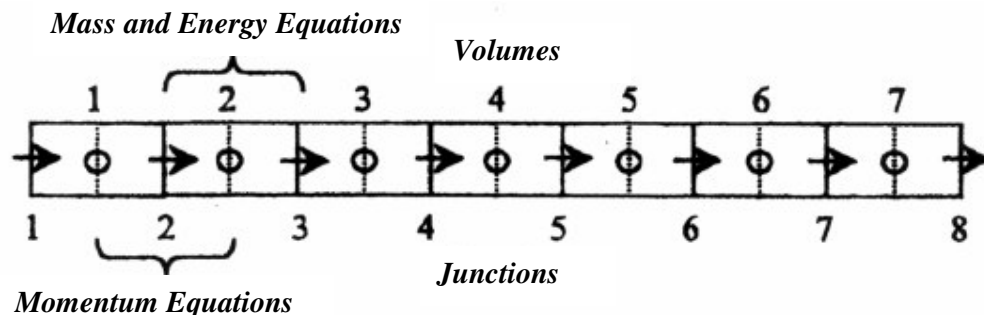


# AN ESSENTIAL SUMMARY ABOUT THE RELAP5 CODE

The demonstrative example in the following section has the aim *to make the reader just a bit acquainted* with basic choices in the use of a widespread thermal-hydraulic system code.

In order to let the reader be aware of the main working principles of it, the following characteristics of the code are summarised:

- it makes use of a two-fluid model in 1D form (latest versions include also multidimensional capabilities, not addressed here);
- the balance equations of mass, momentum and energy for the two fields are solved in control volume form subdividing the systems into volumes, also called *nodes*, joined by *junctions*, having the role of openings through which the fluid may enter or get out;
- in the case of a 1D pipe, the system is mostly as described in the figure:



where it is seen that the code uses of the well known technique of “*staggered meshes*”, i.e., it solves mass and energy equations “*within*” nodes and momentum equations “*across*” the junctions.

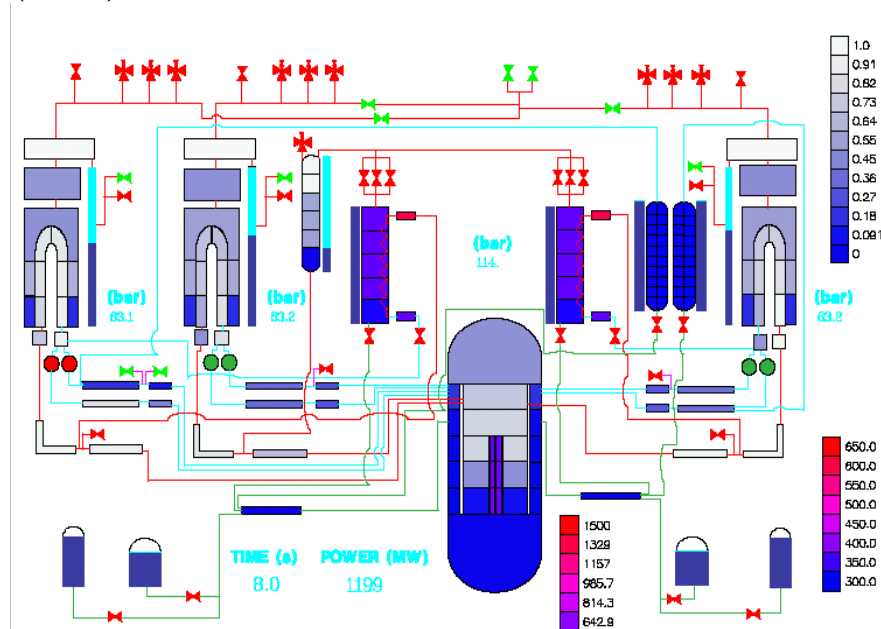
- in more complex situations, nodes may have more than two junctions (in such cases they are called “*branches*”) to simulate particular features (e.g., tee junctions, plena in multichannel systems, etc.);
- about the constitutive laws used in the code, there is a plenty of models for both two-phase and single phase flows;
- in each control volume, the solution of equations is made mostly as for lumped parameter examples (the thing is slightly more complex than that, anyway...): so we can recognise that,

*in the case of the adoption of control volume formulation, solving balance equations in a 1D or even a 3D field requires the solution of many “lumped parameter” equations combined with each other*

An essential step in using the code is *setting up a nodalization*, i.e., a description of the system, no matter its complexity, in terms of nodes and junctions, properly connected with each other.

This job must be performed with due care, considering the physical features of the systems and how the code is able to simulate them: inaccurate or wrong nodalizations will possibly lead to inaccurate or wrong results.

The following figure shows a schematic view of a RELAP5 nodalization for a PWR as obtained in the past by the Nuclear Plant Analyzer (NPA).



Considering the above, it is recommended to look at the examples of application reported below trying to understand the general process required to pass from a physical system to its representation for a system code.

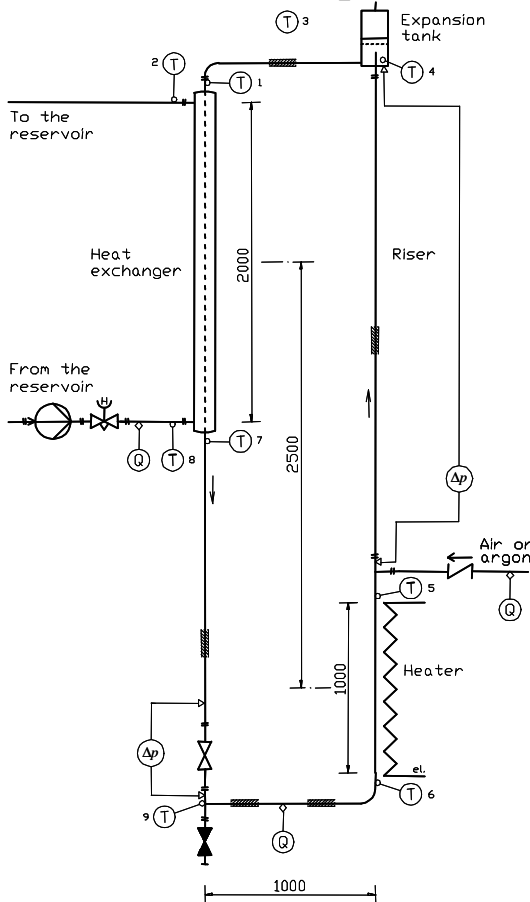
The aim of these demonstrations is not to teach the reader how to use the code (such an objective will need a specific Course!), but to grasp the basic techniques used in modelling nuclear systems with a large thermal-hydraulic system code.

# A NATURAL CIRCULATION LOOP

## The ANGIE Facility Installed at DIMNP

### *Description of the physical system*

The figure below shows a sketch of the experimental facility adopted in the present work. It mainly consists of 1½" I.D. pipes welded or connected by flanges to each other. The main components that can be identified in the loop are:

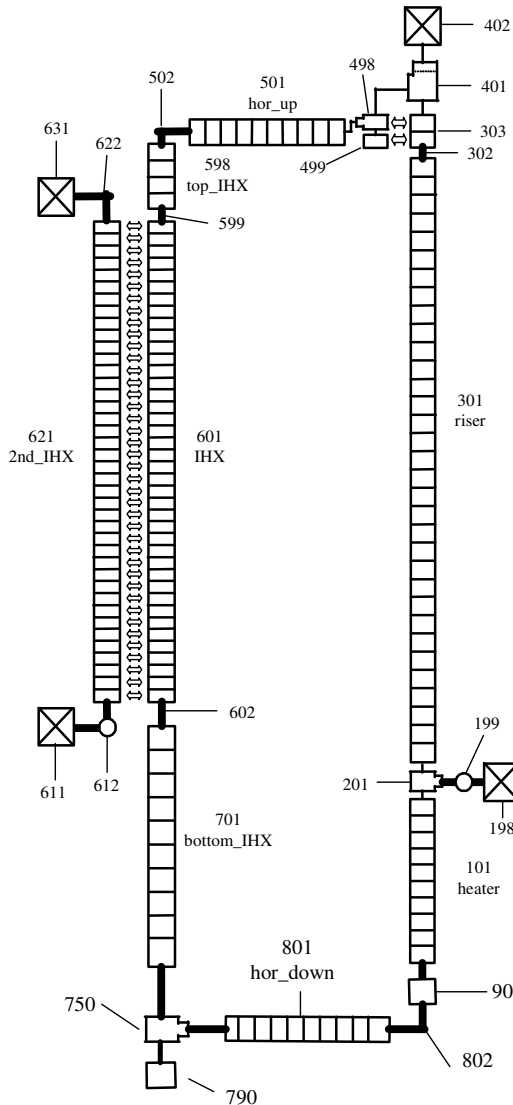


- a 1 m long heated section, with a maximum electrical power of 5 kW, consisting of four 250 mm long band heaters capable of 1.25 kW each;
- a 3 m tall riser, at whose bottom gas injection (presently air) is available through an appropriate nozzle, thus making it possible to study assisted natural circulation;
- an expansion tank, located at the top of the riser, having the twofold purpose to allow for thermal expansion of the fluid during transient operation and to separate the injected gas from the liquid in case of gas-injection enhanced circulation;
- a 2 m long heat exchanger, cooled

by water as a secondary fluid; it consists of a two coaxial tubes, in whose internal gap a helical stainless steel sheet is wound on the outer surface of the internal pipe to enhance heat transfer by swirling flow;

- a spherical valve, which allows for varying the loop friction characteristics;
- a fluid dump line, equipped with a similar spherical valve which is normally kept closed.

## Description of the RELAP5 nodalization



The nodalization adopted in these analyses is shown in the Figure.<sup>1</sup>

It consists of different *pipe*, *junction* and *branch* components, each one equipped with one or more heated structures (not shown in the figure).

Time-dependent volumes and junctions have been used to represent boundary conditions, as the imposed inlet flow and thermal fluid conditions in the secondary loop.

Particular attention was put in representing the expansion vessel, having also functions of air separation, by the use of branch and separator components.

In this frame, the correct representation of the liquid content and of the separation capabilities is fundamental for predicting both steady-state and transient behaviour.

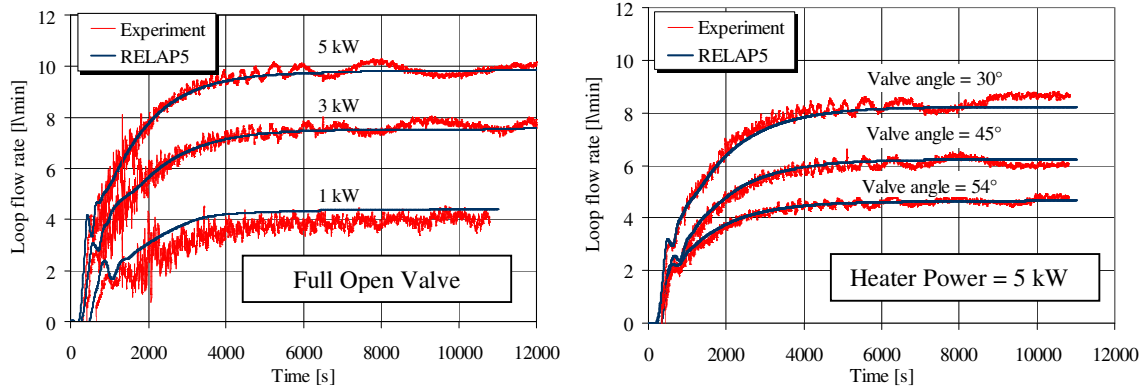
In addition, the particular flow path inside the expansion vessel introduces pressure losses that must be accurately estimated.

Other features that deserved particular attention were some details of the heater structures, including the representation of the thin air gap between the band heater and the heated pipe, which affects the time needed by the loop to reach steady-conditions, and the secondary side of the heat exchanger section, including an helical metallic sheet introduced to promote turbulence.

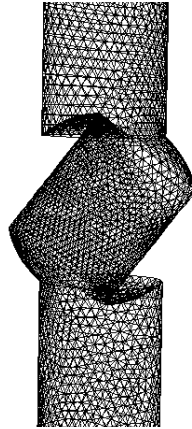
<sup>(1)</sup> The work is drawn from the doctoral work of Ing. Mariano Tarantino.

### ***Obtained Results***

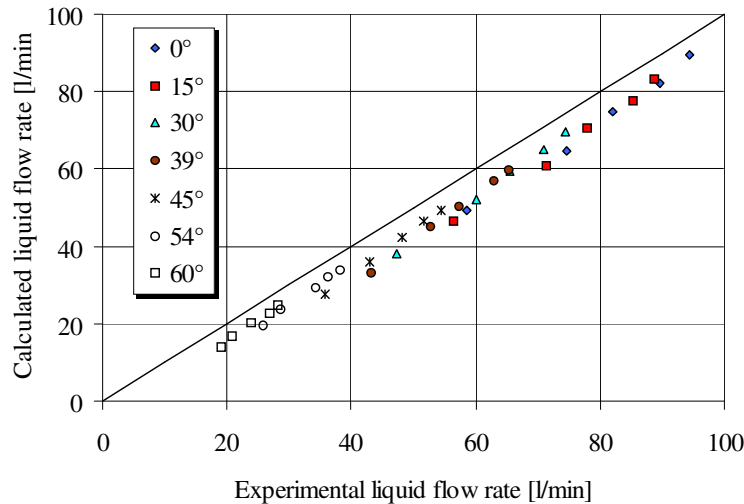
**Prediction of natural circulation behaviour showed a good degree of accuracy at different valve throttling.**



**The pressure drop coefficient to be given in input to the code was both measured experimentally and predicted by FLUENT calculations.**

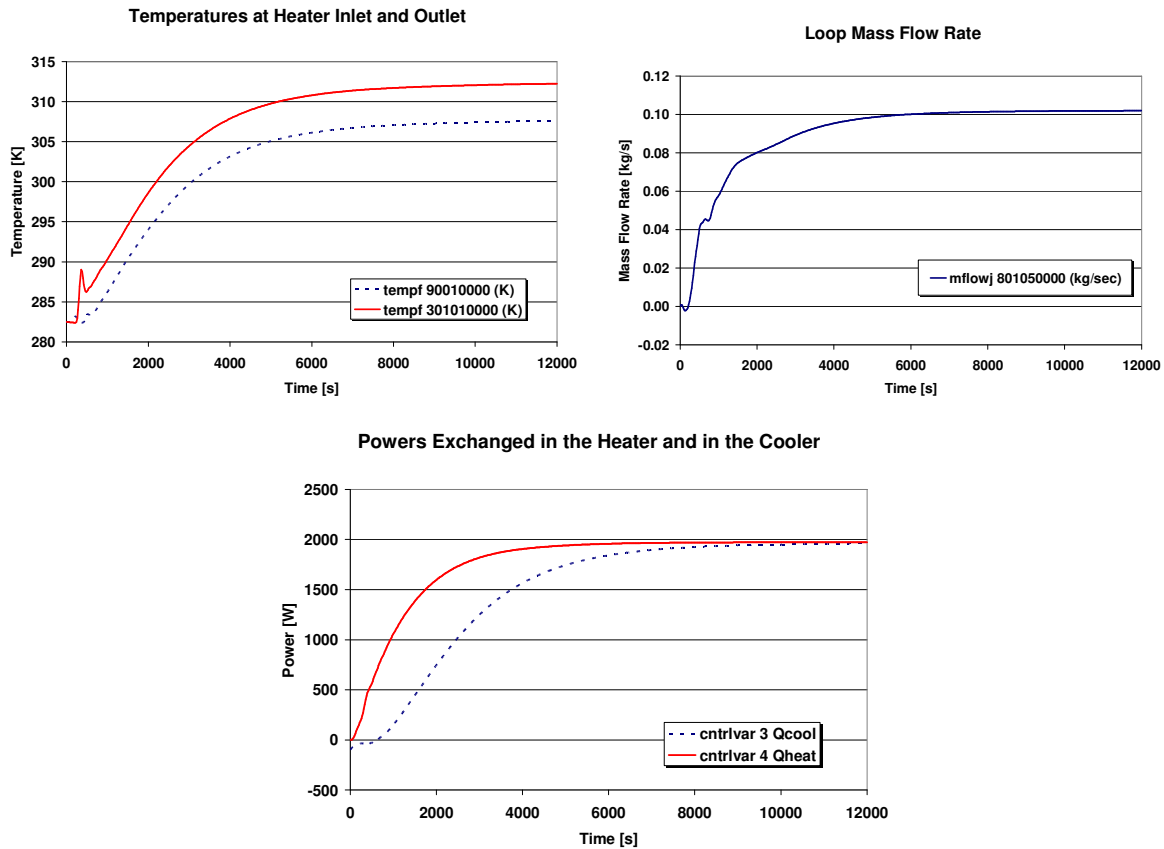


**A slight underprediction of the liquid flow rate in air-lift operating conditions was obtained.**



In the exercise, we will revise the input deck set up for an experimental test and we will run additional cases at different power.

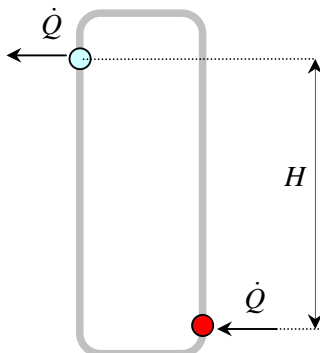
The base case was selected for an experiment performed at a nominal heating power of 2 kW. The results obtained in that case are the following.



In order to put the basis for an approximate hand calculation of flow rate, the following simple theory is reported.

*Steady natural circulation in a simplified loop*

- Let us consider a loop with uniform cross section and point heat source and sink.



- Neglecting the heat losses to the environment, a uniform temperature distribution is obtained at the exit of both the heater and the cooler
- In such case, in fact, the steady-state energy equation states

$$\frac{\partial T}{\partial s} = 0$$

- On the other hand, the balance equation for heated and cooled branches becomes

$$\frac{W}{\rho_0 A} \frac{dT}{ds} = \frac{4q''}{\rho_0 c_p D} \Rightarrow \frac{dT}{ds} = \frac{4q'' A}{c_p W D} \Rightarrow \frac{dT}{ds} = \frac{q'' \pi D}{c_p W}$$

$$\Rightarrow dT = \frac{q'' \pi D}{c_p W} ds \Rightarrow \Delta T = \frac{q'' \pi D}{c_p W} \Delta s$$

- The total exchange power in our case is

$$\dot{Q} = q'' \pi D \Delta s$$

then

$$\Delta T = \frac{\dot{Q}}{c_p W}$$

- Being in steady state conditions it is

$$\underbrace{\rho_0 g \beta \oint_{loop} T(s) \sin \theta(s) ds}_{buoyancy} = \underbrace{\left[ \left( \sum_k K_k + \frac{fL}{D} \right) \right] \frac{|W|W}{2\rho_0 A^2}}_{friction}$$

- Since only vertical and horizontal pipes are available where  $\sin \theta(s)$  is 1, 0 or -1:
  - horizontal pipes do not give any contribution the integral at the RHS
  - only the vertical pipes included within the heights of the heater and the cooler give a positive contribution
  - in fact, the other vertical pipes give contributions that cancel with each other
- It is therefore:

$$\oint_{loop} T(s) \sin \theta(s) ds = T_{hot\ leg} H - T_{cold\ leg} H = \Delta T H = \frac{\dot{Q} H}{c_p W}$$

- By combining this relationship with momentum equation, we find

$$\rho_0 g \beta \frac{\dot{Q} H}{c_p W} = \left[ \left( \sum_k K_k + \frac{fL}{D} \right) \right] \frac{|W|W}{2\rho_0 A^2} \Rightarrow \rho_0 g \beta \frac{\dot{Q} H}{c_p} = \left[ \left( \sum_k K_k + \frac{fL}{D} \right) \right] \frac{|W|W^2}{2\rho_0 A^2}$$

and the steady state flow rate is obtained as a function of the other parameters

$$W = \sqrt[3]{\frac{2\rho_0^2 g \beta A^2 \dot{Q} H}{c_p \left( \sum_k K_k + \frac{f(Re)L}{D} \right)}}$$