18th Brazilian Congress of Thermal Sciences and Engineering
November 16-20, 2020 (online)

ENC-2020-0095
EVALUATION OF DIFFERENT TURBULENCE MODELS FOR THE CHARACTERIZATION OF SHEAR STRESSES IN THE CENTRAL VENOUS ACCESS FOR HEMODIALYSIS

Saulo de Freitas Gonçalves
Matheus Carvalho Barbosa Costa
Universidade Federal de Minas Gerais, Avenida Antônio Carlos, 6627, Belo Horizonte, Minas Gerais, Brasil
saulodfg@gmail.com
mt94_carvalho@hotmail.com

Thabata Coaglio Lucas
Universidade Federal dos Vales do Jequitinhonha e Mucuri, Rodovia MGT 367 – km 583, 5000, Diamantina, Minas Gerais, Brasil
thabataclucas@gmail.com

Jonathas Haniel
Universidade Federal de Minas Gerais, Avenida Antônio Carlos, 6627, Belo Horizonte, Minas Gerais, Brasil
jonathashaniel@hotmail.com

Mário Luis Ferreira da Silva
Universidade Federal de Minas Gerais, Avenida Antônio Carlos, 6627, Belo Horizonte, Minas Gerais, Brasil
marioluisfs@gmail.com

Rudolf Huebner
Universidade Federal de Minas Gerais, Avenida Antônio Carlos, 6627, Belo Horizonte, Minas Gerais, Brasil
rudolf@demec.ufmg.br

Edna Maria de Faria Viana
Universidade Federal de Minas Gerais, Avenida Antônio Carlos, 6627, Belo Horizonte, Minas Gerais, Brasil
ednamfv@ufmg.br

Abstract. *Numerical models based on computational fluid dynamics (CFD) allow the investigation of hemodynamic factors that lead to thrombus formation and propagation in a non-invasive and relatively low cost way. The present study aims to develop a numerical model, based on CFD to compare the use of different turbulence models for the characterization of shear stresses and turbulence intensity near the tip and lateral orifices of a central venous catheter for hemodialysis (CVC). All models evaluated, except k-ε, presented similar results, both in terms of average values and in terms of the general distribution of shear stresses in regions of low turbulence intensity. However, in regions with higher turbulence intensities, the results were considerably different. The k-ω SST and k-ω SST-transition models resulted in the same distribution of turbulence intensity and shear stress near the tip and venous orifices of the catheter, although the transition model has resulted in the characterization of higher values of those variables. However, both models resulted in great agreement with experimental studies that have evidenced the contorted deposition of platelet layers near the venous orifices of the catheter, which may be associated with the presence of vortices in the region.*

Keywords: *Computational fluid dynamics, Central venous access for hemodialysis, Central venous catheter, Turbulence models, Blood Flow, Thrombus formation, Turbulence intensity.*

1. INTRODUCTION

Although the blood flow is essentially laminar in almost the entire circulatory system, under pathological condition or when in contact with artificial devices, the blood flow can become turbulent (Ha et al., 2018; Yen et al., 2014; Mahalingam et al., 2016; Wong et al., 2015).

Numerical models based on computational fluid dynamic (CFD) appears as a useful tool for observation and prediction of the pressure and velocity fields and made possible modeling turbulent energy pattern and others parameters related to the formation and propagation of thrombi in an relatively inexpensive and non-invasive way. In this context, the numerical analysis opens a new perspective for the prediction of flow critical regions, besides enables the development and improvement of medical devices and clinical procedure (Huebner *et al.*, 2010).

Although arteriovenous fistulas are recommended as the preferred choice of vascular access for hemodialysis patients, several patients use central venous catheters (CVC). Catheters have side holes fencing different directions to return the blood for the patient veins. Studies have shown that a region of eddies and separation was mainly found in the side holes region, making important to model turbulence in the central venous access (Lucas *et al.*, 2014).

The blood flow in internal jugulars veins and superior vena cava is normally laminar, however, the central venous catheter (CVC) insertion can generate turbulent pattern in some regions, due to the decrease of free shear flow section or by localized effect of the CVC outflow jets. In respect of turbulent models, the k-ω SST model has being used systemically to model turbulence in blow flow, but there is no full consensus whether this is the most suitable model.

In this study, CFD analysis was used with the aim to characterize and to compare the turbulent flow profile around the tip and lateral orifices of CVC, obtained with the use of different turbulence models: *k-ε, k-ω, k-ω SST and k-ω SST-transition*. The shear stress, turbulence intensity, and the general pattern of flow field obtained with the use of each turbulence model were analyzed and compared.

2. MATERIALS AND METHODS

The *Ansys Fluent 19.2* software was used for numerical calculation of the continuity, momentum and turbulence models transient equations. The blood flow was considered incompressible and modeled as non-Newtonian (*Carreau-Yasuda* model was employed).

2.1 Geometry

The geometry of internal jugulars vein and superior vena cava was obtained from handling computed tomography (CT) images of a male healthy patient of 74 years old. The procedure was approved by the "Comitê de Ética em Pesquisa/Universidade Federal de Minas Gerais (CEP-UFMG) under process number CAAE 02405712.5.1001.5149".

In the *InVersalius3®*, a medical image processing software, it was imported the DICOM (Digital Imaging and Communications in Medicine) files. The data files were processed in order to acquire only the region of interest. Figure 1a illustrates this procedure, with the region of interest highlighted in green. The undesirable parts, such as bone, tissue surface, among others, were removed.

Cleaning, smoothing and removal of surface imperfections to improve the surface quality were carried out in Autodesk Meshmixer® software. Figure 1b shows the geometry resulted of this process.

In order to facilitate manufacturing the experimental model and aiming to improve optical access for future studies, the excess of curvature of the geometry was removed. Basically, the geometry was edited in *ANSYS SpaceClaim*, to become symmetrical in relation to a central plane (XZ plane, as shown in Fig. 1c).

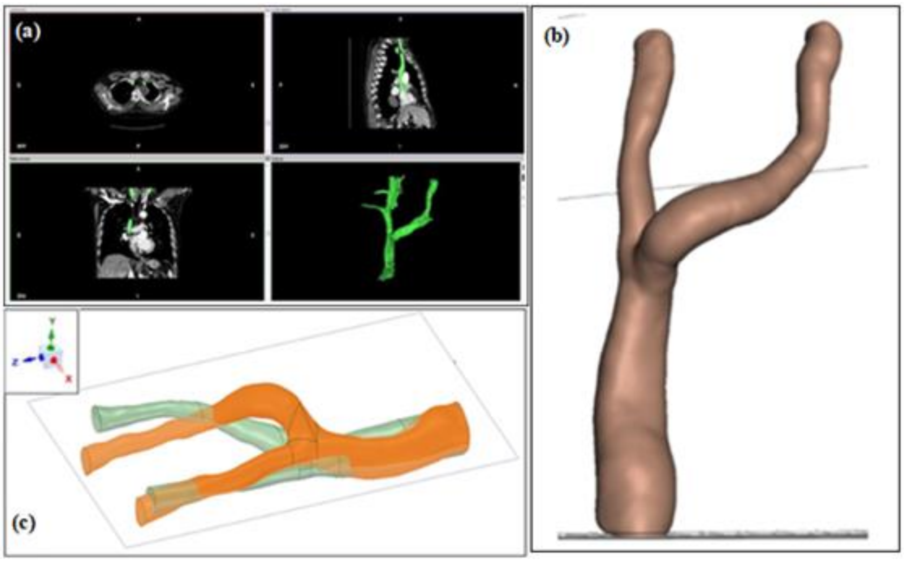

Figure 1. Geometric model of the central veins. Images cleaning and edition in *InVesalius3®* (a). Generation and improvement of the surface quality in *Meshmixer®* (b). Geometry edition in *ANSYS SpaceClaim* (c).

In addition, *SolidWorks (SolidWorks, Inc., Concord, MA, USA)* software was used to develop a CVC similar to clinical model *MedCOMP/HEMO-CATH (Harleysville, PA, USA)*. Figure 2 shows the geometrical model developed for the CVC and Fig. 3 shows the geometrical model of the vein with the CVC inserted (geometric model used in the simulations).

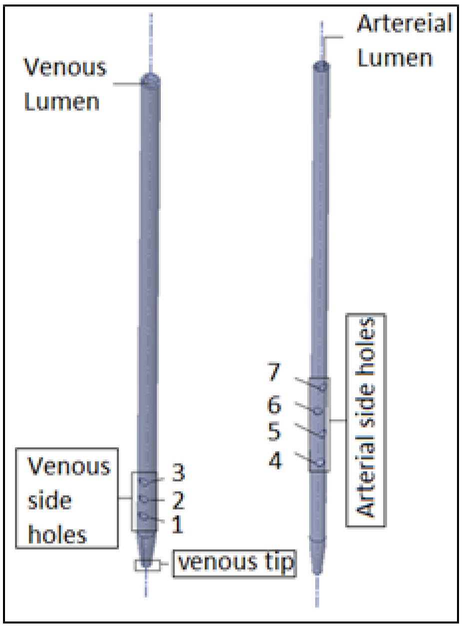

Figure 2. Geometric model of the central venous catheter used in the simulations.

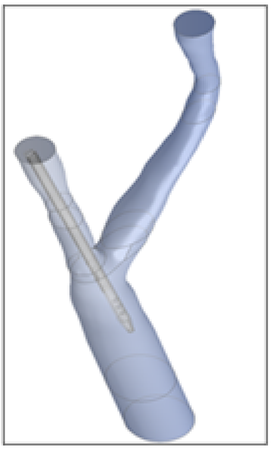

Figure 3. Geometric model of the central venous system with the catheter inserted.

2.2 Set-up

Numerical simulations were performed in transient regime, covering a total time interval corresponding of two complete cardiac cycles of 0.8 s. To meet the adaptive time step criterion, called the *Courant-Friedrichs-Lewis Condition (CFL)*, 800 time steps of 0.002 s were considered. The flow inside the catheter was considered steady, with a total flow rate of 0.00525 kg/s, as occurs in the clinical hemodialysis procedure. The pressure and velocity coupling was modeled by Coupled algorithm and the second order Upwind discretization scheme was adopted for convective terms.

2.3 Mesh convergence test

The mesh test was performed by doubling the number of elements in each mesh until the equivalence between the numerical solutions was verified. The mesh was considered satisfactory when residual differences less than 5% were found in the mean values of pressure and velocities, in four planes close to the tip and lateral orifices of the CVC. In addition, the tests were performed in a transient regime, as well as physiological simulations. The mean values of pressure and velocity over cardiac cycle were considered in each plane.

2.4 Boundary conditions

The boundary conditions used in this work were developed from a literature review of studies. The mean values of the curves by (Markl et al., 2011; Mynard et al., 2015) were used to estimate the total flow in the superior vena cava. For this purpose, the flows in the right and left jugular veins were considered the same and equal to half of the total flow in the superior vena cava. This simplification is justified by the scarcity of data in the literature, in the absence of consensus regarding the flow values in the right and left jugular veins (Hung, 2007; Marr et al., 2018) and the possible similarity of the flow values between these two veins (Gonçalves *et al.*, 2019; Ciuti *et al.*, 2013).

With the estimation of the pulsate flow in each jugular veins and their respective area, was possible to develop the inlet velocities curves for the right and left jugular veins. As a boundary condition at the outlet of the superior vena cava, the pressure curve in the right atrium was adopted, according to (Mynard et al., 2015). Figure 4 shows the curves developed as boundary conditions in this study. It's important to mention that they were implemented in *Ansys Fluent 19.2* software using a language C complier software.

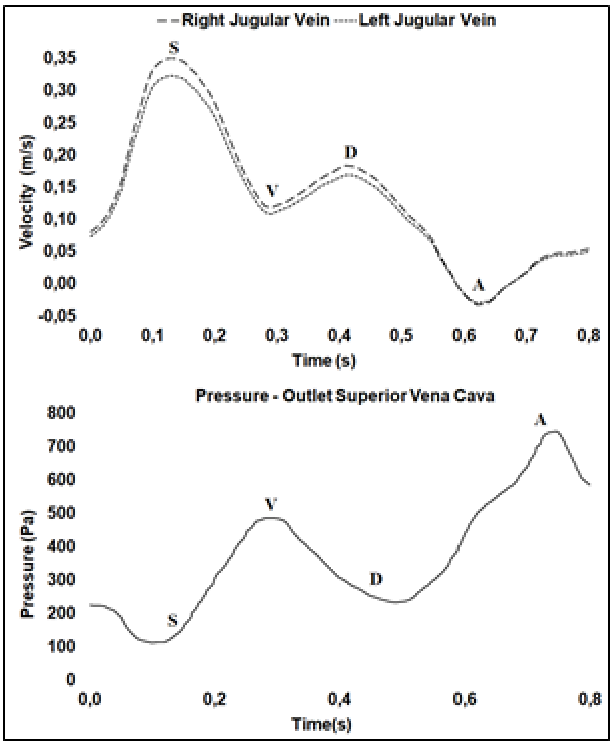

Figure 4. Velocity inlets and pressure outlet curves used as boundary conditions representing the cardiac cycle.

In Fig.4 the (A) wave occurs when the atrium contracts, increasing atrial pressure. At the same time, the blood is propelled in a retrograde direction toward the veins. When the tricuspid valve closes, the systole wave (S) occurs. The transitional (V) wave corresponds to atrial overfilling against a closed tricuspid valve, anticipating the opening of the valve in diastole (D). The inlet and outlet waveforms designed for this study showed a correlation between flow velocity in the internal jugular veins and the pressure in the right atrium.

It's important to mention that using this boundary conditions the Reynolds number in the outlet of the Superior Vena Cava varies between 0 and 1259 during the cardiac cycle.

2.5 Fluid rheology

The *Carreau-Yasuda* model was used to represent the rheological behavior of blood. Table 1 indicates the properties used in the numerical simulation. It was set a valor of 1060 kg/m3 for the blood density (Lucas *et al.*, 2014).

Table 3. Fluid properties, numerical simulations (Lucas et al., 2014).

Rheological Model	Maximum Viscosity (kg/m.s)	Minimum Viscosity (kg/m.s)	Power-Law index	Time Constant (s)
Carreau-Yasuda	0.056	0.0035	0.3568	3.313

3. RESULTS

Figure 5 shows, for the time of highest and lowest velocities values in the cardiac cycle, the regions of highest turbulence intensity in the domain of simulations, obtained with the use of *isovolumes* of turbulence intensity. Each *isovolume* represents three-dimensional regions, in the simulations domain, with turbulence intensities above specified values.

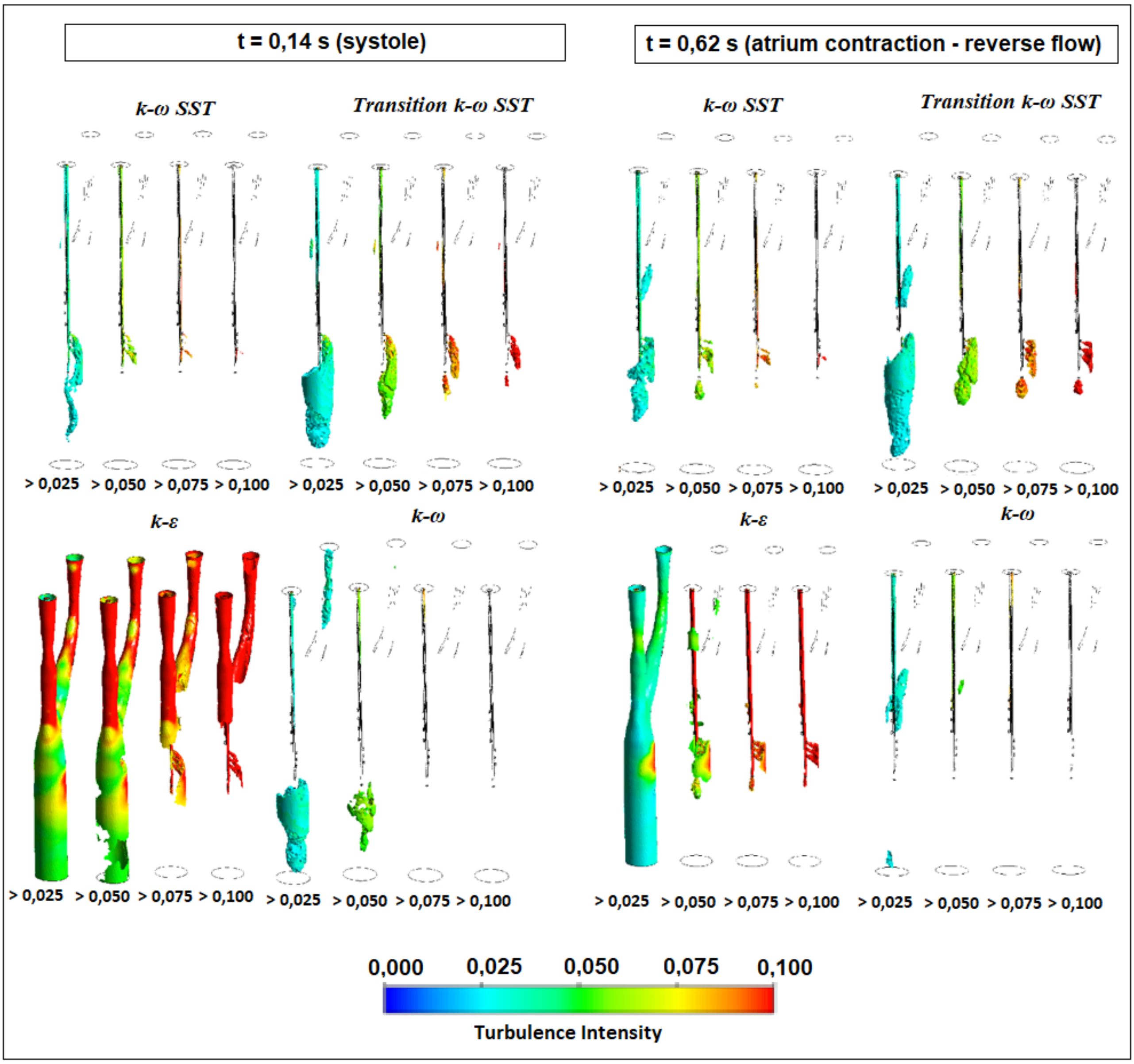

Figure 5. *Isovolumes* of turbulence intensity.

Figure 5 shows that the use of all turbulence models, except *K-ε*, resulted in regions of greater turbulence intensities close to the tip and venous orifices of the catheter, which suggests that, in the central venous access for hemodialysis, turbulence is a localized phenomenon, mainly associated with the outflow jets of the catheter. This result is in agreement with studies in the literature that identified a higher occurrence of vortex formation, flow separation and recirculation, close to the CVC venous orifices (LUCAS *et al.,* 2014). It is important to note that studies involving the histological and morphological analysis of thrombotic tissue have shown the contorted deposition of platelet layers, close to the venous orifices of the CVC, which, possibly, would be associated with the vortices and recirculation present in the region (LUCAS *et al.*, 2014).

The use of *K-ε* model resulted in higher turbulence intensities close to the vein walls, even close to the vein walls left jugular, which is an unlikely condition, once venous flow is essentially laminar and turbulence may be associated with pathological conditions or the presence of clinical devices. (MAHALINGAM *et al.,* 2016). The low precision of the *K-ε* model in characterizing the flow in areas of adverse pressure gradient, or close to the boundary layer (MALALASEKERA, H, K, 2005), appears as a possible explanation for the discrepancy in the results obtained with this model.

The use of *K-ω* model resulted in lower turbulence intensities close to the tip and lateral orifices of the CVC. It can be probably explained by the limitation of this model in characterizing the flow in regions distant from the boundary layer (MALALASEKERA, H, K, 2005). In general, it can be said that this model was not able to adequately characterize the turbulence close to the venous orifices of the catheter, which contradicts the results obtained in studies involving the histological and morphological analysis of thrombotic tissue in the region (LUCAS et al., 2014).

Figure 5 shows that the general distribution of the *isovolumes* of turbulence intensity, obtained with the use of the two hybrid models (*k-ω SST and k-ω SST of Transition*) are very similar. However, the extent of turbulence intensity isovolumes is significantly greater when using a transition model. In other words, this means that the use of the transition model resulted in the characterization of higher values of turbulence intensity, mainly near the tip and venous orifices of the CVC.

Figure 6 indicates the shear stress field, in addition to the mean value of this distribution, in planes that cross the tip and lateral holes of the CVC, for the different turbulence models. Must be emphasized that the results were obtained for instant of time of t = 0.14 s, which is the velocity peak of the cardiac cycle.

Figure 7 indicates the general distribution of the shear stresses obtained with the use of each turbulence model, as well as the average values of these shear stresses in the planes defined near the tip and lateral orifices of the catheter.

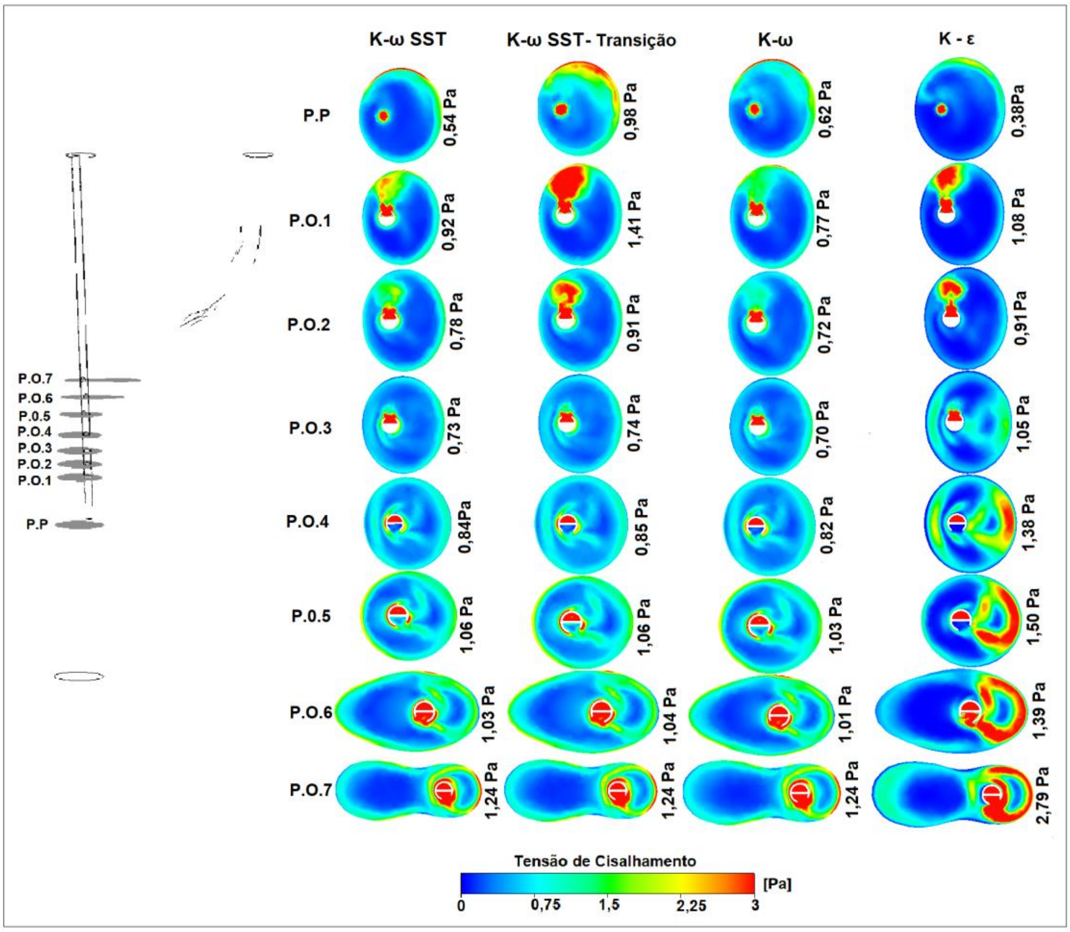

Figure 7. Shear stresses in different planes near the tip and lateral holes of the catheter.

It is observed that the P.P, P.O.1 and P.O.2 planes showed significant differences both in the general distribution of the shear stress field and in the mean values of this variable in each plane. This occurs because these planes present a greater turbulence intensity, which suggests that turbulence occurs in these regions. Besides that, other planes, from P.O.3 to P.O.7, did not show significantly different results for the models *k-ω, k-ω SST* and *k-ω SST-transition*. Nevertheless, the *k-ε* model resulted in discrepant results because this model is ineffective to acquire accuracy results in regions with adverse pressure gradient and boundary layer detachment.

It is noteworthy that similar results were obtained at other times of the cardiac cycle. This probably happened because the turbulence in the central venous access may be associated with the outflow jets of the catheter and not with the venous flow itself.

4. CONCLUSIONS

CFD analysis allows the investigation of hemodynamic factors that lead to the formation and propagation of thrombi in a non-invasive and relatively inexpensive manner. Thus, this tool can be applied to guide studies in order to improve the design of clinical devices.

All models evaluated, except *k-ε*, presented similar results, both in terms of average values and in terms of the general distribution of shear stresses in regions of low turbulence intensity. However, in regions with higher turbulence intensities, the results were considerably different among all models evaluated.

The use of *K-ω* model resulted in lower turbulence intensities close to the tip and lateral orifices of the CVC. It can be explained by the limitation of this model in characterizing the flow in regions distant from the boundary layer. In general, it can be said that this model was not able to adequately characterize the turbulence close to the venous orifices of the catheter.

The use of the two hybrid models (*k-ω SST and k-ω SST of Transition*) resulted in a very similar distribution of shear stress and turbulence intensity. Although, the transition model resulted in the characterization of higher values of those 2 variables. this may be associated with greater robustness of this model and the ability to characterize the transition to turbulence.

shows that the general distribution of the *isovolumes* of turbulence intensity, obtained with the use of the two hybrid models (*k-ω SST and k-ω SST of Transition*) are very similar. However, the extent of turbulence intensity

5. ACKNOWLEDGEMENTS

The authors would like to thank FAPEMIG (APQ-02735-17) for their support in this project.

6. REFERENCES

Ciuti, G. et al. Differences between Internal Jugular Vein and Vertebral Vein Flow Examined in Real Time with the Use of Multigate Ultrasound Color Doppler. p. 2000–2004, 2013

Gonçalves, S. D. F. et al. Comparison of different rheological models for computational simulation of blood flow in central venous access. Artigo apresentado no 25º Congresso Brasileiro Internacional de Engenharia Mecânica (COBEM), 2019.

Ha, H. et al. Age-related vascular changes affect turbulence in aortic blood flow. Frontiers in Physiology, v. 9, n. JAN, p. 1–10, 2018.

Huebner,R. et al. Turbulência e o sitema cardiovascular. Associação Brasileira de Engenharia e Ciências Mecânicas (ABCM) . Livro Turbulência, v.7, 2010.

Hung, C. H. I. H. I. N. G. C. et al. Flow volume in the jugular vein and related hemodynamics in the branches of the jugular vein. v. 33, n. 4, p. 500–505, 2007.

Lucas.T.C. et al. Blood Flow in Hemodialysis Catheters:A Numerical Simulation and Microscopic Analysis of In Vivo-Formed Fibrin. Artificial Organs 2014.

Mahalinga, A. et al. Numerical analysis of the effect of turbulence transition on the hemodynamic parameters in human coronary arteries. Cardiovascular Diagnosis and Therapy. 01 Jan 2016. p. 208-220

Malalasekera, H, K, V. An Introduction to Computational Fluid Dynamics, 2005.

Markl, M. et al. Time-resolved three-dimensional magnetic resonance velocity mapping of cardiovascular flow paths in volunteers and patients with Fontan circulation §. v. 39, p. 206–212, 2011.

Marr, K. et al. Jugular Venous Flow Quantification Using Doppler Sonography. Ultrasound in Medicine and Biology, v. 44, n. 8, p. 1762–1769, 2018.

Mynard, J. O. P. M.; MOLICH, J. O. J. S. One-Dimensional Haemodynamic Modeling and Wave Dynamics in the Entire Adult Circulation. v. 43, n. 6, p. 1443–1460, 2015.

Tu, J., Inthavong, K., Wong, K. Computational Hemodynamics - Theoey, Modelling and Applications. [S.l: s.n.], 2015.

Yen, J. H. et al. The effect of turbulent viscous shear stress on red blood cell hemolysis. Journal of Artificial Organs, v. 17, n. 2, p. 178–185, 2014.

7. RESPONSIBILITY NOTICE

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