ENCIT 2018

17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

STREAM FUNCTION-VORTICITY FORMULATION APPLIED IN THE CONJUGATED HEAT PROBLEM USING THE FEM WITH UNSTRUCTURED MESH

L. H. CARNEVALE, lh.carnevale@gmail.com
G. R. ANJOS, gustavo.anjos@uerj.br
N. MANGIAVACCHI, norberto@uerj.br
Group of Environmental Studies for Water Reservoirs - GESAR, State University or Rio de Janeiro - UERJ, Rua Fonseca Teles, 121, 20940- 903, São Cristóvão, Rio de Janeiro, RJ, Brazil

Abstract. *Conjugated heat transfer problems have an extensive number of applications because a lot of times it is necessary to understand how heat convection in a fluid influences the temperature in solid regions. The stream function-vorticity formulation is a useful alternative to solving the Navier-Stokes equations in two dimensional domain. One of its major difficulties is the lack of boundary conditions for vorticity, and several schemes are proposed to satisfy such a condition. In this paper we propose a finite element scheme for solving the couple problem of the stream function-vorticity formulation using linear triangle elements. A possible application of the method is also demonstrated using the calculated velocity field from the formulation in the heat transport equation to study the temperature distribution in a incompressible single-phase fluid medium as a conjugate heat transfer problem. The results obtained were satisfactory when using low Reynolds number.*

Keywords: *Finite Element Method, Conjugated Heat Transfer, Stream Function-Vorticity, Unstructured Mesh*

1. INTRODUCTION

There are many practical applications for problems where heat conduction in a solid region is directly influenced by heat convection in a region with a moving fluid, those applications are called conjugated heat transfer problems. Being able to successfully model and simulate problems with such characteristics has an important role in understanding the mechanics of how the flow of a fluid can influence heat distribution during different types of processes and those results can be used in optimizing practical systems, an example of apllication is in the study of heat exchangers (Song and Li, 2002). Wansophark *et al.* (2005) says that most researchers in the field use the finite difference and finite volume methods for their numerical analysis, however some assumptions on heat transfer coefficients have to be made in order to compute the temperatures along the fluid-solid interface.

In computational fluid dynamics (CFD), solving the incompressible Navier-Stokes equation is of utmost interst in various applications. The stream function-vorticity (ψ, ω_z) formulation is a way to express the Navier-Stokes equation in terms of ψ and ω_z instead of the primitives pressure and velocity (Peeters *et al.*, 1987). The advantage of using the stream function-vorticity formulation is that the pressure does not appear in the equation and, thus, we do not need to solve the pressure-velocity problem, therefore the use of linear element is suitable. The formulation has been proved to be of great use when considering a two dimensional domain (Hoffmann and Chiang, 2000).

Cesini *et al.* (1998) is an example of the finite element method apllied to the stream function-vorticity formulation used in a conjugate heat problem modeling natural convection. Traditional finite-element analyses are based on finite-difference formulas for computing vorticity at walls which limits their use to regular domains as is shown in M.Vynnycky *et al.* (1998).

In this paper we propose an algorithm based on (Salih, 2013) to combine the (ψ, ω_z) formulation results with the heat transport equation using a finite element formulation to solve both equations in the conjugate heat problem. The FEM is implemented by discretizing the governing equations using the Galerkin method with linear triangular elements in an unstructured mesh. We present a heat transfer validation in a two dimensional problem with variable spatial diffusivity in different solid materials along with a convergence study. It is also presented a counter flow heat exchanger simulation with temperature and velocity profiles calculations for two different fluid regions and their solid interface. The presented results show good agreement with the available literature such as in Chen and Han (2000).

2. METHODOLOGY

2.1 Governing Equations

A two dimensional Finite Element Method approach can be employed to analyse different physical phenomena that appear in engineering problems. The analysis of a problem starts with the mathematical modeling, and in this paper we use the heat transport equation (see Eq. 5) over the domain Ω and with a variable thermal diffusivity $\alpha = k/\rho c_p$ (ρ is the density and c_p the specific heat) as a function of space. The proposed method for solving the heat transport equation in a fluid system makes use of the stream function-vorticity formulation (Hoffmann and Chiang, 2000) to obtain a more precise description of the velocity field in a single-phase incompressible flow.

The 2-dimensional set of equations for the fluid flow using the stream-functino vorticity used in dimensional form are presented below:

$$\frac{\partial \omega_z}{\partial t} + \mathbf{v} \cdot \nabla \omega_z = \nu \nabla^2 \omega_z \tag{1}$$

$$\omega_z = \frac{\partial v_y}{\partial x} - \frac{\partial v_x}{\partial y} \tag{2}$$

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = \omega_z \tag{3}$$

$$\frac{\partial \psi}{\partial y} = v_x \ , \ \ \frac{\partial \psi}{\partial x} = -v_y \tag{4}$$

$$\frac{\partial T}{\partial t} + \mathbf{v} \cdot \nabla T = \alpha \nabla^2 T \tag{5}$$

Where T is the scalar field of temperature, ν is the kinematic viscosity, α is the thermal diffusivity, ω_z is the vorticity field, x and y are the independent spatial variables, $\mathbf{v} = (v_x, v_y)$ is the velocity field, ψ is the stream function, and t is the time variable.

2.2 FEM formulation

Equation 6 is a general transport equation of the variable ϕ. Using the semi-discrete Galerkin method, first the equation is multiplied by a weight function w and integrated over the domain Ω.

$$\frac{\partial \phi}{\partial t} + \mathbf{v} \cdot \nabla \phi = \gamma \nabla^2 \phi \tag{6}$$

$$\int_\Omega w \frac{\partial \phi}{\partial t} d\Omega + \int_\Omega w \, \mathbf{v} \cdot \nabla \phi d\Omega - \int_\Omega w \, \nabla \cdot (\gamma \nabla \phi) d\Omega = 0 \tag{7}$$

Using the Green theorem on the diffusive term, Eq. 7 becomes:

$$\int_\Omega w \frac{\partial \phi}{\partial t} d\Omega + \int_\Omega w \, \mathbf{v} \cdot \nabla \phi d\Omega + \int_\Omega (\nabla w) \cdot (\gamma \nabla \phi) d\Omega - \int_\Gamma w \left[(\gamma \nabla \phi) \cdot \mathbf{n} \right] d\Gamma = 0 \tag{8}$$

The integral evaluated over Γ represents the boundary conditions of the problem. Being ne the number of elements on the domain, the weight function and ϕ are approximated by Eq. 9 making use of the called shape function $\mathbf{N}(x, y)$.

$$\begin{aligned} w &\approx \sum_j^{ne} b_j(t) \mathbf{N}_j(x, y) \\ \phi &\approx \sum_i^{ne} a_i(t) \mathbf{N}_i(x, y) \end{aligned} \tag{9}$$

Writing Eq.8 with Eq. 9, and not writing the sum notation:

$$\int_\Omega b_j \mathbf{N}_j \frac{\partial a_i \mathbf{N}_i}{\partial t} d\Omega + \int_\Omega b_j \mathbf{N}_j \, \mathbf{v} \cdot \nabla a_i \mathbf{N}_i \, d\Omega + \int_\Omega \nabla (b_j \mathbf{N}_j) \, \gamma \nabla (a_i \mathbf{N}_i) \, d\Omega = \int_\Gamma b_j \mathbf{N}_j \, [(\gamma \nabla a_i \mathbf{N}_i) \cdot \mathbf{n}] \, d\Gamma \quad (10)$$

Definig the mass matrix $\mathbf{M}$, the stiffness matrix $\mathbf{K}$ as:

$$\int_\Omega \mathbf{N}_j \mathbf{N}_i \, d\Omega = \mathbf{M} \quad (11)$$

$$\int_\Omega \nabla \mathbf{N}_j \, \gamma \nabla \mathbf{N}_i \, d\Omega = \mathbf{K} \quad (12)$$

and

$$\int_\Omega \mathbf{N}_j \, \mathbf{v} \cdot \nabla \mathbf{N}_i \, d\Omega = \mathbf{v} \cdot \mathbf{G} = v_x \mathbf{G_x} + v_y \mathbf{G_y} \quad (13)$$

$$\int_\Gamma b_j \mathbf{N}_j \, [(\gamma \nabla a_i \mathbf{N}_i) \cdot \mathbf{n}] \, d\Gamma = \mathbf{f} \quad (14)$$

substituting Eqs.12, 11, 13 and 14 in Eq. 10:

$$\mathbf{M} \frac{\partial a}{\partial t} + (\gamma \mathbf{K} + \mathbf{v} \cdot \mathbf{G}) a = \mathbf{f} \quad (15)$$

A first-order implicit forward difference approximation was used to discretize the time derivative in Eq. 15, thus the final matricial equation is obtained and presented in Eq. 16.

$$\left(\frac{\mathbf{M}}{\Delta t} + \gamma \mathbf{K} + \mathbf{v} \cdot \mathbf{G} \right) a^{n+1} = \left(\frac{\mathbf{M}}{\Delta t} \right) a^n + \mathbf{f} \quad (16)$$

The same approach was used with Eqs. 5, 1 , 3, 4 and 2. Their matricial form is respectively 17, 18 , 19, 20 and 21.

$$\left(\frac{\mathbf{M}}{\Delta t} + \alpha \mathbf{K} + \mathbf{v} \cdot \mathbf{G} \right) T^{n+1} = \left(\frac{\mathbf{M}}{\Delta t} \right) T^n + \mathbf{f} \quad (17)$$

$$\left(\frac{\mathbf{M}}{\Delta t} + \nu \mathbf{K} + \mathbf{v} \cdot \mathbf{G} \right) w_z^{n+1} = \left(\frac{\mathbf{M}}{\Delta t} \right) w_z^n + \mathbf{f} \quad (18)$$

$$\mathbf{K} \psi = \mathbf{M} \omega_z + \mathbf{f} \quad (19)$$

$$v_x = \mathbf{G_y} \psi \quad , \quad v_y = -\mathbf{G_x} \psi \quad (20)$$

$$\omega_z = \mathbf{G_x} v_y - \mathbf{G_y} v_x \quad (21)$$

2.3 Element Geometry

The algorithm was developed for the use of linear triangular elements in the discretization of the spatial domain and to build the geometric model and mesh of the different spatial domains an open source software called "GMSH" Geuzaine and Remacle (2009) was used. The advantage of using such elements is that the shape function $\mathbf{N}(x, y)$ have a well known form (Eq. 22) as shown in Lewis *et al.* (2004).

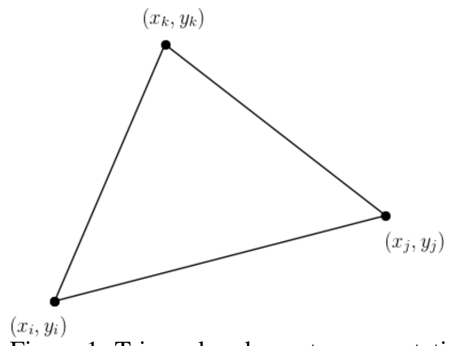


Figure 1: Triangular element representation

$$\begin{aligned} &\mathbf{N}(x,y) = [N_i \;\; N_j \;\; N_k] \\ &N_i = \frac{1}{2A}(a_i + b_i x + c_i y) \\ &N_j = \frac{1}{2A}(a_j + b_j x + c_j y) \\ &N_k = \frac{1}{2A}(a_k + b_k x + c_k y) \end{aligned} \tag{22}$$

Figure 1 shows a representation of the linear triangular element. With the (x, y) coordinates of each node (i, j, k), the coefficients in Eq. 22 are calculated as indicated in Eq. 23, along with the element area that is $A = a_i + a_j + a_k$.

$$\begin{aligned} &a_i = x_j y_k - x_k y_j; \;\; b_i = y_j - y_k; \;\; c_i = x_k - x_j \\ &a_j = x_k y_i - x_i y_k; \;\; b_j = y_k - y_i; \;\; c_j = x_i - x_k \\ &a_k = x_i y_j - x_j y_i; \;\; b_k = y_i - y_j; \;\; c_k = x_j - x_i \end{aligned} \tag{23}$$

Using the coefficients in Eq. 23 the element matrices implemented are Eqs. 24, 25. After computing these element matrices, the values are added to their respective global matrix that is used to solve the model equations.

$$\mathbf{k} = \frac{1}{4A}\begin{bmatrix} b_i^2 + c_i^2 & b_i b_j + c_i c_j & b_i b_k + c_i c_k \\ b_i b_j + c_i c_j & b_j^2 + c_j^2 & b_k b_j + c_k c_j \\ b_i b_k + c_i c_k & b_k b_j + c_k c_j & b_k^2 + c_k^2 \end{bmatrix} \qquad \mathbf{g_x} = \frac{1}{6}\begin{bmatrix} b_i & b_j & b_k \\ b_i & b_j & b_k \\ b_i & b_j & b_k \end{bmatrix} \tag{24}$$

$$\mathbf{m} = \frac{A}{12}\begin{bmatrix} 2 & 1 & 1 \\ 1 & 2 & 1 \\ 1 & 1 & 2 \end{bmatrix} \qquad \mathbf{g_y} = \frac{1}{6}\begin{bmatrix} c_i & c_j & c_k \\ c_i & c_j & c_k \\ c_i & c_j & c_k \end{bmatrix} \tag{25}$$

2.4 Algorithm

The method for solving the stream function-vorticity problem was based on the algorithm proposed on (Salih, 2013). At first an initial $\mathbf{v}$ field is initialized, respecting the boundary conditions for velocity, and the initial ω_z is calculated with Eq. 21. After that, the initial ψ function is obtained by solving Eq. 19 with a constant value Dirichlet condition on each wall boundary for ψ. This ψ boundary condition indicates the flow rate between the walls where it is defined.

The next is to solve Eq. 18 and for that we need its boundary conditions. As previous mentioned various finite differences schemes can be used for calculating it, however these schemes need a regular domain. The proposed way to calculate the vorticity values when using unstructured meshes is to calculate ω_z with Eq. 21 in each time step as the velocity field changes.

With the initial values for $\mathbf{v}$, ω_z and ψ and the boundary conditions calculated, the vorticity in Eq.18 is solved for the next time step, than with the new ω_z field, Eq. 19 is solved for the new ψ field. Afterwards the new velocity field in the fluid region is obtained by Eq. 20. With the velocity values computed we impose $\mathbf{v} = 0$ in the solid region elements and solve Eq. 17.

This process repeats until the change in the velocity components values reach a desired precision and after that only the heat transport needs to be solved as $\mathbf{v}$ is kept constant. And the algorithm stops when the same precision is reached for the temperature values.

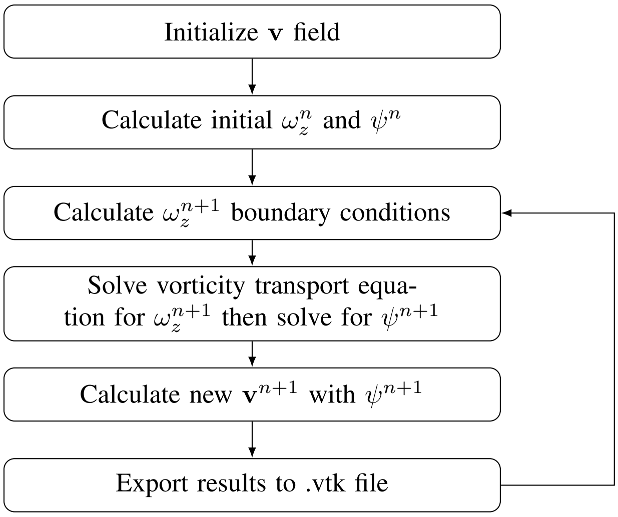


Figure 2: Summarized algorithm flow chart for solving the stream function-vorticity problem

3. RESULTS

3.1 Validation With Different Solid Regions

This is a validation for the heat equation solution in a $1x1$ square domain divided in two different solid regions on $y = 0.5$. The problem conditions and mesh (with 2944 nodes and 6086 elements) are presented in Fig. 3. For this case it is possible to obtain an analytical solution $T = T(y)$ as in Eq. 26. The thermal diffusivity of the solid 1 and 2 regions was set as $\alpha = 0.1$ and $\alpha = 1$ respectively, and on their interface nodes the harmonic mean $\alpha_i n$ of those coefficients was assumed (Eq. 27). The numeric results using $\Delta t = 0.05$ and 400 time steps is presented in Fig. 4 along with its comparison against the analytical solution of the problem.

$$T(y) = \begin{cases} \frac{2}{11}y, & 0 \leq y \leq 0.5 \\ \frac{20}{11}y - \frac{9}{11}, & 0.5 < y \leq 1 \end{cases} \tag{26}$$

$$\alpha_{in} = \frac{2\alpha_1\alpha_2}{\alpha_1 + \alpha_2} \tag{27}$$

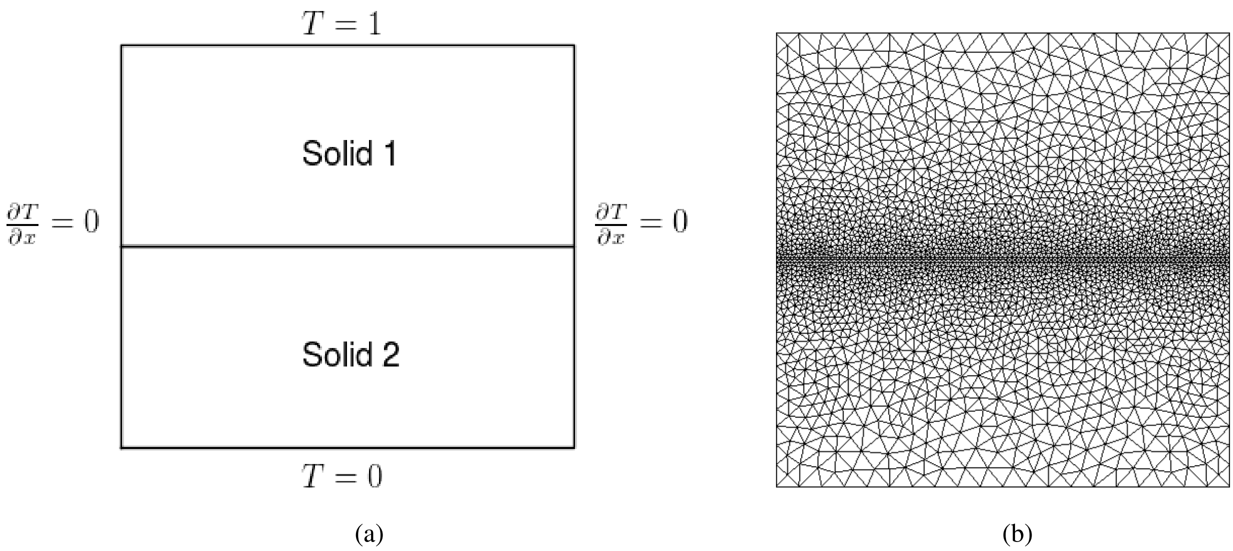
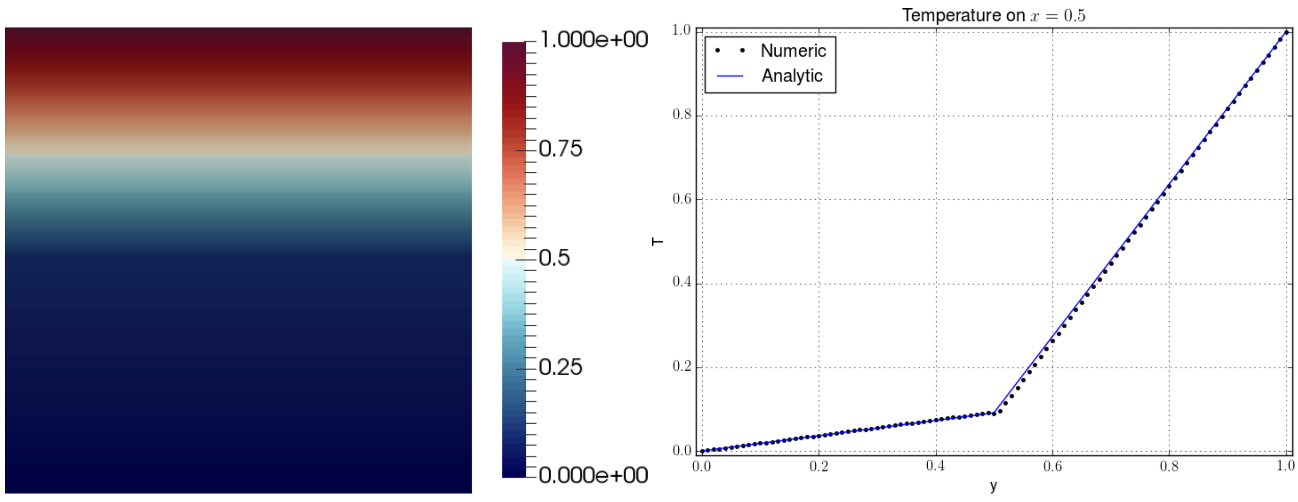


Figure 3: (a) Problem representation and boundary conditions; (b) Mesh with 2944 node and 6086 elements of different sizes closer to the interface generated with Gmsh



(a) (b)

Figure 4: (a) Numerical temperature distribution; (b) Comparison between analytical and numerical solution on a mesh with 2944 nodes and 6086 elements

A convergence analysis using the L^2 norm and five meshes with 436, 558, 812, 1435, 2944 nodes is shown in Fig. 5. From this graphic, it is possible to notice the numerical method is closer to a linear convergence rate. Being $\bar{u_i}$ the analytic solution and u_i the numeric solution at node i, the error is defined in Eq. 28 and the characteristic element length l was defined as the average l between th elements.

$$Error = \sqrt{\sum_{i}^{nodes} (\bar{u_i} - u_i)^2} \tag{28}$$

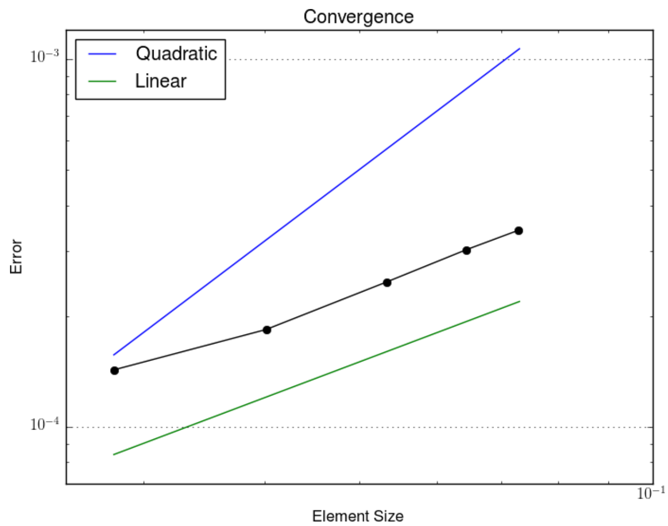


Figure 5: Convergence of the algorithm using L^2 norm, the characteristic length of the elements (l) were approximately 0.073, 0.064, 0.053, 0.040, 0.028

3.2 Heat Exchanger

The heat exchanger is an example of conjugate heat problem, where it is desired to know the temperature distribution in a fluid and a solid region. For this simulation a counter flow heat exchanger with the channels opening and the solid thickness having each a height of 0.1 and length equal to 1. The problem conditions and parameters are presented in Fig. 6 and Tab. 1.The top and bottom boundaries were considered isolated (no heat flux) as well as both sides of the solid region. The (ψ, ω) equations were solved only on the regions where the fluids would flow and the results were used, with setting $\mathbf{v} = 0$ on the solid region, to solve the heat equation in all the domain.

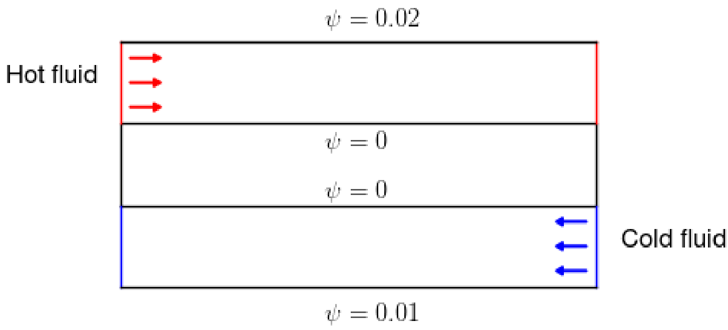


Figure 6: Heat Exchanger problem

Fluids	T (^oC)	α ($\frac{m^2}{s}$)	ν ($\frac{m^2}{s}$)	U ($\frac{m}{s}$)
Hot	800	0.0004	0.00015	0.2
Cold	300	0.0004	0.00015	0.1

Table 1: Fluid properties

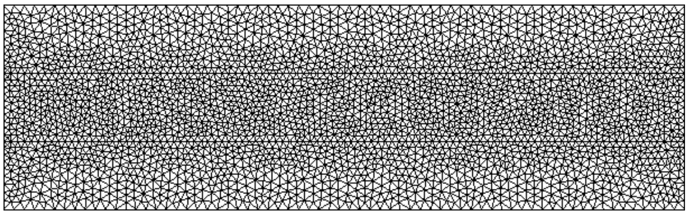
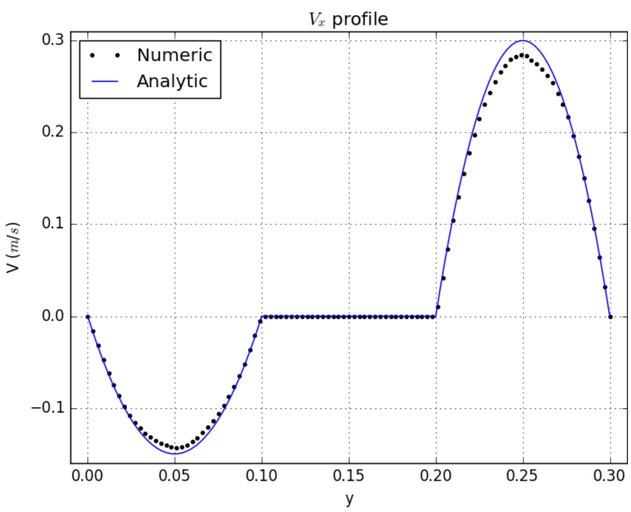
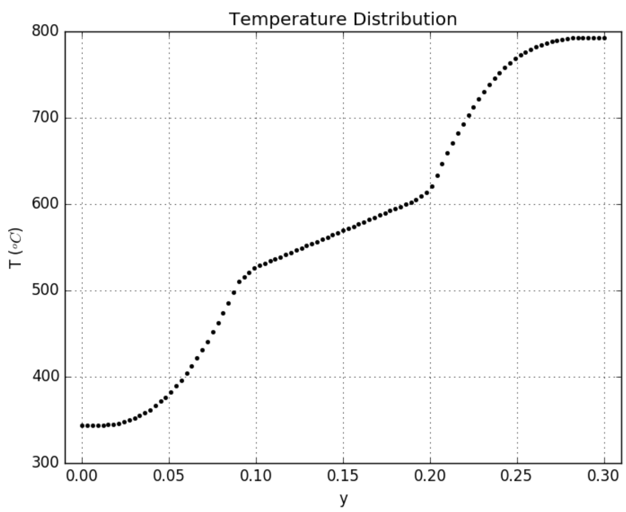


Figure 7: Mesh with 3154 nodes and 6486 elements generated with Gmsh (1944 nodes and 3884 elements on the fluid region)

The results presented in Fig. 9 and 8 were obtained with the mesh in Fig. 7 having 3154 nodes and 6486 elements, the solid region thermal diffusivity $\alpha = 0.002$ and solving for 300 time steps with $\Delta t = 0.5$. It is shown in Fig. 9 (a) a comparison between the numeric velocity calculated and an analytical solution for a Poiseuille flow with the prescribed conditions. Good agreement was found between the current numerical simulation and the solution found in Wansophark *et al.* (2005) and Chen and Han (2000)

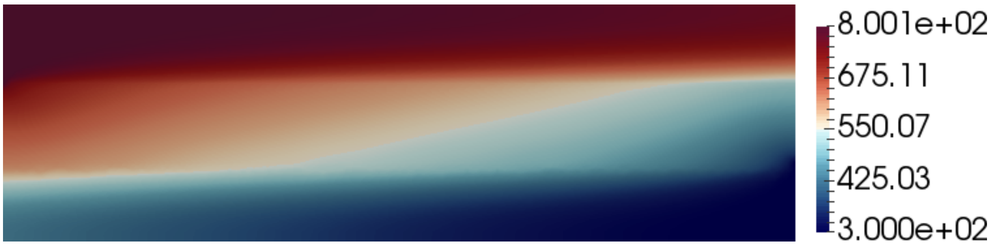


(a) Velocity on $x = 0.5$

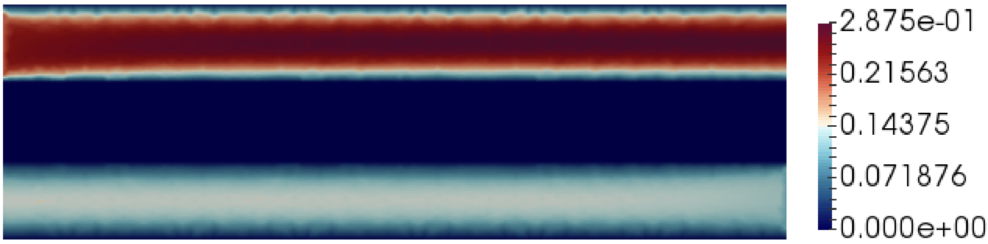


(b) Temperature on $x = 0.5$

Figure 8: Solutions obtained using the (ψ, ω) formulation along side the heat transport equation using the proposed FEM method for a heat exchanger simulation



(a) Complete Temperature Distribution

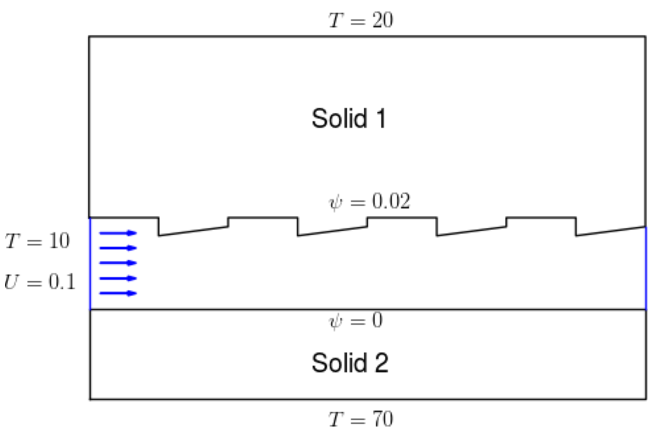


(b) Complete Velocity Magnitude Distribution

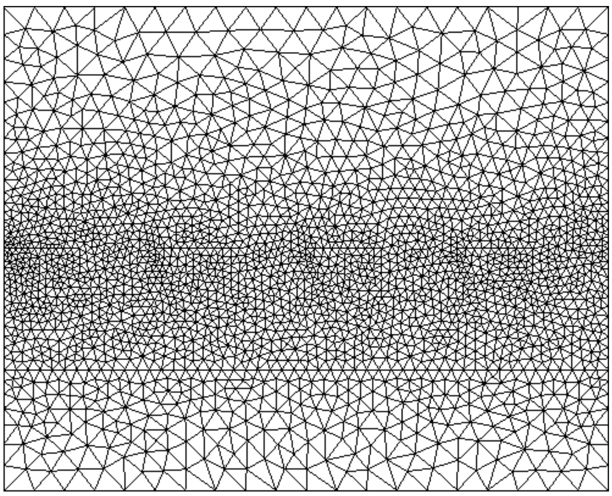
Figure 9: Complete solution of the heat exchanger simulation using the proposed model. The images were made with the open source software Paraview

3.3 Non Smooth Channel Between Different Solids

Here we have a simulation of the heat transfer between two solids that have a non smooth channel with a fluid of a lower temperature flowing between them. The problem geometry and boundary conditions are represented in Fig. 10 (a). The left and right walls of both solids were considered thermically isolated. The properties were arbitrarily defined and presented in Tab. 1.



(a)



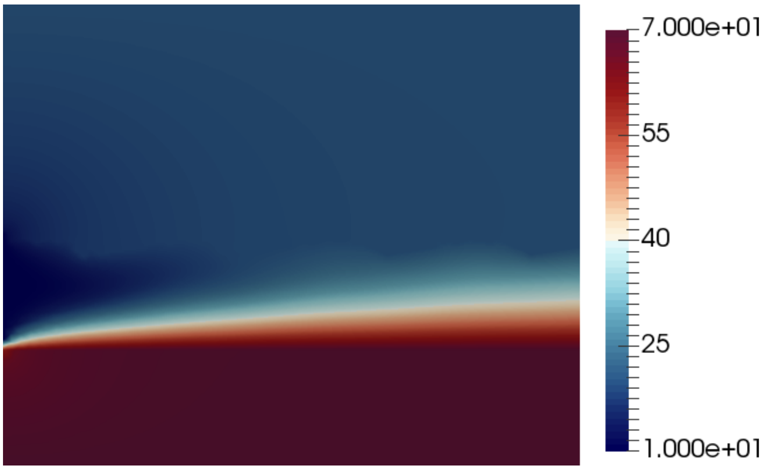
(b)

Figure 10: (a) Problem geometry and boundary conditions; (b) Mesh with different element sizes near the solid-fluid interfaces and in fluid region

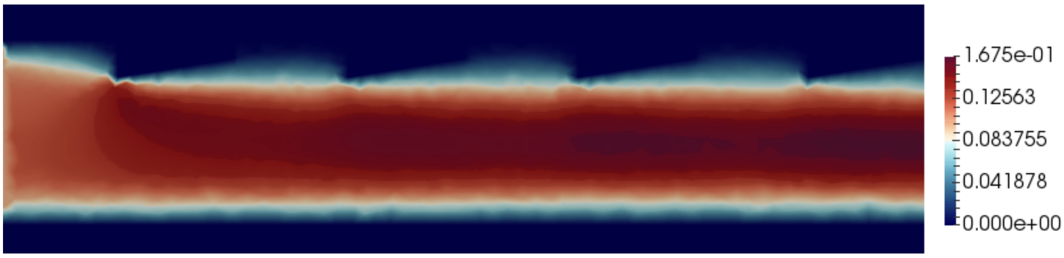
Properties	Solid 1	Solid 2	Fluid
α	1.2	1.0	0.001
ν	-	-	0.00015

Table 2: Defined thermal diffusivity values and fluid kinematic viscosity

The results presented in Fig. 11 were obtained with the mesh in Fig. 10(b) having 2308 nodes and 4756 elements in total (1077 nodes and 2152 elemnts in fluid region). As the heat exchanger case, $\Delta t = 0.5$ and now the problem was solved in 70 time steps.



(a) Complete Temperature Distribution



(b) Velocity magnitude distribution on fluid region

Figure 11: Complete solution of the simulation using the proposed model. The images were made with the open source software Paraview

4. CONCLUSION

In this paper we propose a method of numerically solving the conjugate heat problem using a finite elements algorithm. The Galerkin discretization method for solving stream function-vorticity formulation has good results only when the parameters of the problem are equivalent to low Reynolds numbers and a similar problem is observed when solving the heat equation, we are able to obtain good results only with low Peclet numbers, this happens due to instabilities that occur when the convective term of those equations starts to have more influence than the diffusive term.

The conjugate heat model proposed is still in development so that more details and a better range for Reynolds and Peclet numbers can be considered in the problem. A few possible ways to reduce the instabilities problem found is the use of different discretization schemes such as the Taylor-Galerkin method or a Semi-Lagrangian method. And with that being able to use the model in studying a wider range of aplications that require higher Reynolds and Peclet numbers with a better efficiency.

5. ACKNOWLEDGEMENTS

The authors thank the FAPERJ (Research Support Foundation of the State of Rio de Janeiro) for its financial support.

6. REFERENCES

Cesini, G., Paroncini, M., Cortella, G., and Manzan, M., 1998. “Natural convection from a horizontal cylinder in a rectangular cavity”. *International Journal of Heat and Mass Transfer*.

Chen, X. and Han, P., 2000. “A note on the solution of conjugate heat transfer problems using simple-like algorithms”. *International Journal of Heat and Fluid Flow*.

Geuzaine, C. and Remacle, J.F., 2009. “Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities”. *International Journal for Numerical Methods in Engineering*.

Hoffmann, K.A. and Chiang, S.T., 2000. *Computational Fluid Dynamics*, Vol. 1. Engineering Education Systems, 4th edition.

Lewis, R.W., Nithiarasu, P. and Seetharamu, K.N., 2004. *Fundamentals of the Finite Element Method for Heat and Fluid Flow*. John Wiley & Sons Ltd.

M.Vynnycky, S.Kimura, K.Kanev and I.Pop, 1998. "Forced convection heat transfer from a flat plate: the conjugate problem". *International Journal of Heat and Mass Transfer*.

Peeters, M.F., Habashit, W.G.. and Dueck, E.G.., 1987. "Finite element stream function-vorticity solutions of the incompressible navier-stokes equations". *International Journal for Numerical Methods in Fluids*.

Salih, A., 2013. "Streamfunction-vorticity formulation". Indian Institute of Space Science and Technology, Department of Aerospace Engineering.

Song, W. and Li, B., 2002. "Finite element solution of conjugate heat transfer problems with and without the use of gap elements". *International Journal of Numerical Methods for Heat & Fluid Flow*.

Wansophark, N., Malatip, A. and Dechaumphai, P., 2005. "Streamline upwind finite element method for conjugate heat transfer problems". *Acta Mech Sinica*.

7. RESPONSIBILITY NOTICE

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