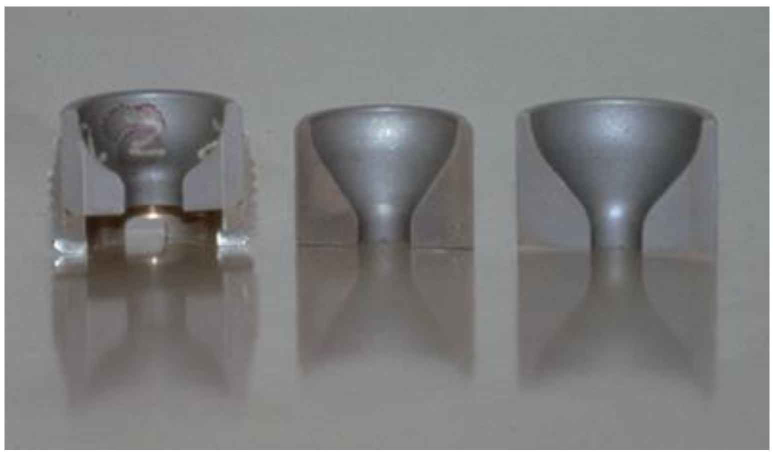


Fig.I.10: Jet nozzles [8]

I.7.8. Pinch valve:

The pinch valve or sometimes called squeeze valve is a flow control valve which operates on the squeezing action of rubber sleeve inside the valve. The valve is opened or closed by compressed air, which squeezes the rubber sleeve and controls the passage area. The valve has no constraints in the open position, allowing a wide range of media to pass through the bore. The valve's flexible interior rubber sleeve isolates the medium, reducing the danger of contamination. (view fig.I.11) [6]

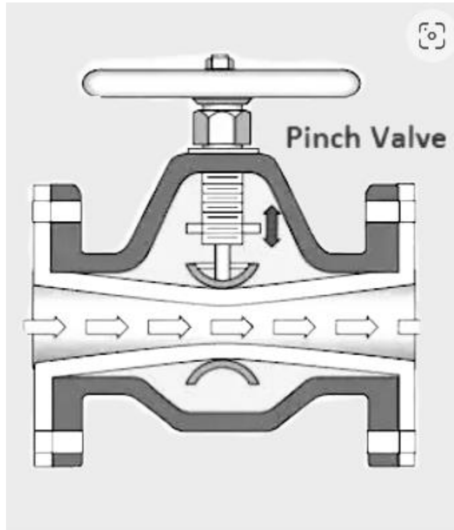


Fig.I.11: Pinch valve [6]

These applications demonstrate the versatility of convergent pipes, including their specific component names, in oil drilling rigs. The specific utilization and configurations may vary based on factors such as drilling rig design, well conditions, and operational requirements

CHAPTER II:

MODELIZATION

II.1. Physical description:

As part of the investigation, a comprehensive dynamic and thermal analysis was conducted on the turbulent flow of non-Newtonian fluid in a convergent conduit. The dimensions of the convergent configuration were rigorously determined, with a length of 45 cm and a width of 5 cm at the outlet. These dimension values were empirically derived based on previous scientific contributions and their practical relevance in industrial contexts. (view fig.II.1).[11]

The convergent channel is studied in two positions, horizontal and vertical. the fact that these two positions are the most widespread in the drilling process

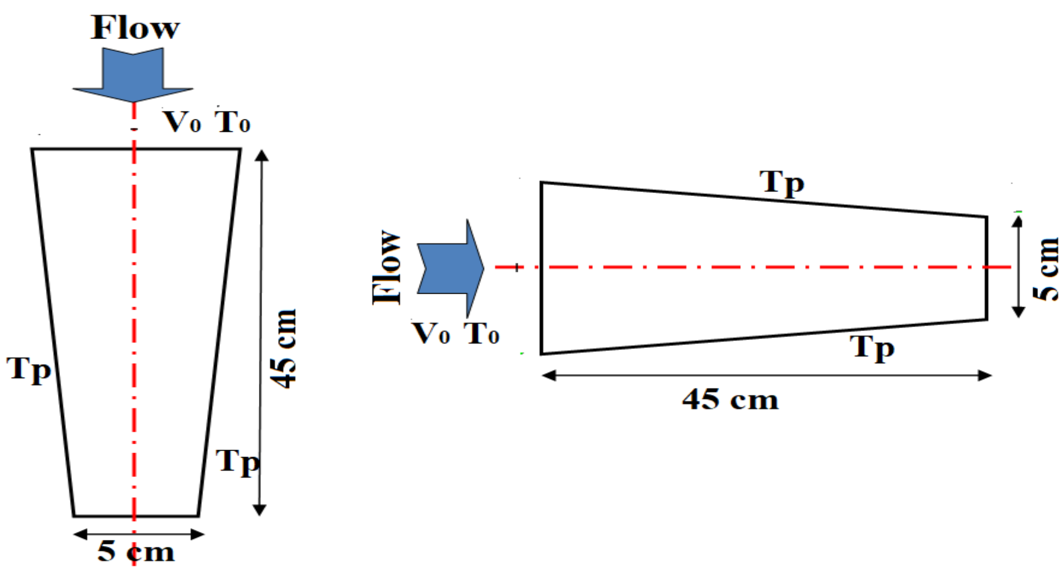


Fig.II.1: Geometry scheme of vertical and horizontal convergent channel

II.1. Hypothesis

- ➢ The flow is incompressible, permanent and turbulent
- ➢ The heat transfer mode is forced convection
- ➢ The thermo-physical parameters (density, thermal conductivity and heat capacity) are constant, calculated at a reference temperature Tref or film temperature
- ➢ All walls are considered impermeable and isothermal

- The drilling fluid, depending on the case studied, is non-newtonian.

II.1. Mathematical formulation:

In two-dimensional system, the flow is permanent, incompressible, forced and turbulent. with neglected viscous dissipations, is managed by:

II.1.1 Continuity equation

$$\frac{\partial \bar{u}}{\partial x}+\frac{\partial \bar{v}}{\partial y}=0 \qquad \text{(II .1)}$$

II.1.2. Momentum Equations

$$\rho\left(\frac{\partial \bar{u}\bar{u}}{\partial x}+\frac{\partial \bar{u}\bar{v}}{\partial y}\right)+\rho\left(\frac{\partial \overline{u'u'}}{\partial x}+\frac{\partial \overline{u'v'}}{\partial y}\right)=-\frac{\partial \bar{P}}{\partial x}+\mu\nabla^2\bar{u} \qquad \text{(II.2.a)}$$

$$\rho\left(\frac{\partial \bar{v}\bar{u}}{\partial x}+\frac{\partial \bar{v}\bar{v}}{\partial y}\right)+\rho\left(\frac{\partial \overline{v'u'}}{\partial x}+\frac{\partial \overline{v'v'}}{\partial y}\right)=-\frac{\partial \bar{P}}{\partial y}+\mu\nabla^2\bar{v} \qquad \text{(II.2.b)}$$

II.1.3. Energy equation

$$\left(\frac{\partial \overline{u}\overline{T}}{\partial x}+\frac{\partial \overline{v}\overline{T}}{\partial y}\right)=\frac{\partial}{\partial x}\left(\frac{\lambda}{\rho cp}\frac{\partial \bar{T}}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\lambda}{\rho cp}\frac{\partial \bar{T}}{\partial y}\right)-\frac{\partial(\overline{u'T'})}{\partial x}\frac{\partial(\overline{V'T'})}{\partial Y} \qquad \text{(II.3)}$$

Where:

u = $\bar{u}+u'$

v = $\bar{v}+v'$

p = $\bar{p}+p'$

The symbol ($\bar{\ }$) represents the statistical average or ensemble average operator and the symbol ($'$) denotes fluctuations or deviations from these averages.

Note that this energy balance resembles that in the laminar regime, except the terms: $\overline{u'T'}$ and $\overline{v'T'}$, called the temperature extra-terms and the stresses of Reynolds, following:

$$-\overline{\rho u'u'} = 2\nu_T\frac{\partial u}{\partial x} - \frac{2}{3}\rho K \quad ;-\overline{\rho v'v'} = 2\nu_T\frac{\partial v}{\partial y} - \frac{2}{3}\rho K \;;-\overline{\rho u'v'} = \nu_T\left(\frac{\partial \bar{v}}{\partial x} + \frac{\partial \bar{u}}{\partial y}\right) \tag{II.3}$$

Where: ν_T is the turbulent viscosity and k is the turbulent kinetic energy. By simulation one can write the extra-terms of temperature according to gradient temperature, as follows:

$$-\overline{\rho\, u'T'} = \Gamma_T\frac{\partial T}{\partial x};-\overline{\rho\, v'T'} = \Gamma_T\frac{\partial T}{\partial y} \tag{II.4}$$

Where: Γ_T is the turbulent diffusivity.

Since the turbulent transport of momentum and heat is due to same mechanisms, mixing by backwater, the value of the turbulent diffusivity can be taken, to be close to turbulent viscosity.

According to the definition of the turbulent Prandtl number, we have: $Pr_T = {}^{\nu_T}/_{\Gamma_T}$

II.2. Numerical Formulation and Solving Method

Solving the set of governing equations from (II.1) to (II.4) of this flow system, is performed by the numerical method of finite volumes. In using CFD code, ANSYS-FLUENT 19.2

II.2.1. Finite volume method

The partial differential equations are solved in an approximate way using of a mesh made up of finite volumes which are small disjoint volumes (in 3D, 2D surfaces, 1D segments) whose union constitutes the field of study. THE finite volumes can be constructed around points of an initial mesh, but this is not a necessity The main characteristics of the finite volume method in mechanics continuous, are:

- The physical approach
- Adaptation to any geometries
- The existence of several schemes for solving nonlinear terms.

- The use by several commercial codes in mechanics such as Fluent and CFX

II.2.2. The diagrams used on the Ansys-Fluent solver

Fluent is a very responsive universal industry solver. It is often considered as a reference in the field of modeling flow systems of fluids. The configuration of the model is done through a graphical interface. It has a scripting interface to automate calculation processes. It has a library rich, containing a relatively large number of models, being able to cope with various multidisciplinary aspects of flow systems and transport phenomena (such as as fluid mechanics, heat transfer, thermodynamics, etc.).

Suitable schemes for solving differential equations that deal with these flow systems under the boundary conditions mentioned above:

➢ To determine the pressure field: the SIMPLE diagram in the case of a laminar flow, the coupled scheme for a turbulent regime

➢ The second-order upwind scheme, for the discretization of convective terms in the dynamic equation, the energy equation, the dissipation rate equation and the kinetic energy equation

II.2.3. Boundary conditions

The table.II.1 shows the boundary conditions of this problem.

Table 1: Boundary conditions

	Inside
At the inlet **V_0 and T_0**	• Fluid Velocity • Fluid temperature
At the outlet	Pressure
Walls Tp	The isothermal temperature at the walls

II.2.4. Mesh

The choice of the mesh is an essential step which influences the precision and the accuracy of the numerical results. Therefore, a mesh meeting the objectives, contains a number of stitches, a distance between the stitches and a shape of the stitch, suitable Practically, there are no precise rules for the creation of a mesh

suitable, however there are different approaches that make it possible to obtain a

grid acceptable, like:

- The maintenance of a good quality of the elements

- The assurance of good resolution in regions with strong gradients.

- The assurance of good smoothing in the transition zones between the parts to be fine mesh and those with coarse mesh.

- The minimization of the total number of elements (reasonable calculation time)

II.2.5. Mesh independence

In general, the finer, the finite element mesh, the more accurately one can capture the contours of a geometry and there are more “data points” on the geometry to generate an accurate displacement and stress response.

A mesh independence (or grid independence) study is to determine the independence of the results on the mesh density.

Six cases of mesh are created to choose the optimal one (view Table.II.2). The water is used for this study.The water is used for this study; its physic characteristics are shown in Tab.II.2. The inlet velocity V_0 equal to 0.4019 m/s (at Re = 10000) and temperature T_0 323 K. the wall temperature equal to 303 K.

Table 2: The thermophysical characteristics of the water

Density (ρ)	998.2 kg/m3
Dynamic viscosity (µ)	0.001003 kg/m.s
Thermal conductivity (λ)	0.6 W/m.s
Heat capacity (Cp)	4182g.K

Table 3 : Comparative table of proposed meshes.

MESH	NODES	ELEMENTS	TIME
1	621	568	0:00:11
2	788	725	0:01:21
3	1166	1084	0:01:02
4	421	371	0:01:02
5	523	472	0:00:04
6	66380	65557	0:00:59
7	1366	1279	0:00:26

Figure II.7 shows the evolution of the speed profile of the cases studied, the mesh chosen was the 7^{th} case (see fig.II.8), given its good accuracy of the results and the minimum of the calculation time taken in execution.

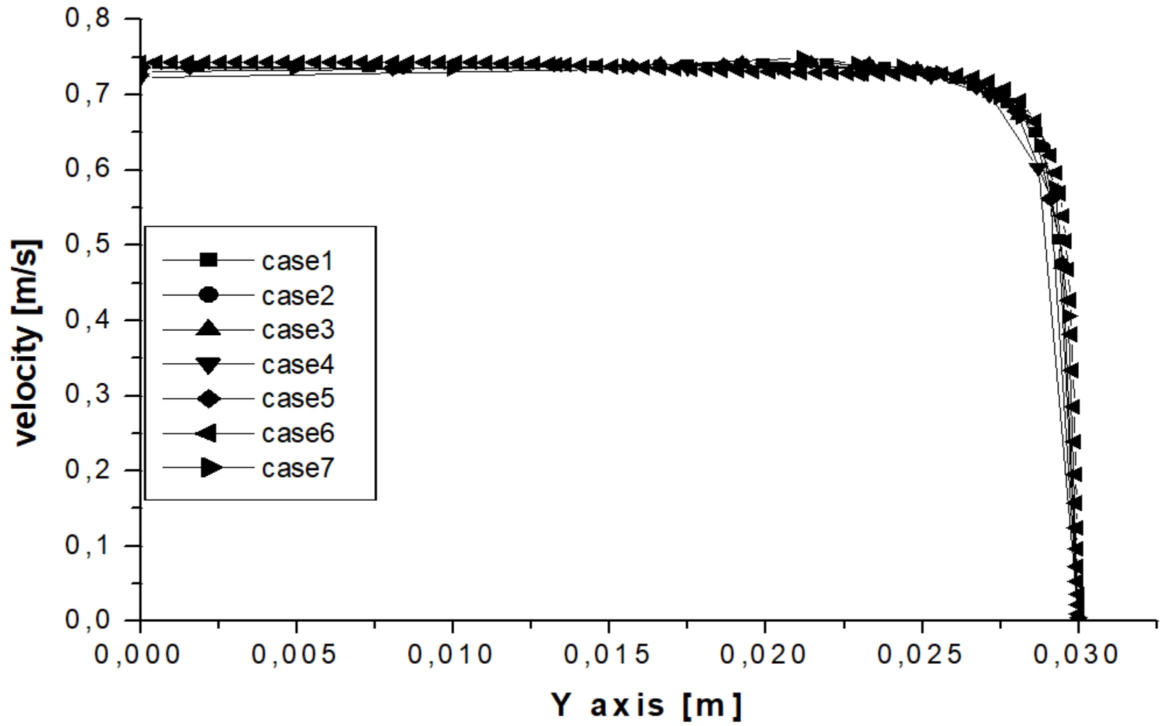


Fig.II.2:Velocity curve for different meshes

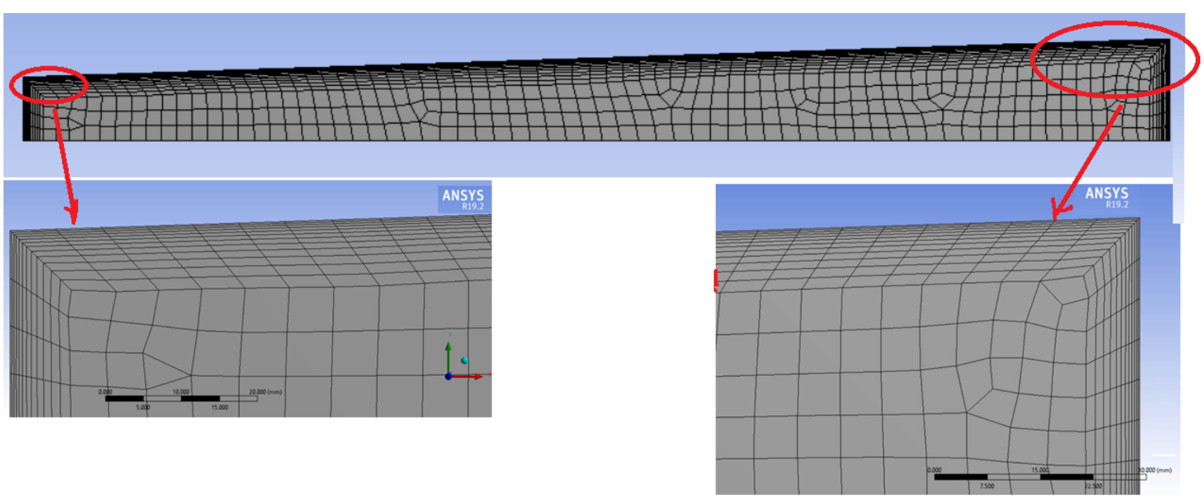


Fig.II.3. Selected mesh scheme of 7th case

CHAPTER III:

RESULTS AND DISCUSSION

III.1. The conditioning of this flow system

The drilling mud used is similar to pseudoplastic fluid. Non-Newtonian fluids obey the power law. Its thermophysical characteristics, are mentioned in the table III.1.

Table 4 III.1.Thethermophysical characteristics of the drilling mud

Density (ρ)	1200 kg/m3
Consistency index (K)	0.0577 kg.s^{n-2}/m
Flow index (n)	0.8549
Thermal conductivity (λ)	0.6 W/m.s
Specific heat capacity (Cp)	4070 j/kg.K

The flow is incompressible, permanent, forced, and turbulent. It studied for four Reynolds numbers: 10000, 20000, 30000, and 50000.Its velocities at the inlet of the duct are mentioned in Table.III.2

Table 5 III.2:Thethermophysical characteristics of the drilling mud

	Case 1	**Case 2**	**Case 3**	**Case 4**
Re	10000	20000	30000	50000
V_0 [m/s]	6.36	11.66	16.62	18,29

III.2. Results and discussions

The pace is studied in several stages at: 0.1 m, 2.5 m and 4.4m from entrance:

III.2.1. Velocity profiles

Figure.III.1-4 show the velocity profiles in the convergent duct for the turbulent flow. Notice that turbulence flattens the velocity profile in the center and increases the velocity distribution near the wall [12]. From the inlet to the outlet of the convergent channel, velocity profiles have the same appearance. The only change is the width of this curve, as the convergent does not have a constant section.

Velocity profiles can be considered by dividing the flow into three zones:

- Laminar sub-layer: this represents a thin layer next to the pipe wall in which the effects of turbulence are assumed to be negligible. Assuming the no-slip boundary condition at the wall, the fluid in contact with the surface is at rest. Furthermore, all the fluid close to the surface is moving at a very low velocity. Consequently, the net shear force acting on any element of fluid in this zone must be negligible, the retarding force at its lower boundary being balanced by the accelerating force at its upper boundary. Thus, the shear stress in the fluid near the surface must approach a constant value.

- Transition zone: This region separates the so-called viscous or laminar sub-layer and the fully turbulent core prevailing in the middle portion of the pipe

- Turbulent core:A fully turbulent region comprising the bulk of the fluid stream where momentum transfer is attributable virtually entirely to random eddies and the effects of viscosity are negligible. The shear stress at any point in the fluid, at a distance y from the wall, is made of ‘viscous’ and ‘turbulent’ contributions, the magnitudes of which vary with distance from the wall. [13]

As the flow transitions into turbulence, rapid pressure and velocity fluctuations (also known as pulsations), fast changes in a given variable, having dynamic spatial regions that are generated as molecular motion swirls collectively, forming “coherent structures.” Coherent structures are “grouped” fluid particles that move in unison as they rotate, stretch, translate, and decay. These are the "distinct curls" envisioned by Reynolds-turbulent eddies. [12]

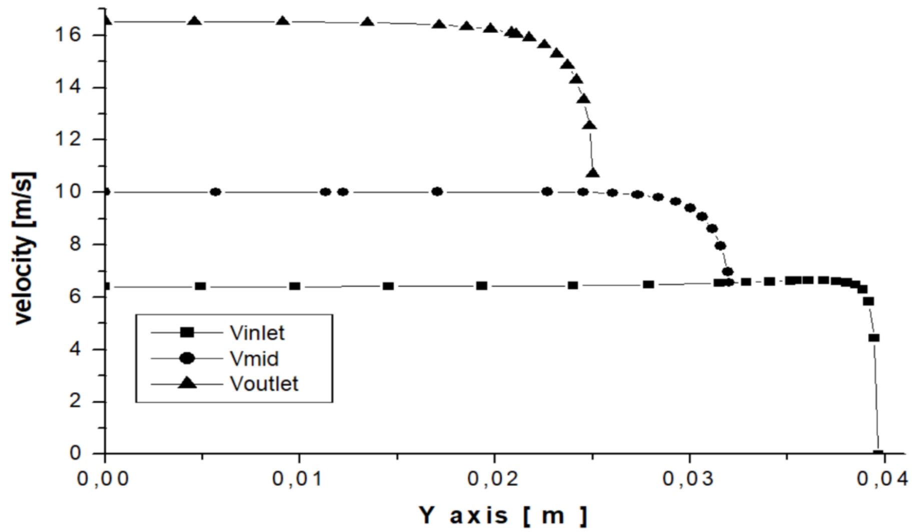


Fig.III.1 Velocity profiles, along the horizontal converging duct (from inlet to outlet)at Re = 10000

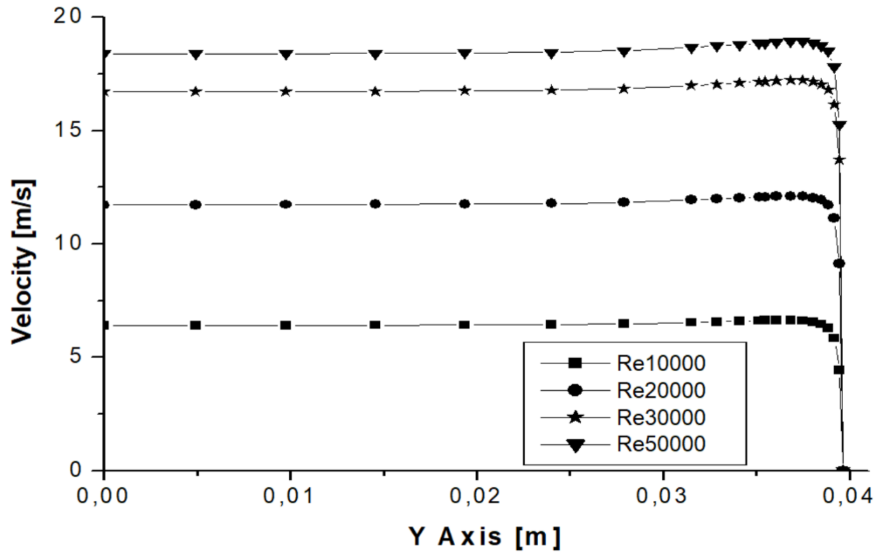


Fig.III.2.Velocity profiles at the inlet of the horizontal convergent duct. Re varied (from 10000 to 50000)

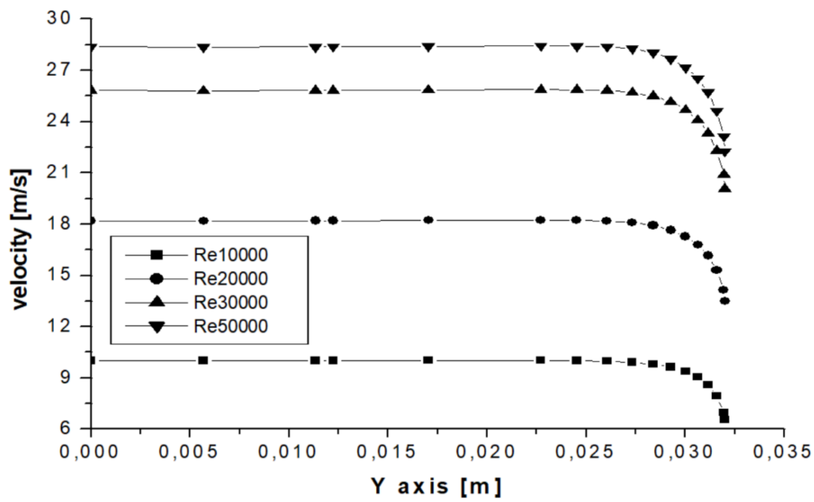


Fig.III.3.Velocity profiles at the middle of the horizontal convergent duct. Re varied (from 10000 to 50000)

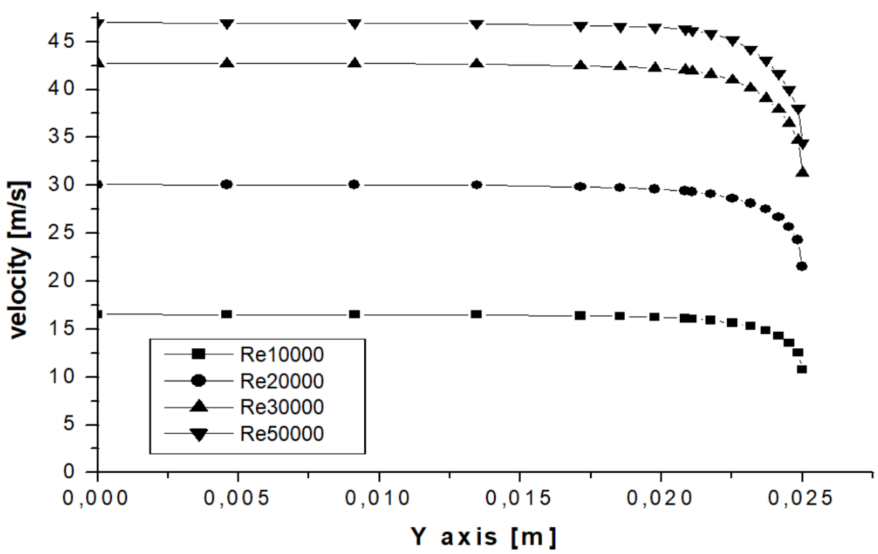


Fig.III.4.Velocity profiles at the outlet of the horizontal convergent duct. Re varied (from 10000 to 50000)

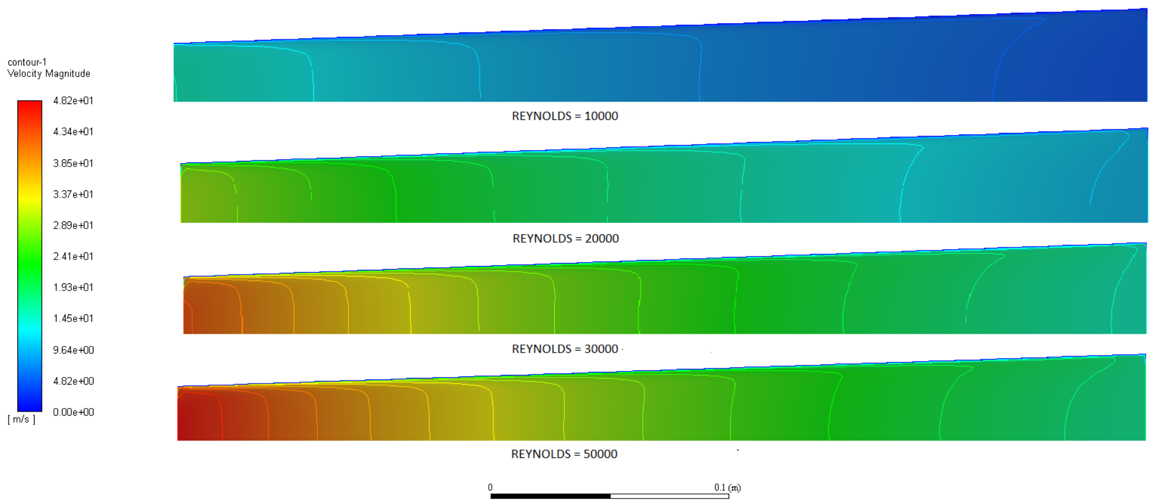


Fig.III.5.Velocity contours in the horizontal convergent duct. Re varied (from 10000 to 50000)

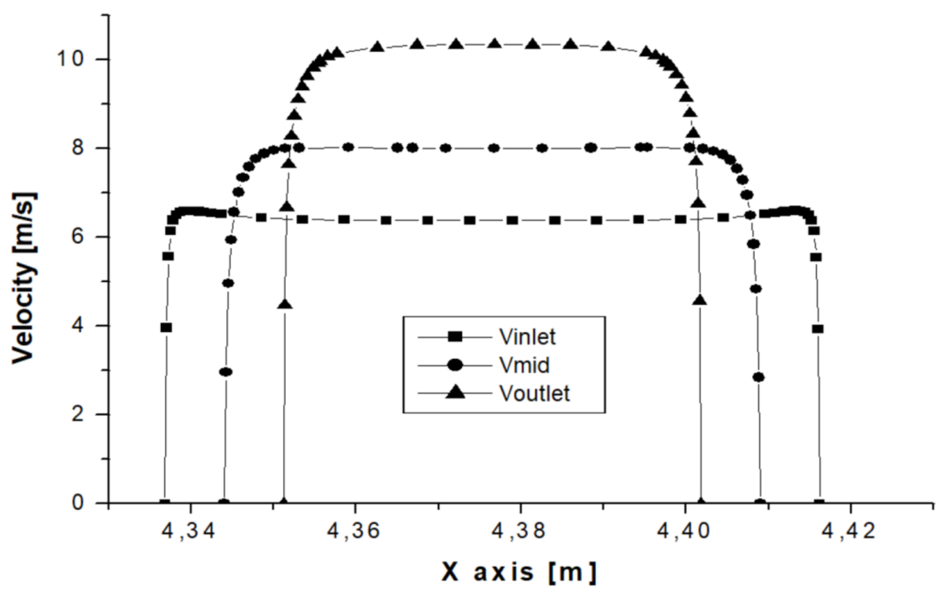


Fig.III.6. Velocity profiles, along the vertical converging duct (from inlet to outlet) at Re = 10000

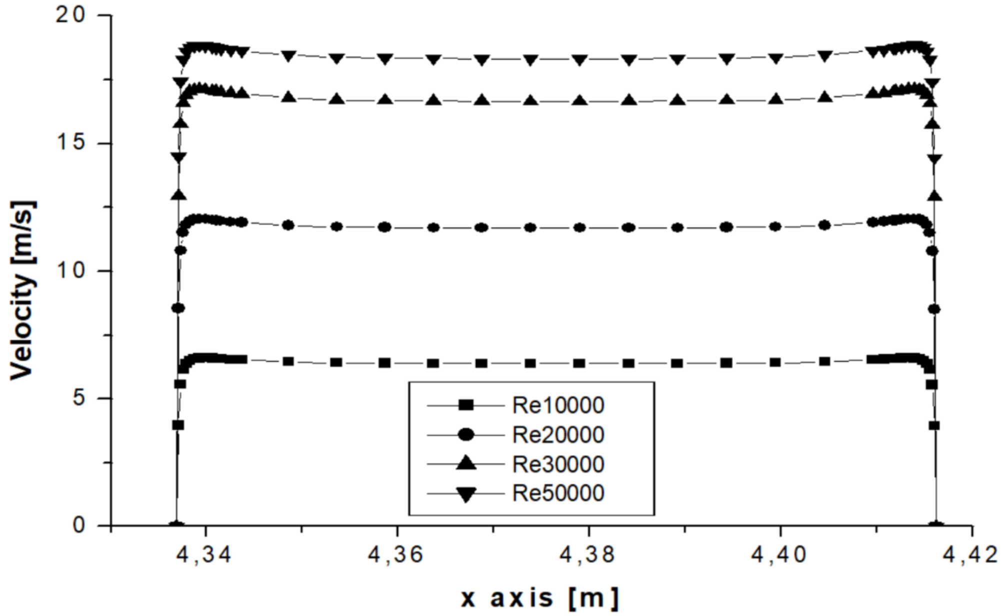


Fig.III.7. Velocity profiles, at the inlet of the vertical converging duct, Re varied (from 10000 to 50000)

The flow accelerates at the outlet of the channel, under the effect of convergence. More Reynolds numbers increase, more the velocity at the outlet More Reynolds numbers increase, more the velocity at the outlet becomes important. At Re = 10000, V_1oulet ≈17 m/s, and when Re = 50000, V_5oulet ≈ 48 m/s, (view Fig. III.4-5). The effect of Re on velocities is not limited to the outlet of the convergent, but it spreads out all the duct, (view Fig. III.2-3).

The figures III.6-9 show that the velocity profiles of vertical convergent duct have the same appearance as those of horizontal channel and the velocity is proportional to the Reynolds number.

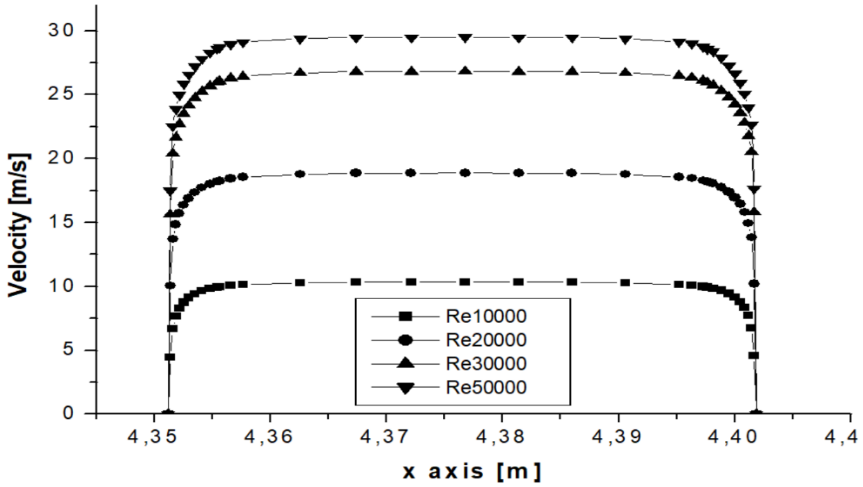


Fig.III.8. Velocity profiles,at the oulet of the vertical converging duct, Re varied (from 10000 to 50000)

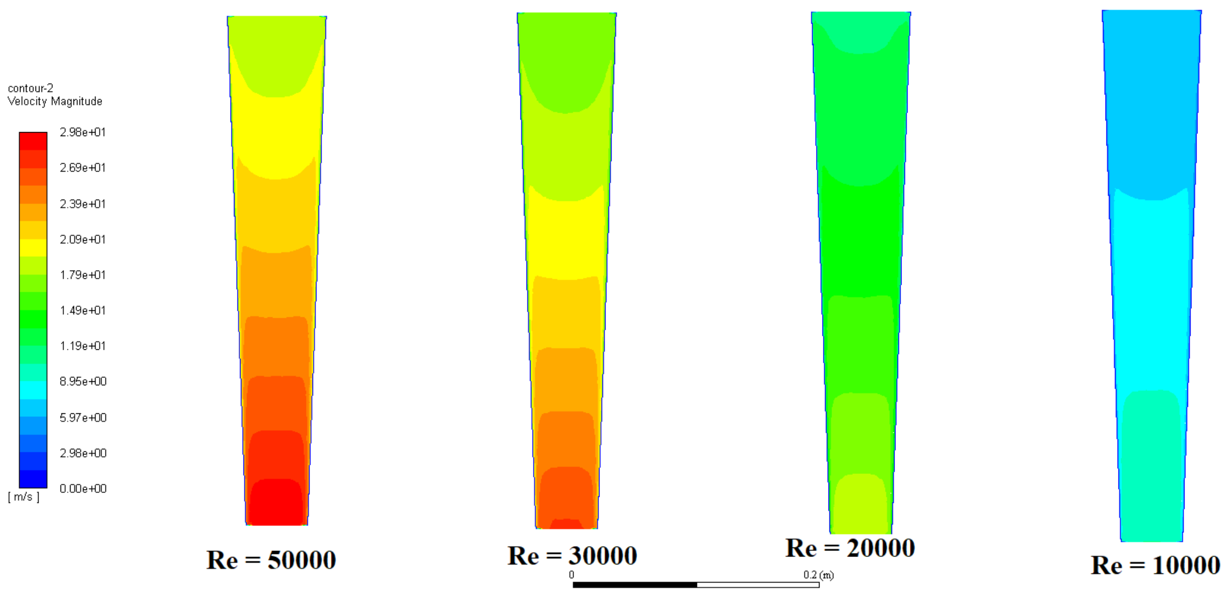


Fig.III.9. Velocity contours in the vertical convergent duct. Re varied (from 10000 to 50000)

III.2.2. Temperature Profiles

The temperature profiles according to Figures III.10 and III.12 in horizontal convergent pipe and III.11 and III.13 in vertical convergent pipe have a flattened shape at the center of the pipe, where turbulence, ensures fluid mixing through large vortices with the same temperature.

Near the walls, these profiles exhibit straight lines with a steep slope, due to significant temperature gradients. At the outlet there is some loses heat, but this thermal exchange with the two walls is relatively weak compared to the exchange that occurs along the rod, given the nature of convection (forced) within this system

At the middle and outlet of the horizontal convergent channel, the temperature rate decreases near the wall compared with the gradient at the inlet, proving the existence of heat exchange between the fluid and the wall. This exchange is not great because there is a small difference between the fluid temperature at the entrance of the duct and the wall temperature, thus the convection mode.

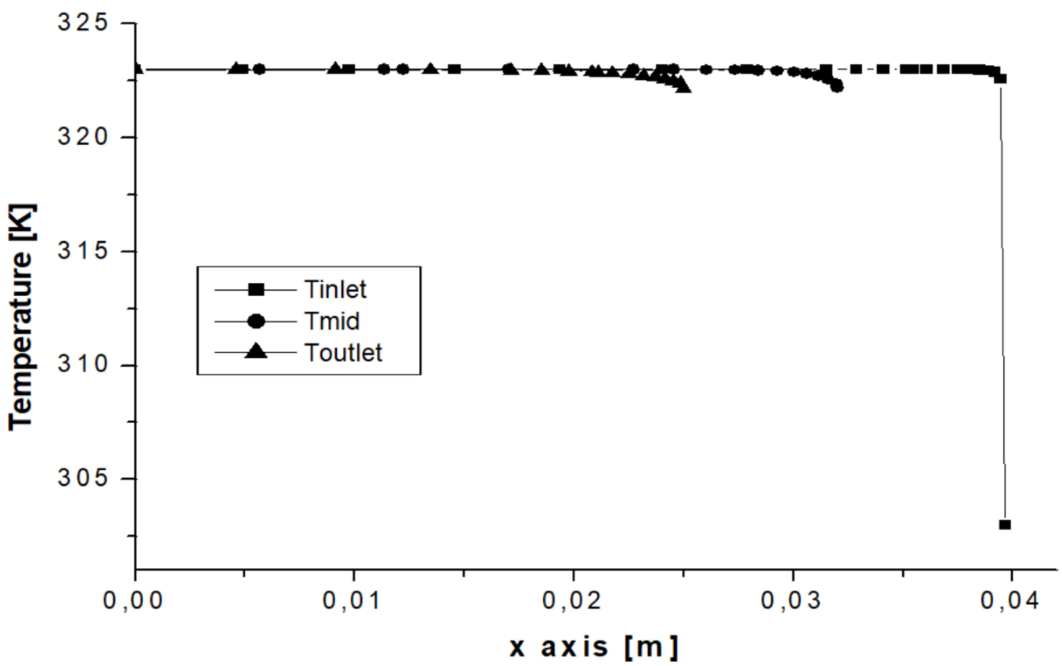


Fig.III.10.The temperature profiles along the horizontal convergentduct (from inlet to outlet)at Re = 10000

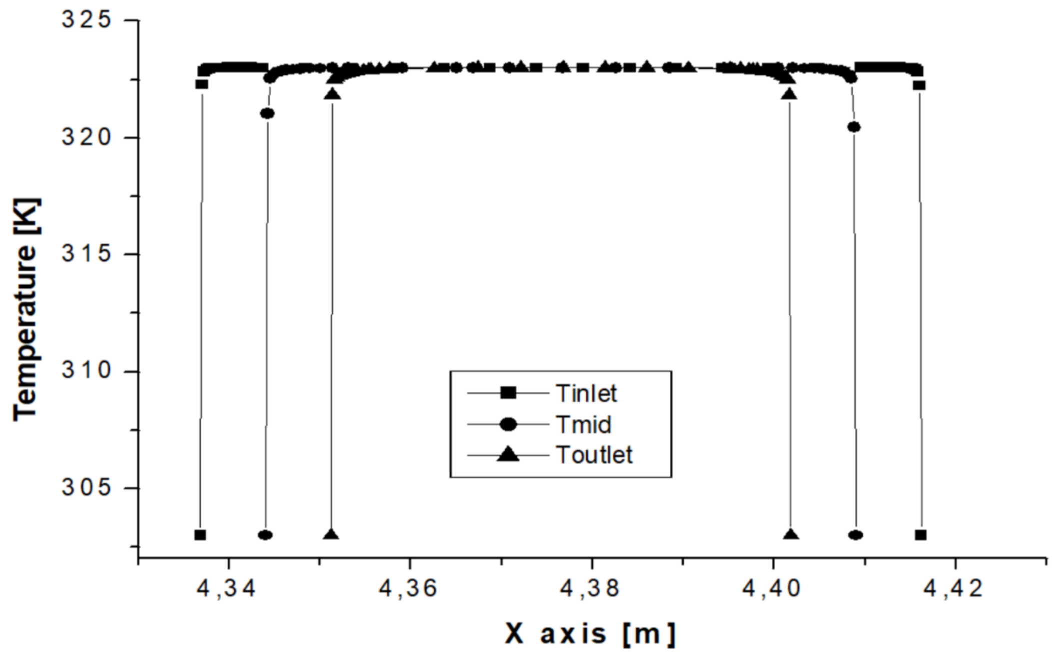


Figure III.11.The temperature profiles along the verticalconvergentduct (from inlet to outlet)at Re = 10000

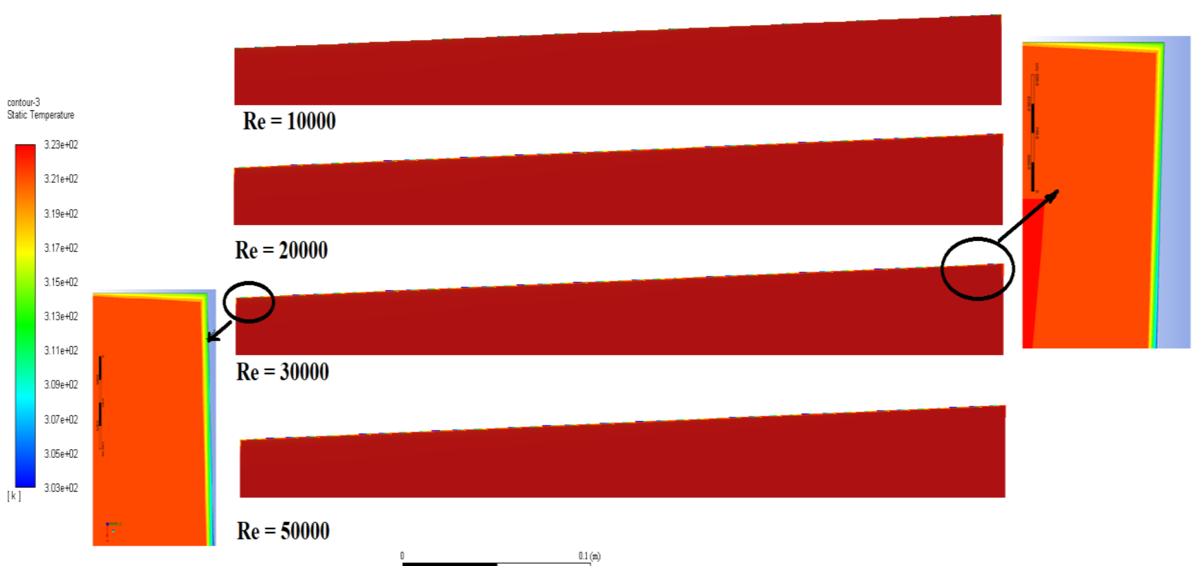


Fig.III.12. Temperature contours in the vertical convergent duct. Re varied (from 10000 to 50000)

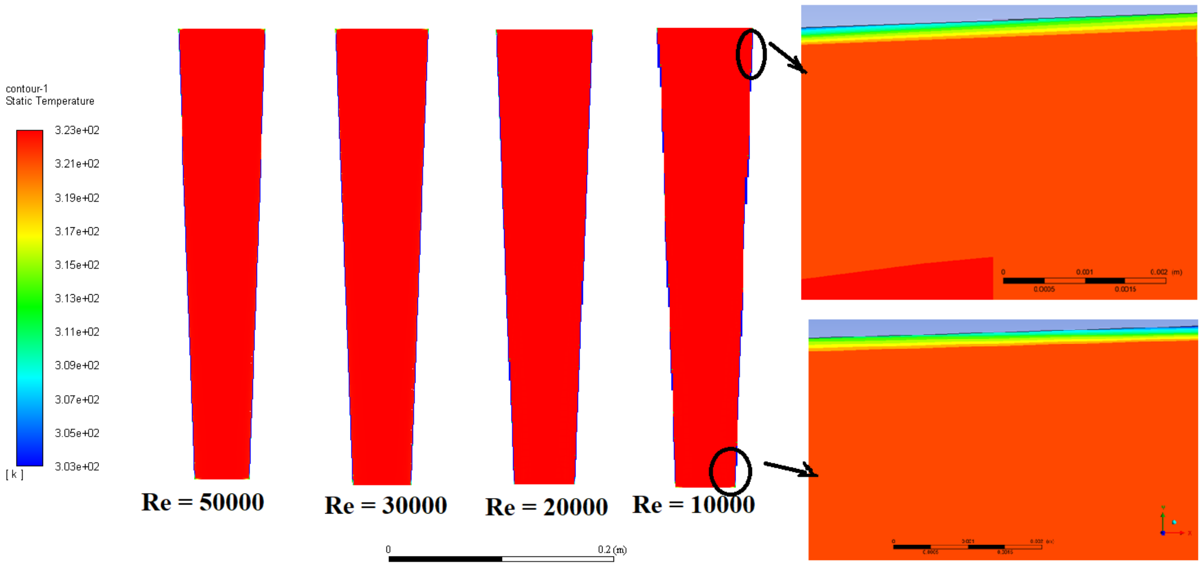


Fig.III.13.Temperature contours in the vertical convergent duct. Re varied (from 10000 to 50000)

CONCLUSION

The forced and turbulent flow of pseudoplastic fluid into a convergent pipe was studied numerically with the Ansys Fluent code. The findings of this study contribute to the understanding of the dynamic and thermal deportment of this flow, under varying Reynolds numbers.

The conclusion can be summarized at the next points:

- The velocity of pseudoplastic fluid in turbulent flow revealing the presence of both laminar sub-layers, transition zones, and turbulent cores.

- The higher Reynolds numbers led to increased velocities throughout the ducts, particularly at the outlets.

- In the core of flow, the fluid keeps its temperature. But in the laminar sublayer of flow, a small heat exchange occurs between the fluid and the wall. It can be negligible under the influence of forced convection.

There are some recommendations to add in order to complete this work:

- Vary the convergence angle.
- Use other times of non-newtonian fluids.
- Study energy dissipations in this flow system.

[1]: . J. Azzar, G. Robello Samuel, drilling engineering , PennWell, 2007

[2]: Chambre syndicale de la recherche et de la production du pétrole et du gaz naturel comité des techniciens, DrillingMud and CementSlurryRheology, Springer Science & Business Media, 1982

[3]: Alun Whittaker, Exlog Staff, Theory and Application of Drilling Fluid Hydraulics, EXLOG®, 1985

[4]:https://global.ihs.com/doc_detail.cfm?document_name=API%20RP%2013B%2D1&item_s_key=00116814

[5]: Oilfield Glossary – Schlumberger : https://glossary.slb.com/ 19/06/2023- 10H00

[6]: DONALD C. RENNELS, HOBART M. HUDSON, Pipe flow : A Practical and Comprehensive Guide, John Wiley & Sons, 2012

[7]: [https://www.bing.com/images/blob?bcid=RP-sUdSyKLsF6wkt4S5Raq90V6Sm.....0U],19/06/2023, 10H00

[8]: [https://www.bing.com/images/blob?bcid=RFOEwe676LsF6wkt4S5Raq90V6Sm.....68]19/06/2023, 10H00

[9]: [https://www.bing.com/images/blob?bcid=RFJXycNli7sF6wkt4S5Raq90V6Sm.....6Y]19/06/2023, 10H00

[10]: [https://www.bing.com/images/blob?bcid=RAgEhyDUjLsF6wkt4S5Raq90V6Sm......4]19/06/2023, 10H00

[11]: Sal Rodriguez, Applied Computational Fluid Dynamics and Turbulence Modeling Practical Tools, Tips and Technique, Springer, 2019

[12]: R.P. Chhabra J.F. Richardson, Non-Newtonian flow and applied rheology : Engineering Applications, 2nd edition, Butterworth-Heinemann, 2008