Numerical simulation of large hydraulic systems using a multi-domain approach

Santiago F. Corzo[a], Johan A. Sarache Piña[a], Dario Godino[a], Antonella L. Costa[b], Claubia Pereira[b], Damian E. Ramajo[a]

[a] Centro de Investigacion de Metodos Computacionales
CIMEC-UNL/CONICET, Santa Fé, Argentina

[b]Departamento de Engenharia Nuclear
Universidade Federal de Minas Gerais, Belo Horizonte, Brazil

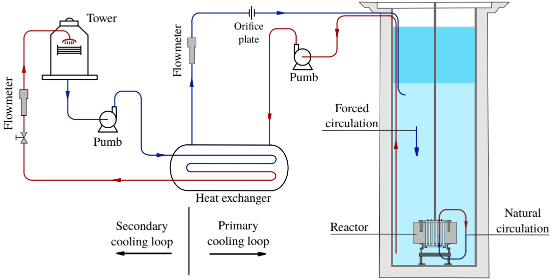
23/10/2019

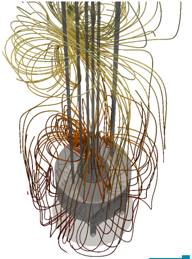
Nuclear New Horizons: Fueling or Future
October 21-25, 2019 - Santos, SP, Brazil

Introduction

Motivation

Sensible engineering fields, like the nuclear safety assessment, involve the analysis of large installations composed of very different scale components, where single and two-phase flows take place. Thus, the verification of plant accidents has forced to the community to use System Codes like RELAP and ATHLET, which are based on the domain reduction and the use of empirical correlations.

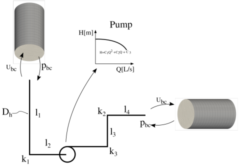




Governing Equations

1. Momentum equation:

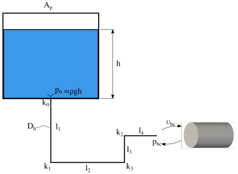
$$\frac{\partial(\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla p + \nabla \cdot \tau + \rho \mathbf{g}$$



2. Volume and temporal discretization:

$$U = U_0 + \Delta t \left[(p_{inl} - p_{out})/(\rho L) + g(h_{inl} - h_{out})/L \right]$$

$$+\Delta t \left[\left(\frac{1}{2} \lambda \frac{L}{D_h} U^2 + \frac{1}{2} K U^2 \right) /L + \Delta P_{ext} \right]$$

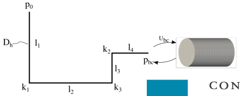


3. Algebraic solution:

$$\mathbf{U^n} = U^{n-1} + \Delta t \left[(\mathbf{p^n_{bc,1}} - \mathbf{p^n_{bc,2}})/(\rho L) + g(h_{inl} - h_{out})/L \right]$$

$$+\Delta t f(\lambda, (\mathbf{U^n})^2)/L + (\Delta t \mathbf{H} g)/L$$

where $H(Q) = C_1 Q^2 + C_2 Q + C_3$

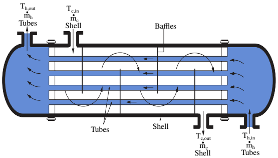


UNL CONICET CIMEC

Government Equations:

In order to solve the heat transfer 0D model, the Effectiveness-NTU Method was implemented.

Equations
$\epsilon = \frac{\dot{Q}}{\dot{Q}_{max}}$
$\dot{Q} = C_c\left(T_{c,out} - T_{c,in}\right) = C_h\left(T_{h,in} - T_{h,out}\right)$
$\Delta T_{max} = T_{h,in} - T_{c,in}$
$\dot{Q}_{max} = C_{min}\left(T_{h,in} - T_{c,in}\right)$
$c = \frac{Cmin}{C_{max}}$
$NTU = \frac{UA_s}{C_{min}}$
$\epsilon = 2\left\{1 + c + \sqrt{1+c^2}\frac{1+exp\left[-NTU\sqrt{1+c^2}\right]}{1-exp\left[-NTU\sqrt{1+c^2}\right]}\right\}^{-1}$
$T_{h,out} = T_{h,in} - \frac{\dot{Q}}{\dot{m}_h C_{ph}}$



UNL CONICET
C I M E C

Governing Equations: 0D Thermal model

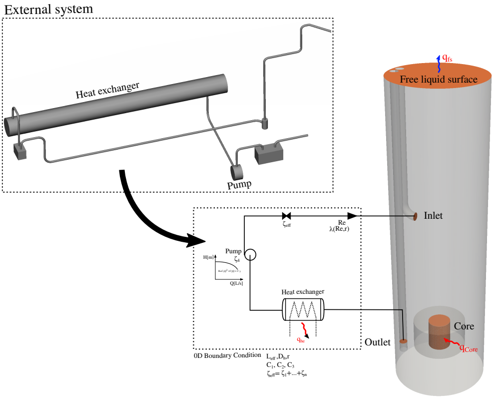
For the BC specification in OpenFOAM, the user data file input contains the following parameters for the velocity and temperature fields at the INLET/OUTLET patches.

```
INL/OUT
   {
    type            couplingPipeFixedValue;
    master          yes;        //Is the master patch?
    neiPatchName    "OUT";      //Nei. patch []
    hStart          0.0;        //Master height [m]
    hEnd            2.978;      //Neighbour height [m]
    Lline           37.0;       //Pipe length [m]
    rug             1e-5;       //Pipe roughness [m]
    kEff            2001.2;     //Eff. loss coeff. []
    Dh              0.1698;     //Hydr. diameter [m]
    pmpC1           -0.01;      //Pump coefficient []
    pmpC2           0.0;        //Pump coefficient []
    pmpC3           13.0;       //Pump coefficient []
   }
```

```
INL/OUT
   {
    type            heatPipePFixedValue;
    neiPatchName    OUT;        //Nei. patch []
    A               47;         //Heat transfer area [m2]
    mS              11.1;       //Secondary mass flow rate [kg/s]
    TiS             299.85;     //Secondary inlet temperature [K]
    Ugl             836.8;      //Global heat transfer coef. [W/m2K]
    rho             1000;       //Mean density [kg/m3]
    Cp              4174.4;     //Mean heat capacity [J/kg/K]
   }
```

Numerical Model

- Computational domain: $\approx 800,000$ cells.
- Domain simplifications: Only the inlet/outlet pipes and the reflector were considered.
- Solver: Compressible (buoyantPimpleFoam).
- Turbulence model: Realizable $k - \epsilon$.
- Core pressure drop: Porous media was chosen. A Forchheimer coefficient $F = 60$ was adopted.
- Core power: It was imposed as a volumetric power source.



CONICET
C I M E C

Results - Steady-state condition

The steady state conditions were assessed for two power conditions:

- Case 1: 100kW
- Case 2: 265kW (*Mesquita et al.*,2011).

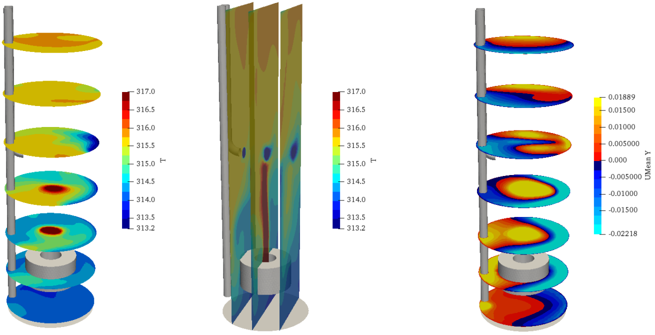
The simulation was carried out for 17h, using parallel computing.

For each hour of problem simulation, 3,42h of CPU time was required.

Hydraulic system		
Parameter	Value	Unit
Total lenght	37.0	[m]
ζ_{tot}	2001.2	[-]
Pump coef.	-0.01, 0.0, 13.0	$[ms^2/L^2]$ $[ms/L]$ $[m]$
Roughness	1×10^{-6}	$[\mu m]$
Heat exchanger		
Parameter	Value	Unit
Area	47.0	$[m^2]$
Ext. flow rate	11.1	$[kg/s]$
Ext. flow inlet ttemp.	299.8	$[K]$
Ugl.	836.8	[W/m2K]

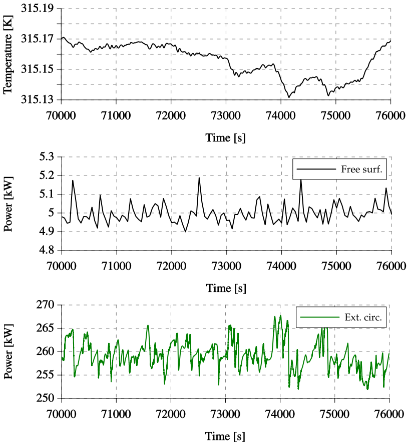
UNL CONICET CIMEC

Steady-state condition



a) b) c)

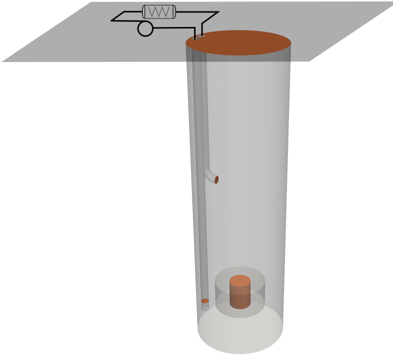
Results - Steady-state condition



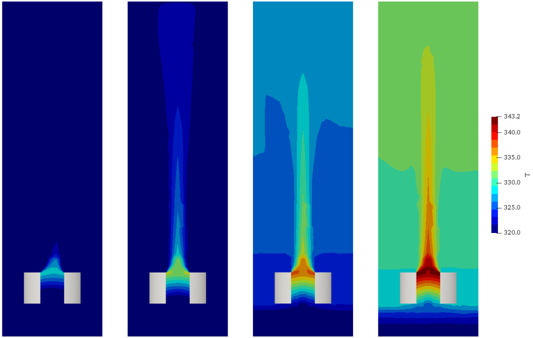
External circuit				
Parameter	Case 1	Case 2		Unit
-	Result	Result	Experiment	Unit
Power	100	265	265	[kW]
$\dot{m}$	7.95	7.94	7.8	[Kg/s]
T_{bulk}	305.85	315.15	-	[K]
T_{inl}	302.54	306.46	-	[K]
T_{out}	305.73	314.26	-	[K]
Core				
$\dot{m}$	3.4	4.7	4.42	[Kg/s]
T_{core}	309.46	321.7	-	[K]
T_{inl}	305.72	314.5	-	[K]
T_{out}	312.72	327.9	-	[K]
ΔT	7.0	13.4	14.3	[K]

Loss of forced convection event

- Event: A primary pump shut-down condition was simulated.
- Consequences: The loss of forced convection leads to a progressive coolant heating because the free-surface heat transfer is not enough.
- Core power: A constant core power of 265kW was assumed.
- The simulation was carried out for 5000s.

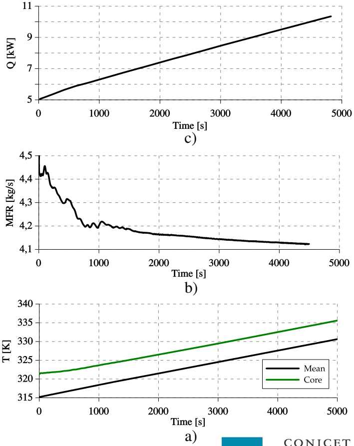


Loss of forced convection event



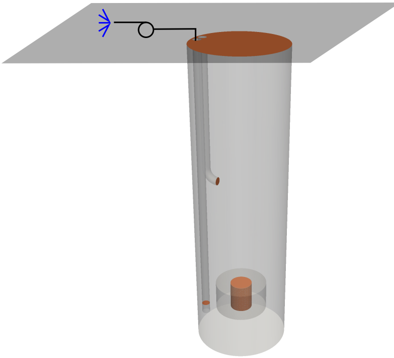
t=0s t=1600s t=3200s t=4800s

~ ***Video*** ~

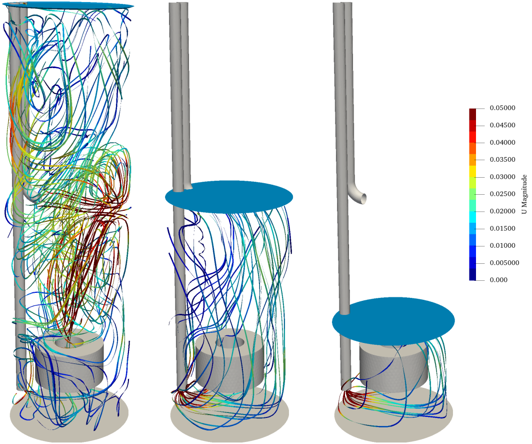


Loss of coolant accident

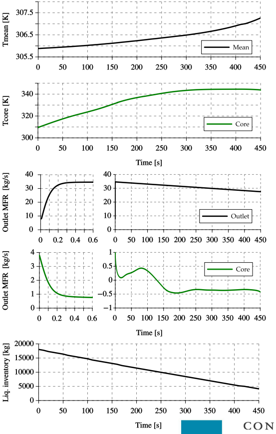
- Event: A downstream 2A-LOCA pump condition was simulated.
- Solver: The compressible Volume of Fluid (VOF) solver (compressibleInterFoam) was used to track the free surface motion.
- Core power: A constant core power of 100kW was assumed.
- Consequences: A quick increase of the flow rate in the pump is caused by the loss of downstream circuit.



Loss of coolant accident



t = 0s   t = 250s   t = 450s



∼ ***Video*** ∼

UNL
CONICET
C I M E C

Conclusions

- The work address with the implementation of a new dynamic boundary condition in OpenFOAM 5.0, for coupling 0D and 3D CFD domains.
- The model was used to simulate operational and accidental conditions in the TRIGA MARK I reactor with different core powers.
- The TRIGA simulation results were in good agreement respect the design and experimental data.
- **Loss of forced convection event:** The pump shutdown quickly reduced the flow in the external circuit. The natural convection governs the flow circulation. In consequence, the reactor heats with a constant rate.
- **Loss of coolant event:** A quick increase of the flow rate in the pump is leaded by the loss of downstream circuit. The core is uncovered after 500s. Even though core temperature rising, saturation condition is not achieved.

Acknowledgments

The authors would like to thank:

- Universidade Federal de Minas Gerais.
- The Brazilian agencies CAPES, CNPq and FAPEMIG.
- Universidad Nacional del Litoral (CAI+D 2016 PIC 50420150100067LI).
- Agencia Nacional de Promoción Científica y Tecnológica (PICT 2016-2908)
- Consejo Nacional de Investigaciones Científicas y Tecnológicas.

UNL CONICET

C I M E C

Thank for your attention!

UNL CONICET
C I M E C