NUMERICAL SOLUTION OF THE DRIFT FLUX MODEL OF TWO-PHASE FLOW USING TIME FINITE ELEMENT METHOD

Tirso Massari de Mello, tirsodemello@yahoo.com.br
Eduardo Gomes Dutra do Carmo, carmo@con.ufrj.br
Jian Su, sujian@con.ufrj.br
Nuclear Engineering Program, COPPE, Rio de Janeiro Federal University, CP 68509, Rio de Janeiro, 21945-970, Brazil

Abstract. *Thermal hydraulic analysis codes are of great importance for the nuclear industry, since they are essential for the safety analysis of the plants and simulation of physical phenomena with an acceptable realism. There is a variety of methods for the modelling of the two-phase flow, ranging from the homogeneous equilibrium model, where the liquid and the vapour form a homogeneous mixture, to the three-fluid model (liquid, vapour and liquid droplets). In this work, we adopt the thermal equilibrium drift flux model, which allows the velocities of the liquid and the vapour phases to be different, providing an algebraic relation between them. This model consists of a system of three partial differential equations, representing mass, momentum, and energy balances of the mixture. We propose here a numerical scheme that consists first of using the finite volume method in the integration in space, with a staggered mesh for the momentum balance in relation to the volume element used for the mass and energy balances. Next we used the finite element method to solve the resulting system of ordinary differential equations from the finite volume method alongside constitutive relations. The results obtained from this methodology are compared with the usual finite difference schemes so that the performance of the proposed method can be evaluated.*

Keywords: *two-phase flow, drift flux model, finite element method*

1. INTRODUCTION

Numerical simulation became, through time and the evolution of computers, an important tool for the design and maintenance of industrial plants. With the nuclear power plants it could not be different: the safety analysis of these plants requires more and more precise codes, to simulate the neutron flux inside the reactor core, to simulate flows in its different circuits, and so on. In the case of flow simulation, the evaluated parameters are given by a set of equations that represent mass, momentum, and energy balances in a control volume. However, the solution of these equations require a great computational effort. So, many simplified models have been created that, if they are not mathematically precise, help to simulate a physical event with a response time the closest to reality as possible.

Among the safety problems for light water reactors (LWR's), the most important are those referring to the loss of coolant accident (LOCA). The LOCA problem is important because it is considered a limitant factor in the reactor safety and therefore is classified as project basis accident. Hence, the main efforts in the research of LWR safety are directed in providing codes with the capactity to predict the consequences of LOCA's with the best estimate possible.

In this work we simulate two-phase flows as a base to create in the future a complete thermal hydraulic plant simulator with a graphic interface for the user. To realize this task we created two programs in Fortran 90: the first to generate the plant and the second to simulate the flow through it.

The mathematical model for the resolution of the equations that rule the flow uses the finite volume method in the spatial integration (Lapa, 1998), and the finite element method in the temporal integration adopting the discontinuous Galerkin variational formulation (DGM). The backward Euler scheme of finite differences can be reached from this formulation when the interpolation spaces are generated by the constant function 1 (Johnson, 1990).

The plant generator is an important tool for those who intend to simulate flows in open and closed circuits. It gives the user a great variety of plant configurations and even using the DOS as an interface it presents various resources of correction and reutilization. Its use will provide other researchers that they use their time only to research other numerical methods of resolution of the balance equations, without concerning in creating a code for the plant generation, what needs a considerable time.

In its turn, the simulator has the advantage of being implemented with the DGM, that degenerates into the backward Euler scheme when the number of points inside the time finite element is equal to 1. It is a known fact that the polynomial basis functions of lower order are more stable than the higher order ones (for instance, the Crank-Nicolson method). However, the DGM allows us to use other kinds of basis functions to recover and maintain the stability of the lower orders for polynomials, and to obtain a greater precision with longer time intervals. Therefore, only this perspective of investigation, justifies the construction of a simulator based in the DGM. Besides, in the investigations of these special functions we have the possibility to use the problem physics to construct them to carry on the precision and the stability.

2. THE DIFFERENT KINDS OF TWO-PHASE FLOWS AND THEIR MODELLING

In the one-fluid model the idea is to replace the two-phase fluid by an equivalent compressible one-phase fluid. The physical properties of the one-phase fluid (like specific mass and viscosity), as well as flow parameters (for instance, velocity and temperature) should then be defined by functions of the properties of each phase. If one of the phases is finelly dispersed, the momentum and energy transfers will be sufficiently fast for the average velocities and temperatures of both phases to be equal. If the temperature is the saturation one, the flow is described by the homogeneous equilibrium model (HEM).

The HEM is the simplest of the mixture models. It assumes that there is no relative velocity between the phases (that is, the flow is homogeneous) and that the vapour and the liquid are in thermodynamic equlibrium. For LWR applications, the HEM can be adequate to predict the pressure drop in a channel under stationary conditions of high pressure.

Other mixture models add complexities to the flow description. The example of model studied here is one of them, the thermal equilibrium drift flux model, which allows the vapour and liquid velocities to be different providing an algebraic relation for this difference.

In the two-fluid model, the liquid and vapour phases have three balance equations each. So this model is also called six-equation model. Some extensions to this model, leading to multifluid models, are possible but have not been applied so extensively like the two-fluid model.

Because of its nature the two-fluid model is the only model consistent with the balance laws for each phase and the interfaces. This model can be described using averages in time as well as in space. If the two-fluid model is written using time averages, it is possible to solve three-dimensional problems of transient two-phase flows, otherwise, if it is described using space averages, the two-fluid model can deal only with problems of transient flows with only one space variable.

There is a plenty of flow simulation programs, academic and industrial. From the nuclear industry we can cite:

- three-equation HEM: PAXITR e CONSEN (Sardain *et al.*, 2001);
- three-equation drift flux model: ALMOD e THYDE (Lapa, 1998);
- five-equation HEM: ECART (Sardain *et al.*, 2001);
- two-fluid model (six equations): CATHARE, MELCOR, RELAP e TRAC, the last two ones add two equations to the set, one for modelling non-condensable gases and other for the boron concentration (Sardain *et al.*, 2001);
- phase separation in a volume: INTRA e CONTAIN, only used to calculate the pressurization in a vacuum vessel (Sardain *et al.*, 2001).

The RELAP code is well known in the nuclear industry and is currently used by the Brazilian National Comission of Nuclear Energy (CNEN) to simulate the functioning of the plants of Angra dos Reis, Rio de Janeiro.

From the oil industry there are four major programs (Masella *et al.*, 1998):

- three-equation drift flux model: TACITE and TRAFLOW;
- two-fluid model: OLGA and PLAC.

The differences between these codes are found in the needs each industry has. Nuclear industry has preference for fast transients, like in a LOCA, while oil and gas industry prefers slow transients, like the transport and delivery of slugs in recepting equipment, a phenomenum known as severe slugging.

3. THE THREE-EQUATION DRIFT FLUX MODEL

In the drift flux model we consider the gas and the liquid as being a mixture whose phases are in thermodynamic equilibrium but allows the gas and the liquid velocities to be different by providing an algebraic relation for the velocity difference (Delhaye *et al.*, 1981). Therefore, the mass, momentum, and energy conservation equations are sufficient to describe the flow.

We have chosen this model because it is mathematically simple without losing so much precision in the informations obtained from it. And we can raise its complexity and precision by adding transport equations to the system.

Here we present the equations adopted in this model (Lapa, 1998), whose simplifications and correlations can be found in Todreas and Kazimi (1990). We should observe that by thermodynamic equilibrium we can neglect the volumetric contraction and expansion, what is valid for subsonic compressible processes.

Mass balance equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \widetilde{u}) = 0 \tag{1}$$

Energy balance equation

$$\frac{\partial}{\partial t}\left(\rho H - P\right) + \nabla \cdot \left[\rho H \widetilde{u} + \alpha(1-\alpha)\left(\frac{\rho_v \rho_l}{\rho}\right)(H_v - H_l)\widetilde{u}_R\right] = -\nabla \cdot \widetilde{q'''} \tag{2}$$

Momentum balance equation

$$\frac{\partial}{\partial t}(\rho \widetilde{u}) + \nabla\left[\rho \widetilde{u} \otimes \widetilde{u} + \alpha(1-\alpha)\left(\frac{\rho_v \rho_l}{\rho}\right)\widetilde{u}_R \otimes \widetilde{u}_R\right] = -\nabla P + \nabla \cdot \widetilde{\tau} + \rho \widetilde{g} \tag{3}$$

where $\nabla\cdot$ denotes divergent and ∇ gradient.

The following relations are used in the equations above:

$$H_l = H_l(P,T) \tag{4}$$

$$H_v = H_v(P,T) \tag{5}$$

$$\widetilde{u} = \frac{\alpha\rho_v \widetilde{u}_v + (1-\alpha)\rho_l \widetilde{u}_l}{\rho} \tag{6}$$

$$\widetilde{u}_R = \widetilde{u}_v - \widetilde{u}_l \tag{7}$$

$$\rho = \alpha\rho_v + (1-\alpha)\rho_l \tag{8}$$

$$\rho H = \alpha\rho_v H_v + (1-\alpha)\rho_l H_l \tag{9}$$

Here, the tilde indicates a vector.

4. THE MATHEMATICAL MODELLING

4.1 The spatial integration of the balance equations

We have integrated the equations using the divergence theorem (Reddy, 1993). We divided our plant in singular components because the spatial integration considers the geometry of those components (Maliska, 1995). The one-dimensional equations obtained after this integration are valid only locally, in the neighbourhood of the singularity points, that indicate the presence of pumps or valves. Hence, the remaining equations correspond to the equations of the model spatially integrated with the terms that count the effects of the singularity points.

The equations presented below are integrated in a control volume element for the components only with one inlet and one outlet and discretized in space. The mass and energy equations use the concept of centred mesh (figure 1) and the momentum balance equation uses the concept of staggered mesh (figure 2) to eliminate the instability existant on the cell interfaces.

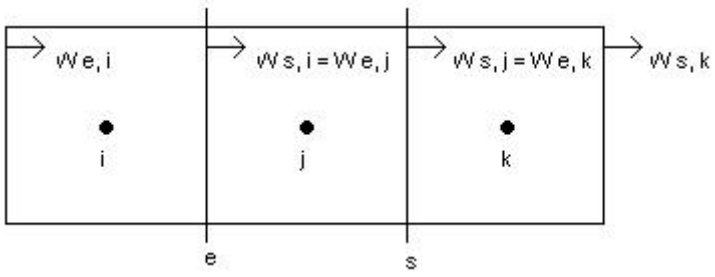

Figure 1. Component with one inlet and one outlet (centred mesh)

Mass balance equation

$$V_j \frac{\partial \rho}{\partial t} = W_{s,i} - W_{s,j} - G_{s,j} \tag{10}$$

where $G_{s,j} = G_{s,j}(H_j, P_j)$ is the term that counts leaks in the component j.

Energy balance equation

$$Q_j = V_j \left[\frac{\partial}{\partial t}(\rho_j H_j) - \frac{\partial P_j}{\partial t} \right] - S_{e,j} + S_{s,j} - H_j G_{s,j} \tag{11}$$

where:

$$S_{e,j} = \begin{cases} W_{e,j} H_i + HW_{e,i} + THB_{e,i} & \text{, para } W_{e,j} \geq 0 \\ W_{e,j} H_j + HW_{e,j} & \text{, para } W_{e,j} < 0 \end{cases} \tag{12}$$

and

$$S_{s,j} = \begin{cases} W_{s,j} H_j + HW_{s,j} & \text{, para } W_{s,j} \geq 0 \\ W_{s,j} H_k + HW_{s,k} + THB_{s,k} & \text{, para } W_{s,j} < 0 \end{cases} \tag{13}$$

being:

$$HW = A u_R \alpha (1-\alpha) \left(\frac{\rho_v \rho_l}{\rho} \right) (H_v - H_l) \tag{14}$$

the enthalpy due to the difference between the phases.

This division of the equation in two parts is due to the fact that we adopted one preferential flow direction. Therefore, if there is a flow reversal, a second consideration about the energy equation has to be done.

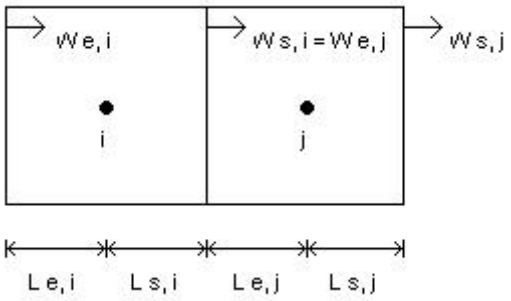

Figure 2. Component with one inlet and one outlet (staggered mesh)

Momentum balance equation on the outlet

$$\begin{aligned} &(L_{s,j} + L_{e,k}) \rho_j \frac{\partial u_{s,j}}{\partial t} - \frac{1}{2} \left[\rho_j (u_j^v)^2 + \alpha_j (1-\alpha_j) \left(\frac{\rho_{v,j} \rho_{l,j}}{\rho_j} \right) (u_{R,j})^2 \right] \\ &+ \frac{1}{2} \left[\rho_i (u_k^v)^2 + \alpha_k (1-\alpha_k) \left(\frac{\rho_{v,k} \rho_{l,k}}{\rho_k} \right) (u_{R,k})^2 \right] + \frac{1}{2} \left[(u_{s,j}) \left| u_{s,j} \right| (\rho_k - \rho_j) \right] \\ &- \frac{1}{2} (u_{R,j}) \left| u_{R,j} \right| \left[\alpha_j (1-\alpha_j) \left(\frac{\rho_{v,j} \rho_{l,j}}{\rho_j} \right) + \alpha_k (1-\alpha_k) \left(\frac{\rho_{v,k} \rho_{l,k}}{\rho_k} \right) \right] \\ &= P_j - P_k + \rho_j g (Y_j - Y_k) + FRW_{s,j} + FLOC_{s,j} + FV_{s,j} + FB_{s,j} \end{aligned} \tag{15}$$

where:

$$FRW_{s,j} + FLOC_{s,j} + FV_{s,j} = -C_{s,j} \left(\frac{W_{s,j} \left| W_{s,j} \right|}{\rho_{s,j}} \right) \tag{16}$$

and

$$FB_{s,j} = \rho_j g HEAD_{s,j} \tag{17}$$

sendo:

$$C_{s,j} = \left[\frac{CFW_{s,j}}{8(A_{s,j})^2} + \frac{CPL_{s,j}}{8(A_{s,j})^2} + \frac{CVAL_{s,j}}{2(AVALV_{s,j})^2} \right] \tag{18}$$

For the inlet the equation is quite similar. The reason for having an equation for the inlet and another for the outlet is that the mass flow rate is calculated on a section (inlet or outlet) rather than in the volume, like the enthalpy and the pressure.

For elements with more than one inlet and/or more than one outlet the procedure is similar, but since we only present tests for one-inlet, one-outlet elements, they are not shown here.

4.2 The finite element formulation in time

Being $A(\psi_1, ..., \psi_{nd})$ and $B(\psi_1, ..., \psi_{nd})$ matrices of order $nd \times nd$ and $f(\psi_1, ..., \psi_{nd})$ a vector of $\Re^{nd}$ for each $t \in [t_j, t_{j+1}]$ fixed.

The initial condition problem in the interval $[t_j, t_{j+1}]$ can be proposed as follows:

$$\sum_{m=1}^{nd} \left[\frac{d}{dt} A_{l,m}(\psi_1, ..., \psi_{nd}) + B_{l,m}(\psi_1, ..., \psi_{nd}) \right] = f_l(\psi_1, ..., \psi_{nd}) \tag{19}$$

$$F_l(\psi_1^j(t_j), ..., \psi_{nd}^j(t_j)) = F_l(\psi_1^{j-1}(t_j), ..., \psi_{nd}^{j-1}(t_j)) \tag{20}$$

$$l = 1, ..., nd \tag{21}$$

where ψ_l^{j-1} is the solution in the interval $[t_{j-1}, t_j]$, $A_{l,m}(\cdot)$ is the component of order $l \times m$ of $A(\cdot)$, $B_{l,m}(\cdot)$ is the component of order $l \times m$ of $B(\cdot)$, $f_l(\cdot)$ is the component of order l of $f(\cdot)$ e $\psi_m^j \in H^1(t_j, t_{j+1}), \forall m$ and $\forall j$.

The problem defined by (19-21) will be equivalent to the thermal hydraulic problem if, and only if, for a $1 \leq l \leq nd$ fixed, there is a cell so that the equation of order l is associated to one of the following equations: mass balance, energy balance, or momentum balance of an outlet of this cell.

Besides, with the kind of equation determined, we can explicit the components $A_{l,m}(\cdot)$ and $B_{l,m}(\cdot)$ of the matrices $A(\cdot)$ e $B(\cdot)$ as well as the independent term $f_l(\cdot)$, by direct correspondence between this balance equation and the equation of order l, noticing that $\psi_m^j(t)$ can only be a mass flow rate for any section of the plant or an enthalpy for a cell, or a pressure for a cell. We should also notice that the $A_{l,m}(\cdot)$ and $B_{l,m}(\cdot)$ that do not have correspondents in the equation of order l will be considered as zero.

4.2.1 The Galerkin formulation of discontinous finite elements

The formulation of the discontinuous Galerkin method (Johnson, 1990) for the problem defined by (19-21) consists of finding $(\psi_1^{h,j}, ..., \psi_l^{h,j}, ..., \psi_{nd}^{h,j}) \in P^{nd,\widetilde{k}}_{[t_j,t_{j+1}]}$, satisfacting the following system of variational equations:

$$\begin{aligned} & F_l(\psi_1^{h,j}(t_j), ..., \psi_{nd}^{h,j}(t_j)) - F_l(\psi_1^{h,j}(t_{j-1}), ..., \psi_{nd}^{h,j}(t_{j-1})) \\ & + \int_{t_j}^{t_{j+1}} \left[\sum_{m=1}^{nd} \frac{d}{dt} A_{l,m}(\psi_1, ..., \psi_{nd}) + B_{l,m}(\psi_1, ..., \psi_{nd}) \right] \eta_l dt \\ & = \int_{t_j}^{t_{j+1}} f_l(\psi_1, ..., \psi_{nd}) \eta_l dt \end{aligned} \tag{22}$$

$$\forall \eta_l \in P^{k(l)}_{[t_j,t_{j+1}]} \tag{23}$$

$$l = 1, ..., nd \tag{24}$$

4.2.2 The choice of the discontinuous Galerkin method

Noticing that if $NH = NP = NW = 1$ and $k_H = k_P = k_W = 0$, implies that $H^{h,j+1}$, $P^{h,j+1}$ and $W^{h,j+1}$ are constant in the interval $[t_j, t_{j+1}]$ and, therefore, the bases are: $\eta_1^H = \eta_1^P = \eta_1^W = 1$, hence, $\frac{\partial \eta_1^H}{\partial \xi} = \frac{\partial \eta_1^P}{\partial \xi} = \frac{\partial \eta_1^W}{\partial \xi} = 0$, $\frac{\partial \eta_1^H}{\partial \xi} = \frac{\partial \eta_1^P}{\partial \xi} = \frac{\partial \eta_1^W}{\partial \xi} = 0$, we can deduct easily from the equations (19-21) that the discontinous Galerkin method degenerates into the backward Euler scheme.

Besides, the discontinuous Galerkin formulation allows by clear form that interpolations of different orders can be used for the thermal hydraulic variables. This suggests the possibility of using it in transients with different sensibility for each variable, allowing researches to establish what should be the adequate order to interpolate each variable and using longers time intervals.

Also should be noticed the natural stability of the discontinuous Galerkin method for each order of interpolation of the variables (Johnson, 1990). This is also a good indication for its use; when it is not the best option we can always go to the backward Euler scheme.

5. TESTS AND RESULTS

Here we present the tests ran for two simple plants (horizontal and vertical). First we made a stabilty analysis of the discontinuous Galerkin method for different time intervals. Next there are the results for the increasing in the heat generation in the circuit.

5.1 Horizontal circuit

In the first section of tests, we used a horizontal circuit of 3.66 m of length and 12 mm of diameter divided in fifteen equal cells. The adopted boundary conditions were: enthalpy of 2000 kJ/kg at the inlet (cell no. 1), pressure of 15 MPa at the inlet and of 14.998 MPa at the outlet (cell no. 15). The initial conditions are, pressure of 14.999 MPa at the interior cells (cells 2 to 14) and mass flow rate of 5.0×10^{-4} kg/s. The cell chosen for the tests was the cell no. 8, for being located at the middle of the circuit. Such conditions were used due to the simulator's limitations – it works only at the water saturation region.

The first round of tests was carried out to evaluate the stability and the robustness of the discontinuous Galerkin method with finite elements of orders 0 (backward Euler scheme) and 1. In this case there is no heat generation at the circuit, so there are only results for the steady-state problem.

We have three different time intervals: 0.1 s, 0.01 s, and 0.001 s. With the initial instant being 0 s and the final being 10 s. So, for:

- time interval = $0.1s \rightarrow 100$ time steps;
- time interval = $0.01s \rightarrow 1,000$ time steps;
- time interval = $0.001s \rightarrow 10,000$ time steps.

Table 1. Finite element of order 0 – steady state (after 10 s).

Interval (s)	Enthalpy (J/kg)	Pressure (Pa)	Mass flow rate (kg/s)
0.1	2000002.67227549	14990004.0479735	0.976734168614151
0.01	2000003.06005573	14990005.1367047	0.976762751448054
0.001	2000003.03227803	14990005.1364533	0.976763222694536

Table 2. Finite element of order 1 – steady state (after 10 s).

Interval (s)	Enthalpy (J/kg)	Pressure (Pa)	Mass flow rate (kg/s)
0.1	2000003.36857604	14990005.1393078	0.976762970189406
0.01	2000003.06003282	14990005.1367614	0.976768474629325
0.001	2000003.03225950	14990005.1365102	0.976768971346970

We see here that even with an initial approach for the mass flow rate of 5.0×10^{-4} kg/s the flow becomes stable, and that even the time interval nor the order of the polynomials change significantly the results. It shows that the method is robust when there is no disturbings in the flow.

In the second round of tests a stability analysis with a heat generation of 1 kW during the steady state and later it was added 10% of this power in the transient regime.

Table 3. Finite element of order 0 – transient state (after 10 s).

Interval (s)	Enthalpy (J/kg)	Pressure (Pa)	Mass flow rate (kg/s)
0.1	1992505.27873945	14989917.2439270	1.14878002865093
0.01	1993087.20170807	14989925.4888350	1.13188301117167
0.001	1993140.64577445	14989926.3043637	1.13037030885089

Table 4. Finite element of order 1 – transient state (after 10 s).

Interval (s)	Enthalpy (J/kg)	Pressure (Pa)	Mass flow rate (kg/s)
0.1	1992504.01940331	14989916.3138907	1.14881571015677
0.01	1993087.20199075	14989925.4888510	1.13188305141233
0.001	1993140.64577445	14989926.3043637	1.13037030885089

Here we see that there is no difference between the results when we look at both orders, but there is a difference a little greater than 1% between the results obtained between the 0.1 s and 0.01 s intervals. Even so, the model still shows the same robustness of the first test.

5.2 Vertical circuit

In the tests with the vertical circuit we adopted the same boundary and initial conditions. We analysed the method stability and then the effect of the heat transient in the mass flow rate. Like in the other tests we present the results for the cell no. 8. In this circuit the cell no. 1 (the inlet) is in the lower extreme and the cell no. 15 (the outlet) is in the upper extreme.

In the first round of tests a stability analysis with a heat generation of 1 kW during the steady state and later it was added 10% of this power in the transient regime.

Table 5. Finite element of order 0 – steady state (after 10 s).

Interval (s)	Enthalpy (J/kg)	Pressure (Pa)	Mass flow rate (kg/s)
0.1	1991739.33297925	14989904.4894712	0.942668565030488
0.01	1992262.31942053	14989964.9230342	0.920604664288268
0.001	1992321.14168076	14989965.3750365	0.919240630830601

Table 6. Finite element of order 0 – transient state (after 10 s).

Interval (s)	Enthalpy (J/kg)	Pressure (Pa)	Mass flow rate (kg/s)
0.1	1990951.90364536	14989951.5405235	0.952680307657673
0.01	1991625.20218273	14989959.9821329	0.935533871007745
0.001	1991687.81815348	14989960.4864341	0.934036412662301

Table 7. Finite element of order 1 – steady state (after 10 s).

Interval (s)	Enthalpy (J/kg)	Pressure (Pa)	Mass flow rate (kg/s)
0.1	1991621.01740294	14989959.2477736	0.935726939788375
0.01	1992262.68953142	14989964.9358524	0.920642748383566
0.001	1992321.67680745	14989965.3722150	0.919303392829483

Table 8. Finite element of order 1 – transient state (after 10 s).

Interval (s)	Enthalpy (J/kg)	Pressure (Pa)	Mass flow rate (kg/s)
0.1	1990944.67584793	14989954.1891226	0.952332536761330
0.01	1991625.20604844	14989959.9822699	0.935534245089586
0.001	1991687.82257871	14989960.4864101	0.934036899956186

As well as in the horizontal circuit, we see here that there is no difference between the results when we look at both orders, but there is a difference a little greater than 1% between the results obtained between the 0.1 s and 0.01 s intervals. Even so, the model provides stable solutions, prooving its robustness.

6. CONCLUSIONS

In this work, we presented a program that creates and simulates thermal hydraulic plants based upon the finite volumes in space and upon the Galekin method of discontinuous finite elements in time, adopting the three-equation drift flux flow model.

Despite the simulator's limitations, it was possible to run a good plenty of tests for two different circuits. The results show that for the specified pressure drop between the inlet and the outlet, independently if the circuit is horizontal or vertical, the mass flow rate reached stable values even with such a small initial value. This result shows the robustness of the Galerkin formulation of discontinuous time finite elements.

The results obtained do not present differences between the tests ran with the Galerkin formulation of order 0 and order 1. That means it does not matter the degree of the polynomial adopted to make the interpolation, that is, when using the order 0 we get the same result of the order 1 with a time saving in the execution of the program without significant precision losses.

We can conclude from this is that the Lagrange polynomials may not be the best shape function to be used to simulate that heat transient. This is a great advantage of the discontinuous Galerkin method: one can research what is the best shape function to simulate a determined kind of transient. And we can change the order of the shape function for each thermal hydraulic variable and then this method can capture a transient that is more sensitive for a determined variable and less sensitive to the other ones.

All of this show the great future the discontinuos Galerkin method has. When the order of the shape functions is zero it is what we call the backward Euler scheme of finite differences, the method used by the famous simulators RELAP and TRAC, well experimented by the scientific community. Besides, the discontinuous Galerkin method can make a thinner sinthony inside the time interval, through the increasing of the order of the shape functions, what makes it to be target of many upcoming studies.

7. REFERENCES

Delhaye, J. M., Giot, M. and Riethmuller, M. L., 1981, "Thermodynamics of two-phase systems for industrial design and nuclear engineering", Hemisphere Publishing Corporation, New York, USA.

Johnson, C., 1990, "Numerical solution of partial differential equations by the finite element method", Cambridge University Press, Cambridge, United Kingdom.

Lapa, N. S., 1998, "Um novo algoritmo computacional para análise termoidráulica de plantas industriais", M.Sc. Dissertation, Rio de Janeiro Federal University, Rio de Janeiro, Brazil.

Maliska, C. R., 1995, "Transferência de calor e mecânica dos fluidos computacional", Editora LTC, Rio de Janeiro, Brazil.

Masella, J. M., Tran, Q. H., Ferre, D. and Pauchon, C., 1998, "Transient simulation of two-phase flows in pipes", International Journal of Multiphase Flow, vol. 24, pp. 739-755.

Reddy, J. N., 1993, "An introduction to the finite element method", McGraw-Hill, New York, USA.

Sardain, P., Girard, C., Andersson, J. Porfiri, M. T., Kurihara, R., Masson, X., Mignot, G., Pinna, T. and Topilski, L., 2001, "Modelling of two-phase flow under accidental conditions fusion codes benchmark", Fusion Engineering and Design, vol. 54, pp. 555-561.

Todreas, N. E. and Kazimi, M. S., 1990, "Nuclear systems I - thermal hydraulics fundamentals", Hemisphere Publishing Corporation, New York, USA.

8. APPENDIX - LISTS OF SYMBOLS

Table 9. Symbols.

g	gravity acceleration	H	enthalpy
q'''	heat generation per unit volume	$HEAD$	pump head
t	time	L	length
u	velocity	P	pressure
A	area	Q	heat generation
$AVALV$	area of flow trhough valve	THB	enthalpy flux due to pump
CFW	loss coefficient by viscous friction	V	volume
CPL	loss coefficient by singularity	W	mass flow rate
$CVAL$	loss coefficient by valve	Y	height of the centre of the cell
FB	head gain through pump	α	void fraction
$FLOC$	head loss due to local losses (singularities)	ρ	specific mass
FRW	head loss due to viscous friction	τ	shear stress
FV	head loss due to valve		

Table 10. Indices and the exponent v.

e	inlet section	s	outlet section
i	cell i (cell before j)	v	vapour
j	cell j (cell being evaluated)	R	relative (vapour - liquid)
k	cell k (cell after j)	exponent v	volume
l	liquid		

9. Responsibility notice

The author(s) is (are) the only responsible for the printed material included in this paper.