MODELLING OF A RADIAL PUMP FAST STARTUP WITH THE CATHARE-3 CODE AND ANALYSE OF THE LOOP RESPONSE

L. Matteo¹,² - G. Mauger² - A. Dazin¹ - N. Tauveron³

1 - Fluid Mechanics Laboratory of Lille - Kampe de Feriet, Arts et Metiers ParisTech. FRE CNRS 2017, Lille, France
2 - Thermalhydraulic and Fluid Mechanics Section, CEA Paris-Saclay university, France, laura.matteo@cea.fr
3 - Thermal Biomass & Hydrogen Department, CEA, Grenoble, France

ABSTRACT
A predictive transient two-phase flow rotodynamic pump model has been developed in the Code for Analysis of THermalhydraulics during an Accident of Reactor and safety Evaluation (CATHARE-3). Flow inside parts of the pump (suction, impeller, diffuser, volute) is computed according to a one-dimensional discretization following a mean flow path. Transient governing equations of the model are solved using a implicit resolution method and integrated along the curvilinear abscissa of the element. This model has been first qualified at the component scale during years 2017-18 by comparison to an existing experimental database. The present study aims at extending the validation at the system scale: a whole experimental test loop is modelled. The ability of the transient pump model to predict flow rate, head and torque as a function of time during a 1-second pump fast start-up is presented. Evolution of the pressure upstream and downstream from the centrifugal pump obtained in the simulation is similar to the measurements during the transient.

KEYWORDS
ROTODYNAMIC PUMP, TRANSIENT MODELLING, PERFORMANCE PREDICTION, CATHARE-3 CODE

NOMENCLATURE

\( \alpha \): absolute angle
\( \beta \): relative angle
\( b \): width
\( D \): diameter
\( H \): head
\( N \): rotational speed
\( N_q \): specific speed (European definition)
\( Q \): flow rate
\( U \): blade velocity
\( V \): absolute velocity
\( W \): relative velocity
\( z \): curvilinear abscissa
2, 4, 6: resp. impeller, diffuser and volute outlet
INTRODUCTION

The French Alternative Energies and Atomic Energy Commission (CEA) is currently developing a predictive rotodynamic pump model in the Code for Analysis of Thermalhydraulics during an Accident of Reactor and Safety Evaluation (CATHARE-3) [17, 18]. The aim is to predict the pump behaviour during reactor two-phase flow transients studied in the frame of system safety analyses [8]. This work is mainly supported by the Generation IV (Gen-IV) nuclear reactor program of CEA. Industrial and academic partners are involved, such as Framatome, Electricité de France (EDF), CEntre Technique des Industries Mécaniques (CETIM), ArianeGroup and the Fluid Mechanics Laboratory of Lille (LMFL). Generation IV reactors are designed to optimize the nuclear fuel cycle and minimize waste generation compared to Generation II and III Pressurized light Water Reactors (PWR). In France, the Gen-IV Sodium cooled Fast neutrons Reactor concept (SFR) is studied [23]. Safety studies of such reactors lead to the necessity of simulating long time (thousand-second) two-phase system transients with a computational (CPU) time smaller or equal to the real time of the transient [22]. CATHARE-3 is the French reference thermalhydraulic system code for nuclear safety analyses. It is owned and developed since 1979 by CEA and its partners EDF, Framatome and Institut de Radioprotection et Sûreté Nucléaire (IRSN) [3]. One-dimensional (1D), three-dimensional (3D) or point (0D) hydraulic elements can be associated together to represent a whole facility. They respectively correspond to one or three directions allowed for the fluid, and to components where fluid velocity is negligible such as capacities. Thermal and hydraulic submodules (as warming walls, valves, pumps, turbines...etc) can be added to the main hydraulic elements to respectively take into account thermal transfer, flow limitation, pressure rise or pressure drop. Six local and instantaneous balance equations (mass, momentum and energy for each phase) make possible liquid and gas representation for transient calculations. Mechanical and thermal disequilibrium between phases can thus be represented [13]. Phase average imposes the use of physical closure laws in the balance equations system [4]. Resolution is made using a implicit scheme which allows variation of the time step during the computation.

The current method to model rotodynamic pumps with the CATHARE-3 code consists in locating the pump at a point of the circuit and giving non-dimensional head and torque performance curves as input data. Associated momentum and energy contributions are taken into account via source terms in the balance equations. This model is not predictive and cannot be used for Generation IV two-phase flow applications as performances degradation is modelled using a void fraction function qualified for PWR applications. For this reason, it was decided to implement a predictive transient, two-phase flow rotodynamic pump model in the CATHARE code and qualify results at component scale (pump) and system scale (reactor) with respect to available experimental data. The global strategy of the CATHARE-3 1D pump model development project and the first pump-scale verification and validation results have been presented earlier in Matteo et al [17] and [18]. In the present paper, the model is qualified at the loop system scale during a pump fast startup transient (from 0 to nominal speed in approximately 1 second). As the whole loop is modelled using the CATHARE-3 code, interaction between the pump and the system can be analyzed. The evolution of the rotational speed as a function of time is the only imposed boundary condition. The loop support of this study is the DERAP test bench installed at LMFL [11].

According to the authors’ knowledge, most of the 1D performance prediction models, even the recent ones, are based on the Euler equations associated to global losses and slip factor correlations [12, 16]. The originality of the present project is to build a real 1D model.
mean streamline in the different parts of the pumps is meshed-able to predict the performance of mixed and radial flow pumps in purely liquid or gas/liquid, and four quadrant operations.

1 DERAP PUMP DESCRIPTION AND 1D MODELLING

1.1 Description of the DERAP centrifugal pump

The DERAP pump has been set at the LMFL to study the pump behaviour in cavitating and non-cavitating operating conditions during fast startups. It was the support of the successive works of Ghelici [1, 14], Picavet [20, 21, 2], Bolpaire [6, 5, 7] and Duplaa [9, 11, 10]. Measurements have been conducted in single-phase and two-phase cavitating conditions in steady and transient regimes. The pump has been operated only in the first quadrant (positive rotational speed and flow rate). DERAP impeller is shown on figures 1 and 2 below. The pump specific speed is 12.3 according to the common European definition of specific speed \( N_q = \frac{N \sqrt{Q}}{H^{3/4}} \) (Gülich [15]). DERAP nominal point is \( Q_N = 23m^3/h, H_N = 50m \) and \( N_N = 2900rpm \).

![Figure 1: DERAP impeller picture and geometric specifications.](image1)

![Figure 2: DERAP impeller scheme.](image2)

A vaneless diffuser of constant width (7 mm), whose inlet and outlet diameters are respectively 206 and 240 mm, is located downstream the impeller. A simple diverging volute is located downstream the diffuser as shown on figure 3 and is then connected to the discharge pipe. More information on DERAP pump geometric data is available in Duplaa [11]. In the present study, only single-phase characteristics are used to qualify the predictive 1D-pump model. The two-phase measurements will be used in a further study dedicated to two-phase qualification of the model.

![Figure 3: DERAP pump scheme.](image3)

1.2 Modelling of the pump

As shown on figure 4 representing the modelling of the DERAP centrifugal pump, each part of the pump is modelled using a one-dimensional element with approximately 30 to 60 cells. The diffuser and the volute are defined together in the same element.

![Figure 4: CATHARE-3 1D modelling of the DERAP pump](image4)
Flow inside parts of the pump (suction, impeller, diffuser, volute) is computed according to a one-dimensional discretization following a mean flow path, as represented respectively on figures 5 and 6 for the case of the impeller and the diffuser parts.

In the impeller, the mean flow path is defined by the evolution of the mean relative angle $\beta$ from the impeller inlet to the outlet. The local relative angle $\beta$ of the cell is calculated assuming that $\sin(\beta(I))$ is a linear function of the cell number $I$.

Remaining discrepancies observable in a previous study [17] have been suppressed by improving the modelling of the pump stator. Previously, the impeller was the only part entirely modelled: centrifugal acceleration and relative velocity were calculated inside the impeller, desadaptation and recirculation losses were taken into account. But the flow was ideally redressed directly when entering the diffuser, what was a first simple approach. The flow redress corresponds to the transition from a two-component velocity to a purely axial velocity according to the curvilinear abscissa. In the present study, the diffuser part is entirely modelled taking into account the two components of the velocity in the diffuser and volute parts.

The mean flow path supposed to be followed by the fluid in the vaneless diffuser corresponds to a logarithmic spiral, as represented on figure ??°. This assumption corresponds to constant absolute angle $\alpha$ in the vaneless diffuser.

In the volute part, flow is progressively redressed in order to get a one-component axial velocity at the end of the volute. Direction of the fluid is given by the evolution of the absolute angle $\alpha$ along the volute. The following relation defines the local absolute angle for each cell of the mesh:

$$\sin(\alpha(I)) = \sin(\alpha_2) + (\sin(\pi/2) - \sin(\alpha_2)) \frac{(z(I) - z_4)^2}{(z_6 - z_4)^2}$$

Additionally to the representation of the velocity profile along the pump, friction, desadaptation, recirculation and diffusion losses are modelled. See [18] for the expression of these correlations. Friction losses depend on the square of the local velocity and the surface rugosity. They are calculated according to the Zigrang and Sylvester correlation [24, 19]. The roughness of the pipe material is provided by the user (here stainless steel). Desadaptation losses increase out of flow rate and rotational speed design conditions. They are located at the inlet of the impeller part. Recirculations in the impeller inter-blade channels act at low flow rate conditions as a power loss. It is spread along the impeller part. Finally, the diffusion losses due to the divergent geometry of the volute are calculated and spread along the volute mesh.
1.3 Prediction of DERAP steady performance curves

First of all, the prediction of steady performance curves of the DERAP centrifugal pump using the developed model is presented in the following. To produce head and torque performance curves of the DERAP pump, a slow evolution of the flow rate (from 10% to 150%) is imposed at the inlet boundary condition of the modelled pump while keeping a constant nominal rotational speed. Total CPU time of the computation is 6 seconds using one core of a classic deck machine. Head and torque performance curves can be observed on the following figures 7 and 8.

![Figure 7: Head performance curve.](image1)

![Figure 8: Torque performance curve.](image2)

Experimental measurements of the torque are located on the shaft what produces motor torque data which include the friction torque. These data cannot directly be compared to the hydraulic torque predicted by the 1D pump model. But, assuming the friction torque as independent of flow rate, they can be estimated at the nominal flow rate and removed on the whole flow rate range. It can be seen as an efficiency applied on the motor torque data to obtain estimated hydraulic torque experimental values, as shown on figure 8.

One of the interests of the CATHARE-3 1D pump model is that profiles inside pump elements can be obtained during the computation. As an example, figures 9 and 11 below have been drawn at nominal rotational speed and flow rate. Figure 9 and 10 respectively represent the total and static pressures and enthalpies along the pump elements. Figure 11 represents the velocity along the pump elements.

![Figure 9: Static and total pressure profiles along the flow path at nominal operating point.](image3)

![Figure 10: Static and total enthalpy profiles along the flow path at nominal operating point.](image4)
It has to be noted that inside the rotating part (impeller) the velocity is the relative one, whereas it is the absolute one in the fixed parts (with only one axial component in the suction part, and with two components -axial and tangential- in the diffuser and volute parts). High velocities in the diffuser part can be observed, which cause friction losses. The flow direction evolves in the diffuser and volute parts. The absolute flow angle $\alpha$ (calculated towards the tangential direction) is gradually increased from its value at the diffuser outlet to $\frac{\pi}{2}$ according to a fitting as previously explained.

The flow section is calculated depending on the local absolute angle along the diffuser and the volute according to the following relation: $S = S_m \cdot \sin(\alpha)$. To validate or improve this profiles, more local experimental data or computations results are necessary. Computational Flow Dynamics (CFD) computations could be used following an up-scaling method to validate this type of 1D results. This will probably be used in the future of this project.

In the following, the modelling of the fast startup transient using the CATHARE-3 code is presented. Simulation results are compared to transient experimental measurements.

2 CATHARE-3 DERAP facility modelling

2.1 Whole facility description

The DERAP test loop set at the LMFL allows two configurations: a closed circuit with the suction and delivery pipes connected to a single tank, and an open circuit with two separate tanks, whose pressures can be set independently. A valve is used to switch from one to the other. The closed configuration has been used for the startup experiments simulated here. Pressure is imposed at the top of the tank (see figure 12) using a injection/extraction valve. A control valve is located on the delivery pipe, whose purpose is to control the flow rate.

Two pressure transducers are used on the test rig : one 50mm upstream of the impeller on the suction pipe and another 100mm downstream on the delivery pipe. Before two-phase tests,
Duplaa [11] conducted a one-phase fast startup, while imposing a sufficient pressure in the tank to avoid cavitation (2.8 bars). Simulation results of this single-phase test are presented in the following sections.

### 2.2 CATHARE-3 modelling

The CATHARE-3 modelling of the circuit was done using 1D and 0D elements. Compared to the figure 4, the suction and delivery elements have been extended in order to represent the whole pipes of the DERAP loop. A singular pressure loss coefficient is used to take into account the losses due to the elbow on the delivery pipe.

![CATHARE modelling of the complete circuit](image)

The tank is a large fluid capacity with several connections, thus a 0D element has been used to represent it. Its two junctions are linked to the suction and delivery pipes (figure 13).

The same geometry as the one used in the component scale data deck (figure 4) is used for the impeller, diffuser and volute parts. Only the orientation of elements has been adapted to well connect the whole circuit. In particular, the volute outlet height of the experimental setup is respected in the simulation. These 1D elements are placed on the bottom right of figure 13.

### 2.3 Fast startup transient

The experimental procedure conducted by Duplaa [11] was the following:

- the pump was started slowly to reach its nominal speed and adjust the flow control valve in order to obtain the desired final flow rate
- the pump was stopped and after stabilization of the circuit, the fast startup was launched

In the computation, the flow control valve is modelled by a singular pressure drop which is chosen to obtain the desired flow rate at the end of the startup transient. The experimental position of the flow control valve in the circuit is respected. Pressure drop caused by the singular element is shown on figure 14 below.
Similarly to what was done in the experiment, pressure at the top of the tank is regulated at 2.8 bar. The tank is full of liquid except at the very top where there is a free surface with compressed air above it. The pressure regulation in the simulation is done in the same way as in the experiment, using a gas injection/extraction valve. Injected gas is non-condensable air gas. Gas flow evolution at the pressure regulation valve is shown on figure 15. It can be seen that some air is injected at the top of the tank in the first 20 seconds in order to rise pressure to 2.8 bar.

In the first 100 seconds of the CATHARE computation, the circuit is stabilized with a pump almost stopped. This means that a residual rotational speed (2%ωN) is defined in order to initialize the circuit with a small flow rate, as 0% flow rate computations are to avoid for numerical resolution reasons.

The fast startup begins at 100 s and finishes at 101.5 s. Rotational speed evolution is imposed by a time dependent law. It follows the experimental one as shown on figure 16. The aim of this validation work is to predict the flow rate, head (pressure difference) and torque evolutions and to compare it to available experimental data.

Predicted flow rate and pressure difference evolutions are respectively presented on figures 17 and 18. A good agreement is obtained when comparing predictions of the model to experimental data. The differences between the flow rate evolutions are probably due to the measurement uncertainty as the flow evolution is very difficult to measure during a fast startup.
Torque evolution is shown on figure 19. Observable discrepancies are mostly due to the fact that the plotted experimental torque is a motor torque measured on the shaft, whereas the computed torque is an hydraulic torque. A peak of the measured motor torque can be observed before 100.4 seconds. This is due to the mechanical inertia of the motor. As the motor is not modelled in this study (only the time evolution of the rotational speed is imposed), this peak cannot be predicted by the simulation. Moreover, it can be seen that the hydraulic torque computed by the model before the startup (t=100 s) is negative. This is due to the initial flow rate and rotational speed conditions that are not completely set to zero. This reflects the difficulties of the model to respect the similarity laws at very low rotational speed and flow rate.

![Pressure difference evolution](image1)

**Figure 18:** Pressure difference evolution (predicted by the model and compared to experiment).

With the complete loop modelling, pressures can be analyzed at different points in the circuit. On figure 20, computed and experimental suction outlet pressure and discharge pipe inlet pressure are shown. It can be seen that pressure is rising at the discharge pipe inlet (this corresponds to the pump pressure rise) unlike it is dropping at the suction pipe outlet. The pressure dip before 100.6 seconds is caused by the flow inertia effects in the pipe. These two pressure evolutions are well predicted by the model by comparison to experimental pressure evolutions from Duplaa et al [11].

![Evolution of static pressure in the circuit](image2)

**Figure 20:** Pressure evolutions (predicted by the model and compared to experiment).

**CONCLUSIONS**

This paper presents a whole pump loop modelled with the CATHARE-3 thermal-hydraulic code. First, the 1D modelling of DERAP pump "alone" is presented and predicted performances are compared to experimental ones. A good agreement is obtained. Then, the complete system scale modelling is presented and results obtained during a 1-second fast startup transient are analyzed. With the complete loop modelling, it is possible to follow the evolution of several
physical quantities of interest at different points of the circuit. This is done in the present study to analyze the evolution of pressure upstream and downstream from the pump, what allows to observe the flow inertia effects in the loop. This first validation at loop system scale is a step bringing to analysis of reactor system scale response during hypothetical accidental transients of interest (pump seizure, pump stop on inertia, pump restart, cavitation occurring in working pumps, inverse flow rate and/or rotation speed...). This paper presents one-phase computations, but two-phase computations have started recently and the next step is to validate the fast startup transient with cavitation occurring in the pump (see Duplaa [11] for experimental description of the two-phase tests).

ACKNOWLEDGEMENTS
Thanks to the LMFL academic lab and Framatome, ArianeGroup, CETIM and EDF industrial companies who support the project by providing data, funding and knowledge.

References


