Available online at www.sciencedirect.com

ScienceDirect

Procedia CIRP 58 (2017) 584 – 589

www.elsevier.com/locate/procedia

16th CIRP Conference on Modelling of Machining Operations

CFD simulation and optimize of LN2 flow inside channels used for cryogenic machining: application to milling of titanium alloy Ti-6Al-4V

Chawki TAHRI [a,*], Pierre LEQUIEN [a], J . C. OUTEIRO[a], Gérard POULACHON [a]

[a]LaBoMaP, Arts & Metiers Campus of Cluny, UBFC, HESAM, Rue Porte de Paris, 71250 Cluny, FRANCE

* Corresponding author. Tel.: +0-33-076-924-5690. *E-mail address:* chawkitahri06@gmail.com

Abstract

Cryogenic machining is seen as a possible solution to increase tool life and improve surface integrity of machined components, by efficiently removing the heat from the cutting zone. Unfortunately, most of the publications on this topic does not investigate properly the physical phenomena occurring inside the pipe connecting the tank to the cutting tools/nozzle. These phenomena will affect the cryogenic flow characteristics at nozzle outlet, thus the performance of the cryogenic machining.

In this work, a CFD model is developed and applied to investigate the liquid nitrogen (LN2) flow inside the pipe connecting the reservoir to the cutting tool/nozzle. The results show that the effectiveness of the cooling process depends not only on the input flow parameters (pressure, velocity, temperature and gas/liquid fraction), but also on the pipe geometry.

Peer-review under responsibility of the scientific committee of The 16th CIRP Conference on Modelling of Machining Operations

Keywords: Cryogenics machining; Analysis; Cutting tool; Simulation; Finite Element Method (FEM); Fluid flow; Lubrication; Modeling; Optimization.

1. Introduction

Nowadays, cryogenic machining has been used extensively in several industries (automotive, aeronautic, etc.).
This process consists to apply a cryogenic fluid, usually liquid nitrogen (LN2) during the cutting process, with the objective to reduce the temperature, thus increase the tool life and improve the surface integrity [1]. Moreover, cryogenic machining is used to machine nickel and titanium based alloys, offering very good cutting performances (reduced tool wear and better surface integrity). LN2 flow performance will depends on many parameters of the flow, such as: pressure, flow rate, temperature and gas/liquid volume fraction. LN2 is a cryogenic liquid (boiling point -196°C [2]) stored in a tank at an appropriate pressure. It will be transported then through channels and valves. Passage through these components causes heat exchange between them (channels and valves) and the LN2, producing a temperature rise within the fluid. This produces the boiling phenomenon, thus a creation of a gas phase. To minimize this problem and have the biggest amount of LN2 as possible at the nozzle outlet, it is recommended to use a phase separator (Fig. 1). There were few studies on the application of cryogenic assistance (LN2) in milling [3, 4, 5, 6]. Up to now, almost of the Computational Fluid Dynamics (CFD) studies in machining have focused on other metal working fluids (MWF), including: Minimum Quantity Lubrication, emulsions, etc.
Duchosal et al; [7, 8] studied the optimization of the inner channels of milling tools used in MQL. Their results showed the influence of inlet pressure and tool rotation speed on the fluid in the nozzle outlet with STAR CD numerical simulation software. Numerical simulation is suitable to analyze the LN2 flow behavior inside pipes with different geometries and flow parameters. This paper is based on two parts. In the first part, a model is developed and simulated in Matlab software, in order to predict the nitrogen flow characteristics inside the pipe. The purpose of this model is to understand the relation between the inlet parameters and the variation of the physical parameters of the liquid in the outlet. The second part is devoted to a numerical simulation of the LN2 flow inside the pipe and inside the internal channels of the rotating tool.

Peer-review under responsibility of the scientific committee of The 16th CIRP Conference on Modelling of Machining Operations
doi:10.1016/j.procir.2017.03.230

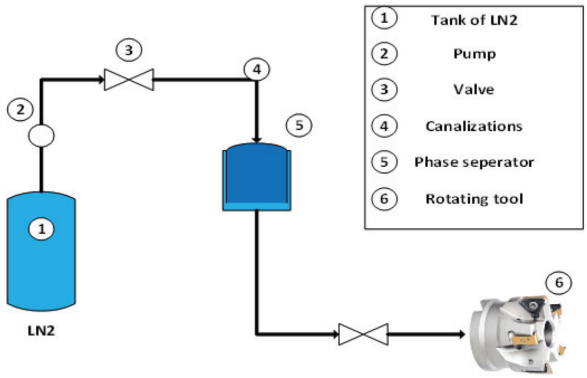

Fig. 1.Nitrogen transportation system from the tank to the nozzle exit.

Nomenclature

P	pressure (bar)
ρ	density (Kg/m^3)
ϑ	kinematic viscosity (m^2/s)
T	temperature (K)
μ	dynamic viscosity $(Pa.s)$
k	turbulent kinetic energy (m^2/s^2)
H	enthalpy (J/Kg)
λ	thermal conductivity $(W/m.K)$
P_r	prandtl number
R_e	reynolds number
N_u	nusselt number
C_{fg}	fluid-gas coefficient
C_{sf}	surface-fluid coefficient
Q	internal heat (J)
σ	surface tension (N/m)
q	heat flux (W/m^2)
U	velocity (m/s)
R_{ij}	reynolds stress tensor
I	turbulence intensity
L	length scale (m)
ϑ_t	turbulent kinematic viscosity (m^2/s)
ε	turbulent dissipation $(1/s)$
A	contact area (m^2)
g	gravity (m/s^2)
h_{fg}	latent heat (J/Kg)
T_w	wall temperature (K)
T_{sat}	saturation temperature (K)

2. Governing equations in fluid dynamics

Fluid dynamics is governed by three physical principles: mass conservation, energy conservation and the Newton's second law. These physical principles are developed into the fundamental governing equations of the fluid dynamics: the continuity, momentum, and energy equations. These equations are also known as transport equations or conservation equations. The form of governing equations that is generally used in CFD is known as the Navier-Stokes equations. These equations include the effect of viscosity on the fluid flow. Reynolds-Averaged Navier-Stokes (RANS) [9] are time-averaged equations of motion for fluid flow. It is largely used to describe continuous flow. Two-dimensional unsteady state LN2 flow in a straight pipe is simulated. Fluid motion is described by the RANS equation given by:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x^2{}_j} + \frac{1}{\rho}\frac{\partial R_{ij}}{\partial x_j} \tag{2}$$

U is the velocity, $R_{ij} = \rho u_i u_j$ is the Reynolds stress tensor (symmetric), with 6 independent components. There are in total ten unknowns for only four equations. Therefore, it is necessary to introduce other equations to solve the system. The turbulent viscosity with the standard $k - \varepsilon$ turbulence model are integrated. This model is based on the Boussinesq hypothesis that expresses R_{ij} as the average-strain rate of the average velocity field:

$$u_i u_j = -\nu_t \left(\frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_i} \right) + \frac{2}{3} k \delta_{ij} \tag{3}$$

v_t is the turbulent viscosity. The system of equation to be solved becomes:

$$\frac{\partial U_i}{\partial x_i} = 0$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + (\nu - \nu_t)\frac{\partial^2 U_i}{\partial x_i \partial x_j} - \frac{2}{3}\frac{\partial k}{\partial x_i}$$

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\nu + \frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x_j} \right) + \nu_t \left(\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_j}{\partial x_i} \right) - \varepsilon$$

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\nu + \frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j} \right) + C_{\varepsilon 1}\nu_t \frac{\varepsilon}{k}\left(\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_j}{\partial x_i} \right) - C_{\varepsilon 2}\frac{\varepsilon^2}{k}$$

k is the turbulent kinetics and ε is the dissipation rate, given by the following equations.

$$k = \frac{3}{2}\left(I u_j \right)^2 \quad \text{and} \quad \varepsilon = \frac{C_\mu^{3/4}}{L} k^{3/2} \tag{4}$$

I is the turbulence intensity set to 0.01 and L is the length scale set to 0.001 m. The standard coefficients are given in Table 1 for the $k - \varepsilon$ model [2].

For nitrogen, the boiling temperature is $-196°C$. The change of state is subjected to heat of vaporization. After the passage of LN2 on the wall, a heat transfer will take place. It is essential to understand what is happening inside the pipe when LN2 is in contact with the internal-surface. Temperature evolution of the fluid was given by the heat equation:

$$-\rho\frac{dH}{dt}+div(\lambda grad(T))+Q=0 \quad (5)$$

ρ is the density, H is the enthalpy and Q is the internal heat.

Table 1. Constants for the $k-\varepsilon$ turbulence model.

$C_{\varepsilon 1}$	1.44
$C_{\varepsilon 2}$	1.92
σ_{ε}	1.3
σ_k	1
C_{μ}	0.09

As the system chills, and as wall temperature decreases, the system approaches the Leidenfrost point [10, 11]. Heat transfer is a minimum at the Leidenfrost point due to the inefficient heat transfer between cold gas and wall. The next regime is the transition boiling in which the heat flux attend critical point. The next region is characterized by liquid nucleate boiling regime. Heat transfer is a maximum at the point of critical heat flux due to the highly efficient cooling process of boiling liquid through the use of sensible and latent heat and due to the fact that the insulating gas layer is absent while the temperature difference between wall and fluid is the greatest.

3. Modeling and analysis of LN2 flow inside a pipe

3.1. Model description and parameters

The current study focuses on modeling of LN2 flow through a pipe of 400 mm of length, 8 mm of diameter. The developed model is represented in Fig. 2. It is used to analyze the variation of the temperature and pressure inside the pipes. Finite difference method is applied to solve the two Navier-Stokes and heat equations.

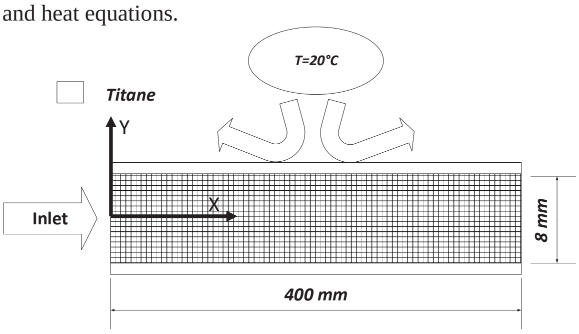

Fig. 2. Model of LN2 flow inside a pipe.

The pipe is discretized in N nodes as shown in Fig. 2.

$$\frac{T_m^{n+1}-T_m^n}{\delta t}-k\frac{T_{m+1}^n-2T_m^n-T_{m-1}^n}{\delta x^2}=0 \quad (6)$$

The finite difference method begins with the discretization of space and time. This simplifies the system considerably, since instead of tracking a smooth function at an infinite number of points, one just deals with a finite number of temperature values at a finite number of locations and times. To solve such a system, it will be easier to create matrix system. Once the system is created, Matlab starts the calculations and after each iteration, it compares the results whether it is true or not.

3.2. Results

The inlet temperature and pressure of the LN2 is 77 K and 2 bar respectively. The temperature of the external surface of the pipe is considered equal to the ambient temperature, which is 293 K (20 °C).

Fig. 3 shows the evolution of the temperature of LN2 inside the pipe along its length.

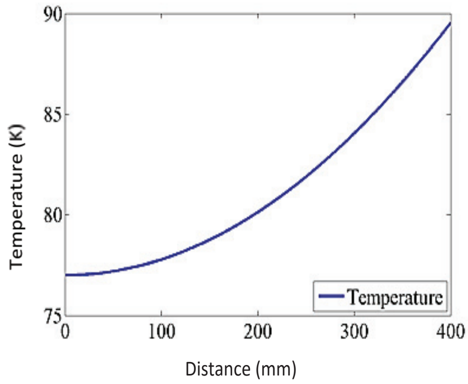

Fig. 3.Evolution of the temperature of the liquid nitrogen (LN2) inside the pipe.

As shown in this figure, the flow temperature increases from 77 K at inlet to 90 K at outlet. Due to the heat transfer from surroundings to the liquid, a gas phase mixed with the liquid one is created and consequently the flow temperature rises.
In this part, equations expressing the way of the flow of cryogenic nitrogen has been developed in 1D. General flows are three dimensional, but many of them may be studied as if they are one dimensional. Whenever a flow in a tube is considered, if it is studied in terms of mean velocity, it is a one dimensional flow. In the following parts, a finite element modeling using STAR-CCM+ based on the discretization of the equation above (Eq. 6) is presented. The results obtained will be compared with those obtained by Matlab.

4. Modeling and simulation of LN2 flow in a pipe and tool channels using CFD

4.1. Model description and parameters

The current study focuses on the simulation of LN2 inside a pipe using the model presented in Fig. 2.

This model is developed to simulate the LN2 flow through pipe then through tool channels with different geometries (Fig. 4 and Fig. 5). It will be also used to describe the boiling phenomena which is caused by the heat transfer from the pipe wall to the LN2 [5, 6, 9]. The present model considers a smooth wall surface, so it does not include the effect of surface roughness. The main objective is to evaluate the influence of several parameters on the LN2 flow through a pipe and then through tool channels with different geometries. The model should be able to predict the amount of liquid and gas phases at outlet knowing the inlet parameters (flow temperature, pressure, velocity, volume fraction).

Since a lot of simulation should be carried out, Matlab software will be also used to generate simulations.

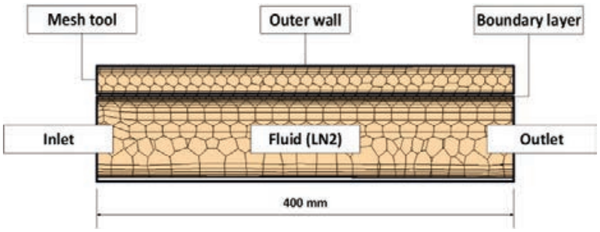

Fig. 4. Meshing model for the pipe.

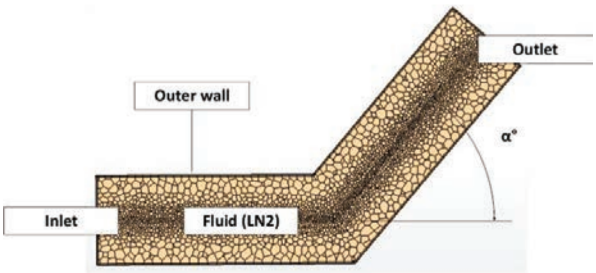

Fig. 5. Meshing model for the tool channels.

As far as the straight pipe flow simulations are concerned, an axisymmetric LN2 flow model inside a pipe submitted to an external ambient temperature of 293 K (20 °C) is proposed. Three inlet pressures and three liquid/gas volume fraction values are considered: i) pressure: 2, 4 and 6 bars; ii) volume fractions: 50%, 70% and 90%.

As far as the tool channels flow simulations are concerned, the only geometrical parameter that is varied is the shape of the pipe axis, which is composed by two straight sections making an angle α between them varying from 50° to 90° (see Fig. 5). Table 2 summarizes the inlet parameters.

Table 2. Inlet parameters of the LN2 flow in the pipe and tool channels.

Pressure (bar)	2	4	6
Volume fraction (%)	90	70	50
Temperature (K)	77	77	77
Velocity (m/s)	20	20	20

Fig. 4 and Fig. 5 show the model's geometry and mesh used in the simulations. This model is composed by LN2 flowing inside the pipe and by tools channels surfaces (outer wall) (Fig. 6). The system is initially at ambient temperature, which is significantly warmer than the cryogenic fluid saturation temperature. As a result, chill down begins in the vapor film boiling regime.

The boiling phenomena is described using Rohsenow correlation [12]:

$$\frac{q}{A} = \mu_L h_{fg} \left(\frac{g(\rho_L - \rho_v)}{\sigma} \right)^n \left(\frac{C_{fg}(T_w - T_{sat})}{C_{sf} h_{fg} \Pr_L^{1.7}} \right)^m \quad (7)$$

q is the heat flux, A is the contact area between the bubble and the liquid, C_{sf} is the empirical coefficient in the rohsenow expression for the wall heat flux, varies with the liquid-surface combination, C_{fg} is the fraction of the wall heat flux due to boiling that is used in creation of vapor bubbles and σ is the surface tension.

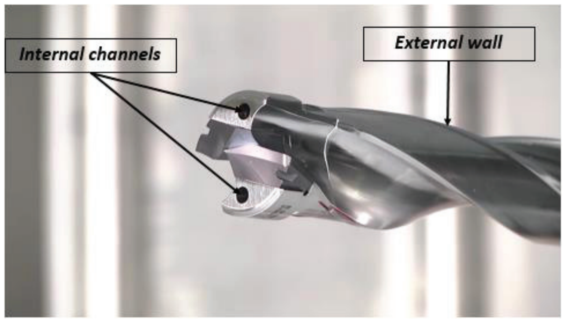

Fig. 6. Internal channels in milling tools.

The STAR-CCM+ preprocessor uses an automatic mesh generator which allows the control of the various modes of mesh generation. The base size of the cell is 0.001 m and the number of prism layer is 10. This approach will be coupled with automation. Many simulation will take place thus it is important to optimize their number. Optimizations studies are always done in order to save the number of simulations. Therefore, it is essential to automate the steps necessary to launch these simulations: i) The pre-processing, which consists of an update of the parameters in the finite element models, ii) The launch of simulations and iii) The post-processing which extracts information. Each time one of the parameters (temperature, volume fraction, velocity) will be changed. STAR-CCM+ software gives the ability to create a java file called "Macro". This file record all the simulation's steps (geometry and mesh definition, boundary conditions, inlet parameters, etc.). Then Matlab will copy the files, modify them and run the simulations.

4.2. Results

4.2.1. Evolution of the different flow parameters inside the pipe

To show the evolution of the temperature inside the pipe, these parameters are considered: 70% as liquid/gas volume fraction, 4 bar as inlet pressure and 20 m/s as a velocity of the fluid. Fig. 7 shows this evolution in transient (10 s) and steady regimes (200 s).The objective is to show the different phase of LN2 inside the pipe. The evolution of the temperature is presented in Fig. 8. Fig. 8 shows the evolution of three parameters (temperature, pressure and volume fraction) inside the pipe. The main objective here is to find a correlation between the wall temperature and the fluid behavior. Inlet pressure is 4 bar, volume fraction at the inlet is 70 %.

In the transient regime, the outlet temperature increases significantly reaching 84 K, while pressure and volume fraction decrease. In the steady regime, the temperature gradient decreases and become almost like the inlet temperature.

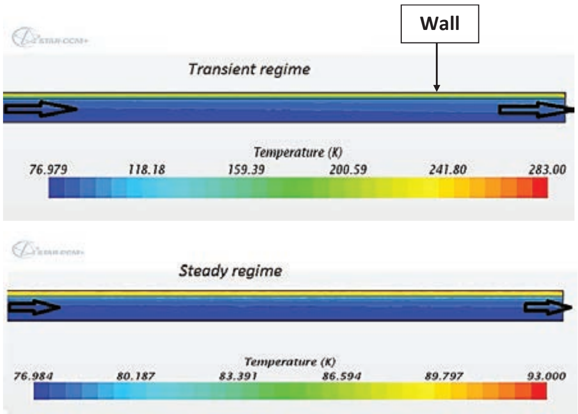

Fig. 7.Evolution of the temperature of the liquid nitrogen (LN2) inside the pipe in transient and steady mode.

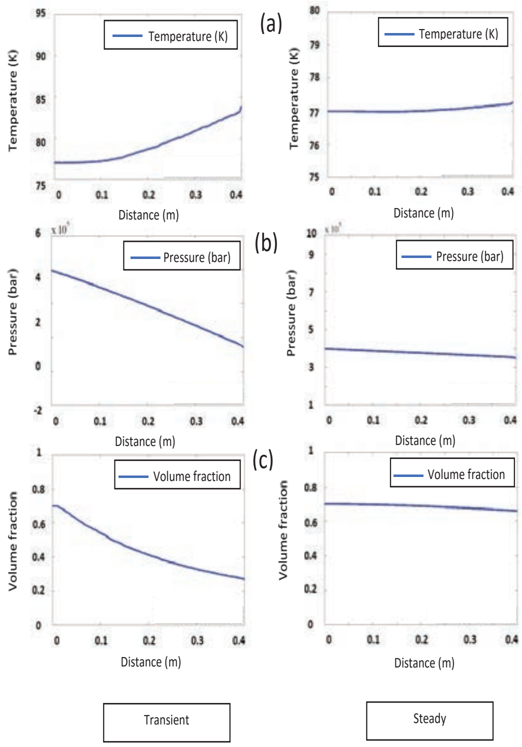

Fig. 8 Evolution of (a) the temperature, (b) the pressure and (c) the volume fraction in transient then steady mode.

4.2.2. Influence of the liquid/gas volume fraction

Fig. 9 shows the effect of the liquid/gas volume fraction on the fluid temperature at the pipe outlet. It represents the evolution of the temperature along the pipe length for two different pressures (2 and 6 bar) and a liquid/gas volume fraction of 50%, 70% and 90%.

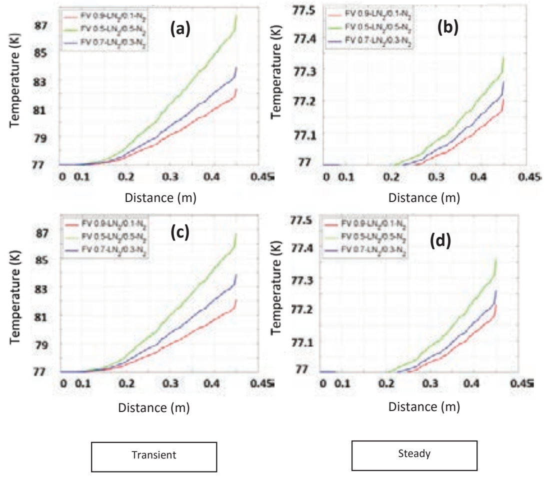

Fig. 9.Effect of volume fraction respectively for (a, b) a pressure of 2 bar and (c, d) 6 bar for transient then steady mode.

The best inlet pressure and liquid/gas volume fraction values which give a minimum temperature and maximum pressure at the outlet, are 6 bar and 90% respectively. These values will be used to run the simulation of the flow inside tool channels.

4.2.3. Influence of the tool channels geometry

Using the best pressure and liquid/gas volume fraction inlet values determined previously, the main goal now is to analyze the influence of the tool channels geometry on the flow behavior. Concerning to the channels geometry, only the shape channel axis characterized by the angle α is varied.

Fig. 10 show the flow temperature along the channel for an inlet liquid/gas volume fraction of 70%, a pressure of 6 bar, a velocity of 20 m/s and an angle varying from 50° to 90°.

It can be seen in Fig. 10 that the temperature is proportional to the angle α. As this angle increases from 50° to 90°, the temperature in the outlet increases from 83 K to 85 K (for the angles which are less than 50°, this variation is not relevant). This increase causes the reduction of the liquid/gas volume fraction at the outlet. This is explained by the phenomenon of shock; for compressible flows, there is singularities where the characteristics of the flow varies strongly and quickly in order to adapt to the constraints. Therefore, the flow's parameters (velocity, pressure, temperature, etc.) will be changed. Therefore, it is recommended to use angles which are less than 50°.

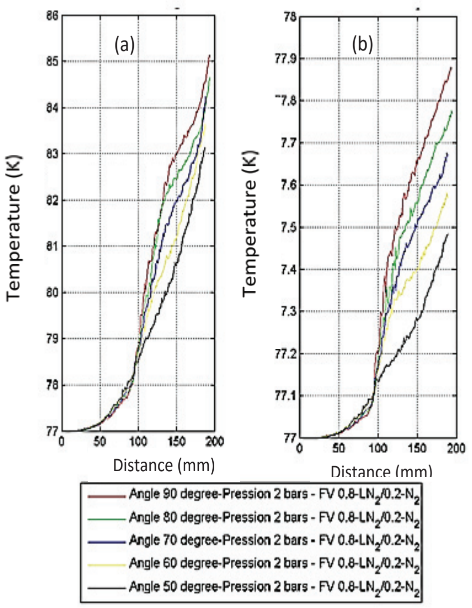

Fig. 10.Evolution of the temperature of the liquid nitrogen (LN2) inside milling channels (a) in transient mode and (b) in steady mode.

4.3. Comparison between Matlab and STAR-CCM+ model's results

This section deals with the comparison between the results obtained by the Matlab and STAR-CCM+ models. Fig. 11 shows two temperature curves obtained from each model, using the same parameters as follows: 90% as liquid volume fraction, 6 bar as pressure. The relative error between the two curves is about 0.01 and it is due to the method of finite differences and to the absence of the boiling phenomenon in Matlab model. Besides that, it can be concluded that the Matlab model predicts very well what happens inside the pipe for temperature.

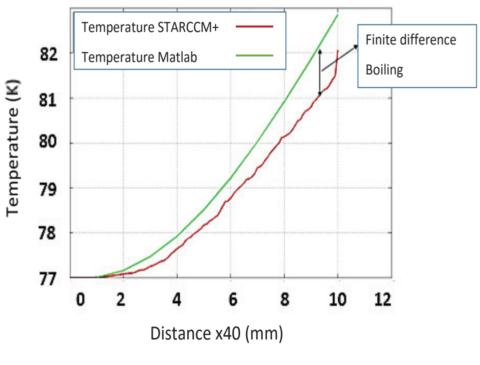

Fig. 11.Comparison between Matlab and STAR-CCM+ model's results.

5. Conclusion

This paper studied the effect of the environment on the LN2 flow. The study of the LN2 flow inside milling channels is based on the identification of flow modes and inlet parameters. This paper lead to the development of a numerical model. The tests at known temperature, velocity, pressure and volume fraction carried out on a straight pipe allowed to identify the relevance of the boiling laws to be used, the flow patterns and initial parameters of the LN2.

It appears that the LN2 behavior is sensitive to the variation of the volume fraction and pressure. Compared with previous work, it is possible to fix a volume fraction of 90% and a pressure of 6 bar.

It will be also carried out a study dealing with the behavior of nitrogen in tools dedicated to milling. The effect of the angle on the outlet parameters was analyzed via the flow inside channels with different geometries. It is important to note that the flow is proportional to the angle. The simulations show that the cryogenic assistance will give a high performance for angles less than 50°.

References

[1] Podgornik, B., I. Paulin, B. Zajec, S. Jacobson, and V. Leskovšek. "Deep Cryogenic Treatment of Tool Steels." Journal of Materials Processing Technology 229 (March 2016) : 398–406.

[2] Randall F.Barron, Cryogenic Heat Transfer, series in Chemical and Mechanical Engineering.

[3] Davoudinejad, A., E. Chiappini, S.Tirelli, M.Annoni, and M.Strano. "Finite Element Simultion and validation of chip Formation and cutting forces in Dry and cryogenic cutting of Ti-6Al-4V." Procedia manufacturing 1 (2015): 728-39.

[4] Olewski, Tomasz, Sam Mannan, and Luc Véchot. "Validation of liquid Nitrogen Vaporisation Rate by Small Scale Experiments and analysis of the conductive Heat Flux from the concrete." Journal of Loss Prevention in the Process industries 35 (May 2015) : 277-82.

[5] Liu, Yi, Tomasz Olewski, and Luc N.Véchot. "Modeling of a cryogenic Liquid Pool Boiling by CFD Simulation." Journal of Loss Prevention in the Process industries 35 (May 2015) : 125-34.

[6] P. Lequien, G. Poulachon, J.C. Outeiro, J. Rech,"Characterization of liquid nitrogen jet in cryogenic machining" 13th International Conference on High Speed Machining.

[7] A. Duchosal, R. Serra, R. Leroy. "Static numerical simulation of oil mist particle size effects on a range of internal channel geometries of a cutting tool used in MQL strategy." International Journal Eng. Sci. Innov. Technol. 3 (2014) 43-59.

[8] A. Duchosal, R. Leroy, L. Vecellio, C. Louste, N. Ranganathan, " An experimental investigation on oil mist characterisation used in MQL milling process." Int. J. Adv. Manuf. Technol. 66 (2012) 1003-1014.

[9] Tae Seon Park, "LES and RANS simulations of cryogenic liquid nitrogen jets." The journal of supercritical fluids 72 (2012) 232-247.

[10] Zhang, Xiaobin, Jianye Chen, Wei Xiong, and Tao Jin."Visualisation Study of Nucleate Pool Boiling of Liquid Nitrogen with Quassi-Steady Heat Input." Cryogenics 72 (December 2015) : 17-21.

[11] Fang, Xiande, Alenxander M. Sudarchikov, Yafeng Chen, Anqi Dong, and Run Wang. "Experimental Investigation of Saturated Flow Boiling Heat Transfer of Nitrogen in a Macro-Pipe." International Journal of Heat and Mass transfer 99 (August 2016) : 681-90.

[12] Rohsenow, W.M.1952. "A method of correlation Heat Transfer Data for Surface Boiling of Liquid", Trans. ASME, 74, 969 .