Computation of unstable flows using system codes

J C Ferreri

Autoridad Regulatoria Nuclear, Av. Del Libertador 8250, (1429) Buenos Aires, Argentina
E-mail: jferreri@arn.gob.ar

Abstract. The prediction of unstable flows by means of so-called systems-codes is discussed in this paper. In the nuclear field, verification and validation (V&V) of codes applicable to predict the behavior of complex nuclear installations is a matter of huge collective effort presently. Multi-Physics–Multiple-Scales models, coupled with the needed wall laws is just one example if length scales go into wall layers or minute geometrical details. The validity of physical models as applicable to situations of interest in developing nuclear technology systems (e.g. natural circulation systems, super-critical conditions flows, very high temperature gas-cooled reactors) is now under consideration in the technical literature. This even applies to very simple problems, like cavity flows or one-dimensional (1D) systems. Some basic problems are presented for V&V that must correctly be solved by any systems code. Based on this, conclusions are drawn in relation to some present systems-codes, regarding the sensitivity to code options influencing its numerical approach and physical closure correlations.

1. Introduction

It must be recognized that the development of systems codes, like the ones usually utilized in the computation of the thermal-hydraulics of nuclear installation in relation to the nuclear safety evaluations impose huge efforts for a SYS-TH code developer group, in charge of its development, implementation and V&V. When adopted by a large technical community during decades, like in the case of RELAP5 [1], these codes are subject to the testing of practically all their options. Problems are continuously reported to the developers. These actions originate corrections and periodic updates. Needless to say, life is far easier for a SYS-TH code user (or one reporting on code capabilities and limitations) than for a code developer. Assessments constitute a combination of many aspects, involving standard problems of V&V and the code user, who gets results after setting up a discretization for the problem or installation under analysis. The present paradigm consists in the coupling of complex codes for analyzing complex systems. This increases the complexity of the problem, because of non-linearity of the models and the uncertainty related to code shortcomings and accuracy and also to possible implementation errors. Things become worse if different levels of approximations are used, like in the coupling of one-dimensional, time dependent systems codes with detailed nodalizations coming from computations fluid dynamics (CFD) codes. The present trend

---

1 Revised, full text and updated version of a presentation to FONESYS: FORUM & NETWORK OF SYS-TH CODES IN NUCLEAR REACTOR THERMAL-HYDRAULICS, San Piero a Grado Nuclear Research Group headquarters, University of Pisa, Pisa, Italy, May 13-14, 2010
2 Member of CIC, Conicet and Fellow of National Academy of Sciences of Buenos Aires, Argentina
consists in using CFD codes as a support to nuclear safety analysis, given the lack of enough detailed data to perform the V&V of CFD codes in situations of real practical interest in Nuclear Engineering.

CFD issues have been purposely left almost apart (only one example will be considered), because it would deserve much more than this presentation. In this case, discussion should be more focused into the physics and its associated equations because -from the numerical/computational point of view- the nuclear application of CFD codes benefits from well-established methods of many other branches of Engineering, such as Computational Mechanics.

The validity of physical models as applicable to situations of interest in developing nuclear technology systems (e.g. natural circulation systems, super-critical conditions flows, very high temperature gas-cooled reactors) is now under consideration in the technical literature. This even applies to very simple problems, like cavity flows or 1D systems. In the nuclear field, V&V is a matter of huge collective effort presently. Multi-Physics/Multiple-Scales, coupled with the needed wall laws is just one example if length scales go into wall layers or minute geometrical details.

The physical model of an installation implies the specification of a system of conservation partial differential equations (PDEs) and their initial and boundary conditions (I&BCs). The physical model may sometimes specify a mathematical problem with undesired properties (it is not a new here that an ill-posed system of PDEs may be accepted by a whole technical community). The mathematical analysis of the PDEs establish how the I&BCs must be formulated and specified. The numerical formulation relates this mathematical model to an algorithm allowing computing the solution. The numerical model may imply approximations that are not coherent with the mathematical properties of the system, even when said approximations are consistent and stable (implying solution convergence). I&BCs are usually coherent with the mathematical model despite the equations really solved (i.e. the algebraic system including the effects of truncation errors) by the computer implementation of the numerical model. As a consequence of the above considerations, a solution is “always” obtained (due to the so-called “code robustness”), which may show properties different from what must be obtained from the exact solution of the mathematical model, even more in the case of stabilization or posedness modification.

Differences may come from different causes, among them:

- Non detected errors in the code implementing a given numerical algorithm
- Artificial, numerical diffusion due to low order numerical approximation coupled with coarse nodalization
- Appropriate, stabilizing interphase correlations
- Use of steady-state correlations valid outside of the range of the code application (i.e. using friction laws, heat transfer correlations, etc., valid for steady-state, fully developed flow, in real life installation geometries)
- Inadequacy of component models, lack of appropriate closure laws
- User effects

Many other causes may not be easily discernible, like in the case of compensating errors originating in closure correlations uncertainties and problem representation specification.

The “User effects” is perhaps the most influencing source of differences between the solution of an exact code implementation of a numerical algorithmic representation of a physical model and an experiment. It has received great attention since many years ago. One example of this may be found in [2] and quite recently in [3]. In all these years, the formalization of the so-called “User effects” definition has consolidated and, again e.g. in [3], is stated as “the cumulative effect of (code) user community members producing a range of solutions for a well-defined problem with rigorously specified boundary and initial conditions”. Contributions to these effects are: incorrect nodalization, wrong specification of the many input flags for a given system, use of different types of processors, etc. Ways to reduce these effects are the quality assurance of the input decks preparation and user training. User effects can not be eliminated completely.

In this paper, some results coming from a particular systems code (RELAP5/MOD3.3) and from in-house developed codes exercising are shown in the case of unstable single and two-phase flows.
few examples would hopefully illustrate some considerations to be taken into account for the simulation of this subclass of physical problems.

2. Single-phase, unstable, natural circulation flows in closed loops

This Author and his colleague at the University of Pisa, Prof. W Ambrosini, have studied the effects of nodalizations, closure correlations and their interaction in the prediction of flow behavior in single-phase, unstable, natural circulation flows since many years ago. One prototypical problem with known stability map boundary has been considered, namely the so-called Welander’s loop [4], which may be considered a verification case. This loop consists of two parallel adiabatic tubes with a heat source at bottom and a heat sink at top. It has been represented with “real life” dimensions in our studies, namely two pipes 10 m height and 0.1 m in diameter and “pointwise” heat source and sink (0.1 m in length). In [5] the effects of different order nodalizations and number of nodes have been extensively discussed and illustrated, showing how the stability map of this system changes. In reference [6] three different geometries were considered, namely toroidal, square and again the Welander’s loop, showing the effects of friction laws. In [7] all these results have been reviewed and a particular way of sensitivity analysis to parameters was introduced in this reference and a new one in [8]. Just to give an example of the coupled effects of nodes number and friction in this prototypical problem, the following may be mentioned: consideration of a numerical first order fully upwind approximation for the equation of energy and 90 nodes gives a correct unstable flow solution showing flow reversals. If 66 nodes are used, the flow becomes neutrally stable. Considering again 90 nodes and a friction factor magnified 1.43 times, the flow becomes again neutrally stable. This is just an example of the well known damping effects of numerical viscosity that, in this case, may be simply assimilated to friction.

One interesting example of non-conservative results (i.e. results that show stability for a unstable flow situation) using a generalized 1D, single-phase code [9] for natural circulation in loops was documented in [10], showing how a physically fully unstable behavior map may have some stability island obtained by computational models due to the effects of adopting the standard law for friction, as implemented in systems codes. It must be remembered that an unstable flow with reversals traverses the zone of transition from laminar to turbulent flow. Fortunately in this case, the experimental map is known and adopting a smooth transition for friction (from Poiseuille to Blasius friction laws) as suggested by the experimenters avoids the problem. It may be difficult to discern what law must be used in an industrial case, when dealing with unstable, cyclical flow oscillations and non fully-developed flow in relatively short pipes with bends and large flow area transitions. In a natural circulation flow, the driving forces originate in fluid buoyancy. Then, it is tempting to artificially diminishing by a known factor (simulating an error in a correlation) the heat input to the system. Again, using 90 nodes and multiplying the driving force by 0.6, the neutrally stable oscillations are obtained. Since flow rate variation comes from the balance between buoyancy and friction increasing the friction 1.43 times, the buoyancy by 1.66 and keeping 66 nodes nearly preserves this artificially neutrally stable condition. This simple, stereotyped example shows how errors may compensate.

In dealing with natural circulation in 1D loops, some problems may be found when the flow tends to stagnate. This case has been discussed in [11]. A square loop formed by tubes of 0.027 m of internal diameter and 2.2 m high and 1.4 wide, was heated with 100 W at its bottom horizontal pipe and this power was removed by a horizontal heat exchanger in its top horizontal pipe. The flow is unstable in almost all the range of power investigated ( >105 - 600 W). Only for a heating power less than 105 W the flow was stable and one directional. This case was investigated using three codes, two 1D (TRANLOOP [9] and RELAP5\textsuperscript{3} [1] and a CFD code (FLUENT 6.2 [12]). Both the 1D codes for 100 W arrived to a point of flow stagnation after a period of flow reversals that remained up to nearly 3600 s. The CFD code predicted the oscillation behavior in a correct way. The stopping of the flow with the 1D codes was due to the appearance of 1 hot fluid pocket at the top lasting for ever in the case.

\textsuperscript{3} The use of this particular systems code is due to its availability to the present Author and his colleagues. The same peculiarities of the solutions may also be found using other codes.
of TRANLOOP because of the assumption of the Boussinesq approximation for the fluid. In the case of RELAP5, the situation persisted up to the point in which the temperature of the fluid in the heater zone reached 273 K and, due to boiling, minute perturbations in the fluid allowed flow bursting, fluid cooling and a new period of stagnation. Then, the lessons learned were: a) 1D codes may fail to predict flow instability due to inherent limitations in the physical model or in the dimensionality; b) a detail code (like a CFD one) may be important to discern whether a model (the Boussinesq approximation) may still be useful due to dimensionality problem avoidance and c) these conclusions can not be generalized to any situation.

3. Two phase unstable flows in heated channels

Two-phase unstable flows may be studied to test the prediction capabilities of a code in order to perform V&V evaluations in the range that covers normal operating conditions in a nuclear installation or to reproduce the behavior of some experimental loop in controlled laboratory conditions, once again aiming to validate the computational predictions. The posedness of the governing equations for two-phase flows has been a subject of discussion. As mentioned before, most system codes solve an ill-posed system of PDEs. However it can not asserted that predictions are wrong, because there is always a method that allows circumventing the problem that consists in the introduction of numerical viscosity, like Von Neumann and Richtmyer did in 1950, or explicit physical viscosity in the equations. Up-winding is the usual technique. RELAP5 includes a term not affecting the steady-state solution. It is accepted that the least contribution of these terms regularize the system. Also, there are numerical ways to deal with ill-posedness, e.g. matrix pre-conditioning affecting only the time dependent terms. In this way, the steady-state solution is not influenced. This is important too when using pseudo-transients to get steady-state solutions.

3.1. A theoretical boiling channel simulation

In [13], the results of a computational study on boiling channel stability have been discussed. The study compared the information obtained by two programs, developed in previous work for the linear and the nonlinear stability analysis of boiling channels based on a simplified flow model, with the predictions of RELAP5. The phenomena, as described by the results of the latter, adopted with different flow models and numerical methods, have been discussed in a systematic way with the aid of the description obtained by the simplified code which, although limited in generality, permits a valid analysis in the case considered. The effects on the results obtained with RELAP5 were analyzed using different options like equilibrium and non-equilibrium fluid mixture, different numerical discretization schemes and variable number of nodes, evaluating their influence on the predicted stability boundaries.

In what follows, the results and conclusions that were arrived in [13]will be considered briefly. The adopted channel geometry and working conditions were as follows: channel diameter 0.0124 m; channel length 3.66 m (i.e., 12 ft.); inlet plenum pressure 7.0 MPa; pipe roughness 2.5×10^{-5} m; input and output concentrated loss coefficients Ki = 23 Ke =5; pressure drop between input and output was 0.11 MPa (imposed) and variable number of nodes (12, 24 and 48). Concerning the working fluid (water and its vapour), the homogeneous equilibrium model (HEM, in the following also named Equal Velocities Equal Temperatures, EVET) and the non-homogeneous non-equilibrium model (Unequal Velocities Unequal Temperatures, UVUT) were activated by choosing the related options in the “fcvcahs” and the “tlpvbfe” codes. Regarding the numerical approaches, both the “semi-implicit” and “nearly-implicit” numerical methods were selected by choosing the related flags in the time step advancement cards.

The detailed discussion on the physics and computational results in [13], can not be considered here for the sake of brevity. The following conclusions may, anyway, be mentioned: a) the semi-implicit numerical method is considerably less prone to calculation problems than the nearly-implicit one in the application to boiling channel instability studies, showing a known code limitation; b) the
latter method, as expected, introduces greater numerical diffusion, though this is evident only at low subcooling number, while at larger subcoolings the behaviour is less clear; c) the UVUT model is more “robust” than the EVET model and d) the ratio of the fluid transit time to the oscillation period at the stability boundary can be considerably different from 0.5, especially at high subcooling.

3.2. Simulation of a boiling channel in a closed loop

The computational simulation of the experimental results of [14] will be considered in the following. The code used was RELAP5 and a more detailed explanation will be included, since these results have not been published elsewhere. Figure 1 shows a schematic diagram of the two-phase forced circulation loop. It consists of a heated test section with a large bypass, a riser, an open tank and a downcomer. The fluid circulates by means of a centrifugal pump. Most of the flow is derived to the bypass, in order to maintain a constant pressure drop across the test section. In this aspect, the behavior of the heated channel would be similar to the previous example. The downcomer, the upper and lower horizontal major sections, the bypass and the riser are made of 50-mm inner diameter stainless steel pipes. The plena are cylinders 150 mm I.D. and 100 mm in height. Both plena and its corresponding exit tubes (1-inch i.d.) are made of brass. The valve at the inlet of the test section is 25 mm (1 inch nominal) I.D and the valve at the riser is 51 mm (2 inch nominal) I.D. The test section consists of an annular heated channel. It is connected to system through 25 mm I.D. tubing. The heated channel is a 29 mm I.D. transparent Pyrex glass tube with a coaxial electrical heater (9.5-mm O.D., 1000-mm length). A D.C. power supply unit provides the electrical power.

The experiment used water as working fluid, and the system was subject atmospheric pressure. The fluid temperature at the inlet and exit of the heated channel was measured with thermocouples. The temperature in the inlet plenum was also recorded. The flow rate in the channel was measured at the inlet of the heated section by means of an orifice plate and a fast response differential pressure transducer, variable reluctance type. The pressure drop and temperature signals were acquired by computer means. The sample interval was 0.25 seconds. The position of the valves was fixed in all experimental runs. The total loss coefficients upstream and downstream of the test section were 18 ± 5 and 8 ± 4, respectively. In this paper, the results corresponding to gravity driven oscillations will be considered, so the valves were completely open.

The experiments in [14] were carried out maintaining constant power (3400±100 W). Starting with high inlet subcooling (T_i = 68 °C) the inlet temperature was slowly increased (at approximately 0.1 °C/min.). At a certain point the stability margin is reached and the flow begins to oscillate. Further increasing the inlet temperature produced the transient excursions of the boiling boundary out of the channel, leaving the channel completely in liquid phase. Afterwards, the inlet temperature was forced to decrease until the system returned to stable conditions. Figure 2, taken from [14] shows the records of the flow measurements at different inlet temperatures. In this figure the oscillations start at T_i = 72.8 °C and it may be observed that the instabilities continue even if the temperature decreases below that value. The vertical bars are the standard deviations of the flow signal, and can be taken as indicators of the oscillation amplitude.

This experiment has been simulated using RELAP5 and some results will be shown in what follows. The heated channel was discretized using 52 nodes. Coherently with previous experience this would be an appropriate one. This was simply verified by increasing the number of nodes without success. Figure 3 shows the results obtained heating the system after getting steady state with the power turned off. The heating ramp, the plateau and the cooling down ramp allowed arriving to T_i = 80.2 °C. As may be observed, at this temperature the flow underwent constant amplitude oscillations. The code options were the ones corresponding to default conditions. The period of the oscillations was 18 s, the same that the period in the experiment and also coincident with the fluid transit time. These aspects are illustrated in figure 3 that shows these satisfactory results. Also, the expected hysteresis was reasonably represented. Most unfortunately, explicit activation of the UVUT or the EVET options led to non satisfactory results in the detection of the threshold temperature for the beginning of the
flow oscillations and their amplitude. The causes for this behavior remain in the code and are uncertain.

Loop components

| Component | Description |
|-----------|-------------|
| i         | Upper tank open to atmosphere |
| ii        | Downcomer |
| iii       | Main pump |
| iv        | Lateral bypass with a valve |
| v         | Main input valve at pump exit |
| vi        | Main output valve upstream of tank |
| vii       | Lower and upper plena with connections to the heated channel |
| viii      | Test section input valve |
| ix        | Heated test channel |
| x         | Test section output valve |
| xi        | Secondary cooling system with a heat exchanger in the upper tank |

Dimensions in cm

**Figure 1.** Schematic loop representation and nomenclature

**Figure 2.** Volumetric flow rate in the heated channel vs input fluid temperature and oscillations ranges for gravity driven oscillations. Experimental data from [14]
3.3. **Implications of the results shown on future research and development relevant to codes V&V**

In the preceding subsections some limitations of present day’s systems codes have been exemplified. Some of them are due to the so-called user effects. Many other are not so evident and are at the root of the need for uncertainty evaluation of computational results. The present trend, given the huge work done in the past on V&V of systems codes, seems turning to V&V of CFD codes. The international community is aware of the lack of detailed and reliable experimental information on two-phase flows in the vicinity of walls and is also aware of the need of advanced measurement techniques that permits obtaining said data. Complex flows (like two-phase flows, heterogeneous media flow like debris flows, localized heat transfer effects models, etc.) are progressively incorporated as CFD code capabilities. These type of codes (FLUENT, STAR-CD, ANSYS-CFX and many others) are typically considered as capable of giving support to safety evaluation of nuclear installations. As a consequence, V&V of CFD codes used in the nuclear industry constitutes nowadays a vast field for people dealing with the main topics treated here and need to be validated. A very valuable source of information on the needs and the current status of the activities may be found in the Nuclear Energy Agency site, namely at: [http://www.nea.fr/html/nsd/csn/cfd/index.html](http://www.nea.fr/html/nsd/csn/cfd/index.html)

![Figure 3.](image)

(a) Comparison of predicted and experimental two-phase oscillating flow in the heated channel of [14] showing coincidence of experimental and predicted oscillation period.

a) Time variation of the computed flow rate and fluid inlet temperature

b) Zoom of the flow rate variation in figure 3a. Oscillation period is 18 s

c) Time variation of experimental [14] volumetric flow rate variation. Oscillation period is 18 s.
4. Conclusions and recommendations

In the previous sections, part of the experience gained along fifteen years of use of system codes and from the development of computational codes to deal with natural circulation flows in loops has been reviewed. Additionally, the simulation of two cases of forced two-phase flow has been presented. The one corresponding to an experiment showed good agreement, although the code used suffered from some limitations. In this case, user effects may also affect the accuracy of the predictions and can not be dismissed.

On the basis of Author’s experience, there is a unique opportunity for researchers considering the physics and measurements of fluid flows in complex geometries and for the development of closure laws with a degree of detail compatible present technological demands. In the next decade, at least in the nuclear field, closure laws in two-phase bubbling flows considering multi-field approximations will be needed. This implies enhancing present measurement techniques near to walls to consider non-macroscopic space scales and the simultaneous consideration of non-isothermal flow regimes. Simultaneously, 1D system codes will be applied supported by coupled CFD detailed calculations adding complexity to nuclear safety evaluations. Then, the need for research in the physics of fluids in complex flows will be a challenge to face in the near future.

References

[1] Carlson K E et al., RELAP5/MOD3 Code Manual Volume I: Code Structure, System Models and Solution Methods, 1990 US NRC NUREG/CR-5535

[2] Ferreri J C 1998 IAEA Specialists Meeting on User Qualification Effects for the Safety Analysis of Nuclear Power Plants, (Vienna, Austria, Aug 30th –Sept 3rd 1998)

[3] Bucalossi A, Petruzzi A, Kristof M and D’Auria F 2010 Nuclear Technology 172 29-47

[4] Welander P, On the oscillatory instability of a differentially heated fluid loop 1967 J. Fluid Mech. 29 17-30,

[5] Ambrosini W and Ferreri J C 1998 Nuclear Eng. and Design 183 53-76

[6] Ambrosini W and Ferreri J C 2000 Nuclear Eng. and Design 201 11-23

[7] Ferreri J C and Ambrosini W 2002 Nuclear Eng. and Design 215 153-170

[8] Ferreri J C and Ambrosini W 2001 Int. J. Computational Fluid Dynamics 16, 277-281

[9] Ambrosini W and Ferreri J C 2003 Annals of Nuclear Energy 30 1505-37

[10] Ambrosini W, Forgione N, Ferreri J C and Bucci M 2004 Annals of Nuclear Energy 31 1833-65

[11] Pilkhwal D S, Ambrosini W, Forgione N, Vijayan P K, Saha D and Ferreri J C 2007 Annals of Nuclear Energy 34 339-355

[12] FLUENT Inc., FLUENT 6.2 User’s Guide 2003 Centerra Resource Park, Lebanon, NH USA

[13] Ambrosini W and Ferreri J C 2006 Proceedings of ICONE 14, 14th International Conference on Nuclear Engineering (Miami, Florida USA July 17-20 2006) ICONE PAPER 14-89863

[14] Juanico L, Identification of dynamic basins in boiling flows, 1997 PhD Thesis Inst. Balseiro UNCuyo Argentina