System Code Models and Capabilities

Dominique Bestion

CEA-Grenoble, 17 rue des Martyrs, 38054 GRENOBLE CEDEX, France

Abstract. System thermalhydraulic codes such as RELAP, TRACE, CATHARE or ATHLET are now commonly used for reactor transient simulations. The whole methodology of code development is described including the derivation of the system of equations, the analysis of experimental data to obtain closure relation and the validation process. The characteristics of the models are briefly presented starting with the basic assumptions, the system of equations and the derivation of closure relationships. An extensive work was devoted during the last three decades to the improvement and validation of these models, which resulted in some homogenisation of the different codes although separately developed. The so called two-fluid model is the common basis of these codes and it is shown how it can describe both thermal and mechanical non-equilibrium. A review of some important physical models allows to illustrate the main capabilities and limitations of system codes. Attention is drawn on the role of flow regime maps, on the various methods for developing closure laws, on the role of interfacial area and turbulence on interfacial and wall transfers. More details are given for interfacial friction laws and their relation with drift flux models. Prediction of choked flow and CFFL is also addressed. Based on some limitations of the present generation of codes, perspectives for future are drawn.

1. INTRODUCTION

If best-estimate system codes play a key role in nuclear technology, it is mainly due to the impracticality of executing full-scale safety related experiments and to the absence of simplified scaling laws for the governing processes which would allow a direct transfer of results from small scale test facilities to the full size plant.

Early attempts in reactor safety analysis based on an “evaluation model” approach including the definition of a limited number of worst case scenarios in combination with conservative modeling assumptions have been replaced by “best-estimate” methodologies. The best-estimate approach aims at providing a detailed realistic description of postulated accident scenarios based on best-available modeling methodologies and numerical solution strategies sufficiently verified against experimental data from differently scaled separate effect test and integral effect test facilities. Best-estimate system codes are now commonly used for reactor thermalhydraulic studies. The main system codes which have reached a high quality of predictions with an extensive validation are the US-NRC codes RELAP5 and TRAC, followed by the TRACE code, the French code CATHARE-2 (sponsored and developed by CEA, EDF, FRAMATOME-ANP and IRSN) and the German code ATHLET developed by GRS. TRACE V5.0 was released in 2007; it is now the validated code of US-NRC based on RELAP and TRAC. NEPTUNE is a new multi-scale software platform developed by EDF and CEA which includes a CFD module and CATHARE-3 as system code; it is not considered here since it has not yet the same level of validation.

The thermalhydraulic models used in these system codes will be presented and the capabilities and limitations of system codes will be summarized. The author has been developing the CATHARE code and has a better knowledge of this code than of the others. However, information from the documentation of RELAP and TRAC codes are used, and examples are given of all these codes. A more limited information about the ATHLET code was available when writing this paper. As for the evaluation of capabilities and limitations of system codes, the results of the EUROFASTNET [1,2] European project reflect a consensus of many experts of the scientific community and they are extensively reported here. Proceedings of OECD/CSNI Workshops on Thermalhydraulic codes in Aix en Provence [3], Annapolis [4], and Barcelona [5] were also used in this paper.
2. THE SYSTEM CODES AND THEIR MAIN CHARACTERISTICS

2.1 The RELAP5 Code

RELAP5 is the advanced version of the RELAP (Reactor Excursion and Leak Analysis Program) code developed at the Idaho National Engineering Laboratory. The most recent version, RELAP5/Mod3 [6] (see RELAP5/Mod3 Code Manual, 1995) includes some contributions and improvements from the member countries of the joint International Code Assessment Programme (ICAP) and its successor, the Code Application and Maintenance Programme (CAMP).

The description of the thermal-hydraulics is based on a two-fluid model for two-phase flow with a flow-regime based modelling of interfacial transport processes for mass, momentum and energy. In the US-NRC version, only a 1D model is available with possible cross-flow junctions to allow some description of multi-D flows in porous bodies (core, horizontal SG,…).

2.2 The TRAC and TRACE codes

The Transient Reactor Analysis Code (TRAC), developed at the Los Alamos National Laboratory is an advanced, best-estimate computer program for the numerical simulation of postulated accident and related transient behaviour in Pressurized Water Reactors (PWR). A first preliminary version (TRAC-P1) was publicly released in 1977. Since then the code has been continuously improved (TRAC-PF1, TRAC-PF1/MOD1, TRAC-Pf1/Mod2) in order to extend the code robustness and range of applicability. A BWR version of the code was developed as a side branch at the Idaho National Engineering Laboratory following the same modelling and numerical approach as its PWR counterpart. The most recent (“modernized”) version TRACE [7] (2007) is written in FORTRAN90 and combines both PWR and BWR predictive capabilities. It is supposed to replace RELAP and TRAC codes for all light water reactors at the US-NRC. The first validated version TRACE V5.0 was released in 2007.

The basic modelling approach for transient two-phase flow is a two-fluid model with flow regime dependent correlations for the interfacial heat, momentum and energy transfer processes.

2.3 The CATHARE Code

The Code for Analysis of Thermalhydraulics during Accident of Reactor and Safety Evaluation is the French counterpart to the RELAP5 or TRAC code. The code has been developed by three partners, the French Atomic Energy Commission with the safety institute CEA & IPSN (now IRSN), the French utility EDF and the vendor FRAMATOME (now FRAMATOME-ANP) for the best estimate analysis of postulated accidents in pressurized water reactors.

As in most best-estimate codes, the description of thermal non-equilibrium inhomogeneous two-phase flow is based on a two-fluid approach using mainly algebraic constitutive relations for the modelling of interfacial coupling, wall friction and wall heat transfer processes. An interfacial pressure difference term has been introduced, which results in an unconditionally hyperbolic system of equations. In addition to the basic one-dimensional pipe component, more simplified lumped parameter models are used for tees, branches and capacities which are needed to build complex thermal-hydraulic networks. The code is based on a 2-fluid 6-equation model. In the last released version CATHARE 2 V2.5 [8], a validated 3D pressure vessel modelling is available. The code is able to model any kind of experimental facility or PWR (Western type and WWER), and is usable for other reactors ( Fusion reactor, RBMK reactors, BWRs, research reactors).

2.4 The ATHLET Code

The thermal-hydraulic computer code ATHLET (Analysis of Thermal-hydraulics of LEaks and Transients) is being developed by the Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) for the safety analysis of light water reactors [9] The code can model the whole spectrum of design basis and beyond- design basis accidents (without core degradation) like anticipated and abnormal plant transients and small to large break loss of coolant accidents in PWRs and BWRs. For the fluid dynamics the code user can choose between three options having a different degree of sophistication:
1. The basic option is a 5-equation drift flux model with separate balance equations for mass and energy and one mixture momentum equation; mechanical non-equilibrium is calculated by a full-range of drift flux correlations for horizontal and vertical flow conditions.

2. A two-fluid model with separate balance equations for mass, momentum and energy with corresponding algebraic source terms for the description of interfacial transport processes.

3. A 4-equation homogeneous thermal non-equilibrium model including separate mass balance equations and mixture equations for momentum and energy.

2.5 Objectives of the Present Generation of System Codes

The development of the present generation of system codes was initiated in the years 1970 to 1980. The main objective of developing these codes was to replace Evaluation Models which used many conservative assumptions by the best-estimate approach for more realistic predictions of PWR or BWR accidental transients. Such codes may have a wide range of applications from research to safety and design purposes. Examples of applications are:

- Safety analysis
- Quantification of the conservative analyses margin
- Investigation of Plant Operating Procedures and Accident Management
- Definition and verification of Emergency Operating Procedures
- Investigations for new types of fuel management
- Preparation and interpretation of experimental programs
- Licensing when used together with a methodology to evaluate uncertainties on the code predictions.
- Design of new reactors and systems including passive features for the 3rd and 4th generations of NPP
- System code application has been often extended to the field of severe accidents. For this purpose they are coupled with other codes, which model core degradation and fission product release.
- Implementation in Full Scope Plant Simulator: simplified versions were first used to obtain real time simulations and later, thanks to the increasing computer performance, the standard versions were used.

2.6 Modelling Reactors

System codes have a modular structure. Several modules can be assembled to represent the primary and secondary circuits of any Reactor and of any Separate-Effect Test or Integral Effect Test facility. The main modules are:

- A 1D module to describe pipe flow
- A 1D module with tees may be used to represent a main pipe (1D module) with a lateral branch (tee-branch). The T module predicts phase separation phenomena, and a specific modelling effort has been paid for cases where the flow is stratified in the main pipe.
- Other modules (PLENA, CAPACITY) are available to describe large size plena, such as the pressurizer, the accumulator, the steam generator dome or the lower plenum of a PWR, or to model components with several connections.
- TRAC and CATHARE have a 3D module to describe large scale multidimensional effects in the vessel.

An example of nodalization of primary and secondary circuits of the LOFT test facility (with the RELAP code) is given in Figure 1.

Apart from the basic thermalhydraulic modules other systems are modelled through “sub-modules”:

- Metallic structures with heat conduction calculations (often 1D or 2D for quench front)
- Heat Exchangers for exchanges between circuits (Steam Generators) or between components (core-downcomer)
- Fuel thermo-mechanical modules
- Neutron kinetics: simple Point-kinetics is generally embedded in the code and coupling with 3D kinetics may be available
- Valves, Safety Valves, Control valves or Check valves
- Breaks, Fill (source), Sinks
- Pumps with a 0D model including two-phase characteristic functions
- Specific modelling for Separators and Dryers
- Boundary Conditions
2.7 The Main Challenges of System Codes

Building such system codes was very challenging with respect to a number of difficulties:

- Geometrical complexity of the systems: the geometry of the flows within reactor circuits is extremely complex and has to be drastically simplified in order to allow a coarse nodalization while keeping reliable predictions of macroscopic parameters.
- Variety of two-phase flows: The very wide scope of transients to simulate with system codes covers all types of two-phase flow regimes and all regimes of heat transfers with heating or cooling walls.
- Wide range of physical parameters ($0.1<P<20\text{MPa}$, $T<1200^\circ\text{C}$, Velocity up to sonic).
- CPU time must remain compatible with an industrial use: all type of accidental transients including Large Break LOCAS must be simulated in a reasonable time (e.g. 12 hours) allowing many sensitivity tests to be performed in addition to a base case calculation.
- Selecting phenomena which deserve a specific modelling effort: since the variety of flow regimes, heat transfer regimes, geometrical configuration is extremely wide, most important phenomena for safety issues have to be identified and a specific effort has to be paid for an appropriate modelling of these sensitive phenomena.
- Selecting the right level of complexity of the models: the level of complexity of the model must remain compatible with both the available scientific knowledge and experimental data and the required reliability of predictions for safety.

3 THE TWO-FLUID MODEL AS A BASIS OF THE PHYSICAL MODELLING

Nearly all current two-phase flow models used in present “best estimate” thermal hydraulic system codes are based on the “two-fluid model”. Phases are treated as interpenetrating continua and “macroscopic” separate balance equations for each phase are obtained by a space and/or time or ensemble averaging of the local instantaneous basic flow equations, with source terms representing the interfacial transfers for mass, momentum and energy. Due to the averaging, information on local flow processes, in particular at the interface separating the two phases or at the region near the walls, is lost and has to be compensated by additional modeling.
3.1 The Averaging Procedure and the System of Equations

In the two fluid, mass momentum and energy balance equations are written for each phase. These equations can be derived from exact local-intantaneous equations. The process includes several steps: space and time averaging, simplifications through physical assumptions, derivation of closure relations. Models are restricted to zero-order closure so that no more partial differential equations are derived.

The averaging process which restricts predictions to macroscopic phenomena is necessary to allow reasonably coarse meshing and to make comparison with experiment easier. The time integration or averaging suppresses from calculated quantities fluctuations due to the turbulent nature of the flows. The space averaging is also very helpful in two phase flows as it allows forgetting the complex structure of phase repartition and interface movements. The effects of small scale processes on macroscopic evolution can be taken into account by appropriate closure relation. However the time and space scales at which fluctuations can be filtered is not always evident. It is implicitly assumed that the spectrum is cut into two separate zones, the range of small scale processes and the macroscopic range.

As a result of the spatial/time averaging of the instanteous phase equations, the “macroscopic” balance equations of the two-fluid model can be formulated for vapour \(i=g\) and liquid \(i=l\)as follows:

mass:

\[
\frac{\partial}{\partial t}(\alpha_i \rho_i) + \nabla \cdot (\alpha_i \rho_i \vec{v}_i) = \Gamma_i
\]

with \(\sum_{i=g,l} \Gamma_i = 0\) \[(1)\]

momentum

\[
\frac{\partial}{\partial t} \left[ \alpha_i \rho_i \left( \vec{u}_i + \frac{1}{2} \vec{v}_i^2 \right) \right] + \nabla \cdot \left[ \alpha_i \rho_i \vec{v}_i \left( h_i + \frac{1}{2} \vec{v}_i^2 \right) \right] + p \frac{\partial \alpha_i}{\partial \tau} - \nabla \cdot \left( \alpha_i \vec{T}_i \cdot \vec{v}_i \right) = \vec{F}_i^{\text{int}} + \Gamma_i \vec{v}_i^{\text{int}} + \vec{F}_i^{\text{ext}}
\]

with \(\sum_{i=g,l} \vec{F}_i^{\text{int}} = 0\) \[(2)\]

energy:

\[
\frac{\partial}{\partial t} \left[ \alpha_i \rho_i \left(h_i + \frac{1}{2} \vec{v}_i^2 \right) \right] + \nabla \cdot \left[ \alpha_i \rho_i \vec{v}_i \left( h_i + \frac{1}{2} \vec{v}_i^2 \right) \right] + p \frac{\partial \alpha_i}{\partial \tau} - \nabla \cdot \left( \alpha_i \vec{T}_i \cdot \vec{v}_i \right) = \vec{Q}_i^{\text{int}} + \frac{\sum_{i=g,l} \vec{Q}_i}{\Gamma_i} \left[ h_i + \frac{1}{2} \vec{v}_i^2 \right]^{\text{int}} + \vec{F}_i^{\text{int}} \cdot \vec{v}_i^{\text{int}} + \vec{F}_i^{\text{ext}} \cdot \vec{v}_i + \vec{Q}_i^{\text{ext}}
\]

with \(\sum_{i=g,l} \vec{Q}_i + \frac{\sum_{i=g,l} \vec{Q}_i}{\Gamma_i} \left[ h_i + \frac{1}{2} \vec{v}_i^2 \right]^{\text{int}} = 0\) \[(3)\]

3.2 Single-Pressure Model

One of the problems in solving the system of equations (1) to (3) is that they contain more major governing parameters than equations. This problem may be avoided assuming simple relations between phase averaged pressures and interfacial averaged pressure such as: \(p_g = p_l = p\) and the general balance equations of the two-fluid model can be further simplified

mass:

\[
\frac{\partial}{\partial t}(\alpha_i \rho_i) + \nabla \cdot (\alpha_i \rho_i \vec{v}_i) = \Gamma_i
\]

\[(4)\]

momentum:

\[
\frac{\partial}{\partial t} \left[ \alpha_i \rho_i \vec{v}_i \right] + \nabla \cdot (\alpha_i \rho_i \vec{v}_i \vec{v}_i) + \alpha_i \nabla p + \left( p^{\text{int}} - p \right) \nabla \alpha_i - \nabla \cdot \left( \alpha_i \vec{T}_i \right) = \vec{F}_i^{\text{int}} + \Gamma_i \vec{v}_i^{\text{int}} + \vec{F}_i^{\text{ext}}
\]

\[(5)\]
Although a single (local) pressure is assumed for both phases in the mass and energy balance equations, a pressure difference between the bulk of phase $i$ and the interface is considered in some of the codes, as indicated in the two momentum equations (5). In the CATHARE code for example, this pressure difference has been modeled as:

$$\Delta p^{\text{int}} = p - p^{\text{int}} = \frac{\alpha_i \rho_i \rho g + (1 - \alpha_i) \rho_l g}{\alpha_i \rho_i + (1 - \alpha_i) \rho_l} |v^e - v^l|^2$$

This term ensures the hyperbolicity of the system of equations. Another expression of this pressure difference can be derived in case of a stratified flow assuming a hydrostatic transverse pressure gradient. In this case $p = \alpha p^g + (1 - \alpha) p^l$.

### 3.3 One-Dimensional Flow

The separated balance equations can be written for quasi one-dimensional flow in pipes with varying cross section $A(x)$ as:

**mass:**

$$\frac{\partial}{\partial t} (\alpha_i \rho_i) + \frac{1}{A} \frac{\partial}{\partial x} (\alpha_i \rho_i v_i A) = \Gamma_i$$

**momentum:**

$$\frac{\partial}{\partial t} (\alpha_i \rho_i v_i) + \frac{1}{A} \frac{\partial}{\partial x} (\alpha_i \rho_i v_i^2 A) + \alpha_i \frac{\partial}{\partial x} \left( p^{\text{int}} \frac{\partial \alpha_i}{\partial x} \right) = F_i^{\text{ext}} + F_i^{\text{vm}} + \Gamma_i (v^l - v^i) + F_i^{\text{wal}}$$

**energy:**

$$\frac{\partial}{\partial t} \left[ \alpha_i \rho_i \left( u_i + \frac{1}{2} v_i^2 \right) \right] + \frac{1}{A} \frac{\partial}{\partial x} \left[ \alpha_i \rho_i v_i \left( h_i + \frac{1}{2} v_i^2 \right) A \right] + p \frac{\partial \alpha_i}{\partial t} = Q_i^{\text{int}} + Q_i^{\text{wall}} + Q_i^{\text{ext}}$$

Due to the pressure and void fraction coupling terms, the separated momentum and energy equations cannot be put into a fully conservative form which might have some consequences for their numerical integration. In present TH-codes the balance equations are often used in an expanded form or (e.g. in RELAP5) as the sum and the difference of the phasic balance equations in order to facilitate a transfer between single phase liquid or vapour and two-phase conditions.

The source terms on the right-hand side of the balance equations (8) to (10) include three different categories:

- For external body forces normally only gravity is of interest with $F_i^{\text{ext}} = \alpha_i \rho_i g$. As external energy sources $Q_i^{\text{ext}}$ the instantaneous fission power and the power related to the decay of fission products have to be considered which are calculated by neutron kinetics models.
- Non-viscous interfacial forces $F_i^{\text{vm}}$ which have been introduced in order to compensate for the loss of information on local flow processes resulting from the averaging. It is expected that these forces are exclusively related to time and space derivatives with regard to flow parameters, and as such directly affect the characteristics (eigenспектrum) of the system of governing equations.
- Sources resulting from the interfacial mass, momentum and energy exchange processes, $\Gamma_i$, $F_i^{\text{int}}$, $Q_i^{\text{int}}$. It is assumed that these dissipative terms can be described exclusively by algebraic functions of the governing flow parameters.
- Momentum and energy coupling with solid walls or structures including wall friction $F_i^{\text{wall}}$ and wall heat transfer $Q_i^{\text{wall}}$. 

86
The virtual mass forces result from the coupling with the surrounded fluid in case of the acceleration of a particle (bubble or droplet). Apart from ideal flow conditions (e.g. potential flow around spheres), there is no way to derive more general formulations for the virtual mass effects from basic principles. Several slightly different forms of this term can be found in the literature which are often not free of some heuristic assumptions. A rather general formulation has been derived by Drew et al. [10] based on the objectivity principle (independence on coordinate system transformation) which can be written as:

$$\ddot{E}^{\text{vm}} = \rho_c C^{\text{vm}} \left[ \frac{\partial \dot{v}_g}{\partial t} + v_1 \cdot \nabla \cdot \dot{v}_g \right] - \left( \frac{\partial \dot{v}_1}{\partial t} + \dot{v}_g \nabla \cdot \dot{v}_1 \right) - \left( (1 - \lambda) \left( \dot{v}_g - \dot{v}_1 \right) \right) \nabla \left( \dot{v}_g - \dot{v}_1 \right)$$

(11)

In the recent version of RELAP5, the expression for the virtual mass term has been simplified to

$$F^{\text{vm}} = C^{\text{vm}} a_g a_f \rho_m \left[ \frac{\partial (v_g - v_1)}{\partial t} \right]$$

(12)

In the CATHARE code a different formulation for the virtual mass forces is used which can be related to the objective form for \( \lambda = 2 \) as

$$F^{\text{vm}} = C^{\text{vm}} a_g a_f \rho_m \left[ \frac{\partial (v_g - v_1)}{\partial t} + v_g \frac{\partial v_g}{\partial x} - v_1 \frac{\partial v_1}{\partial x} \right]$$

(13)

The virtual mass coefficient \( C^{\text{vm}} \) is normally bounded between 0 and 1. In some cases a flow regime dependent value is used, e.g. \( C^{\text{vm}} = 0.5 \) for dispersed bubbly or droplet flow or \( C^{\text{vm}} = 0.0 \) for fully separated phases such as stratified flow conditions. Although some authors report a reduction in the possible numerical oscillations or instabilities, the effect of virtual mass terms is often relatively small or largely overwhelmed by high artificial viscosity inherent in the numerical technique applied. Nevertheless, the virtual mass forces have a strong effect on the sound velocity in two-phase media. Often specific formulations for the virtual mass forces are applied in order to ensure hyperbolicity of the system of governing equations.

### 3.4 From Two-Fluid to Drift Flux Model

In some codes (e.g. ATHLET) a simplified form of the two-fluid model is applied based on just one mixture momentum equation which can be obtained by adding the two phasic momentum equations (9).

$$\frac{\partial}{\partial t} (\rho v) + \frac{1}{A} \frac{\partial}{\partial x} \left( \rho v^2 A \right) + \frac{1}{A} \frac{\partial}{\partial x} \left( \frac{a_g a_f \rho_g p_1}{\rho} \Delta v \right) + \frac{\partial \rho}{\partial x} = F^{\text{ext}} + F^{\text{int}}$$

(14)

with the mixture density \( \rho = a_g \rho_g + a_f \rho_f \), and the weighted flow velocity \( v = \frac{a_g \rho_g v_g + a_f \rho_f v_f}{a_g \rho_g + a_f \rho_f} \).

The relative velocity (slip) \( \Delta v = v_g - v_f \) is calculated from empirical “drift flux” correlations. From the relation:

$$v_g = Co j + v_{g_i} = Co \left( j_g + j_i \right) + v_{g_i} = Co \left( a_g v_g + a_i v_i \right) + v_{g_i}$$

(15)

one can express the velocity difference provided that models are available for the distribution parameter \( C_o \) and the drift velocity \( v_{g_i} \).

Momentum equations (9) can be combined to eliminate the pressure gradient, giving in the following equation:

$$\left[ \begin{array}{c} a_i \frac{\partial}{\partial t} \rho_i v_i \\ a_g \frac{\partial}{\partial t} \rho_g v_g \\ \frac{\partial}{\partial x} \end{array} \right] = -\Delta p^{\text{int}} \frac{\partial}{\partial x} a_i + F^{\text{vm}}$$

$$= \Gamma (v^{\text{int}} - a_g v_g - a_f v_f) - F^{\text{int}} - \left[ a_g F_{g, \text{null}}^{\text{int}} - a_f F_{f, \text{null}}^{\text{int}} \right] + a_g a_f (\rho_i - \rho_g) g_x$$

(16)
which can be simplified in case of steady established flow without phase change:

\[
F_{\text{int}} = -\left[ a_g F_{g}^{\text{wall}} - a_f F_{f}^{\text{wall}} \right] + a_g a_f (\rho_f - \rho_g) g x
\]  

(17)

This equation expresses the equilibrium between buoyancy and drag forces B and D when wall friction is negligible. This equilibrium for a bubble of size \( \delta \) can be written as:

\[
B \approx \left( \rho_1 - \rho_g \right) \delta^3 g x ; \quad D \approx \delta^2 C_d \rho_1 \frac{v_{gi}^2}{2} \]

(18)

Eliminating the buoyancy force between eq. (17) and (18) one can write:

\[
F_{\text{int}} = a_g a_f (\rho_f - \rho_g) g x = a_g a_f (\rho_1) \frac{v_{gi}^2}{f^2 \delta} = a_g a_f \left( 1 - \frac{\alpha}{\alpha_g} \right) \rho_1 \frac{(v_g - C_k v_1)^2}{f^2 \delta}
\]

(19)

This shows how a drift flux model can be translated into an interfacial friction model. It also shows how and when a mixture momentum equation with a drift flux model is equivalent to two momentum equations. Drag and buoyancy forces must be much larger than inertial forces, virtual mass forces, wall friction, and mass change effect.

In practice both models are equivalent for quasi-steady and quasi-established bubbly–slug–churn flows in vertical ducts or in horizontal pipes without strong phase change (core flow with \( 0 < \alpha < 0.7 \))

Two momentum equations have better capabilities than the drift flux algebraic equation for:

- Annular flow or stratified flow since wall friction force is not much larger than wall friction
- Droplet flow since inertial force is not negligible compared to interfacial friction
- Flashing flows or highly condensing flows since phase change effects and acceleration are not negligible
- Accelerating flows.

3.5 Characteristic Velocities and Hyperbolicity

The system of equations such as eq. (8) to (10) may be written in the following form:

\[
A \frac{\partial U}{\partial t} + B \frac{\partial U}{\partial Z} = C
\]  

(20)

When the system is hyperbolic [11] six real characteristic velocities are found by solving the characteristic equation:

\[
\text{Det}(B - \lambda A) = 0
\]  

(21)

The characteristic velocities are:

- \( v_g \) transport velocity of gas enthalpy \( h_g \)
- \( v_l \) transport velocity of liquid enthalpy \( h_l \)
- \( w - c_u, w + c_u \) pressure wave propagation velocities
- \( w - c_u, w + c_u \) void wave propagation velocities

with the following expressions used in the CATHARE code:

\[
c_s^2 = \left[ 1 + \beta \frac{\rho_m^2}{\rho_g} \right] \frac{\rho_g \rho_f c_s^2 c_1^2}{a_g \rho_1 c_1^2 + a_f \rho_f c_f^2} \frac{a_g \rho_1 + a_f \rho_f}{\rho_g \rho_1}
\]

(22)
\[ c^2_n = \frac{\tilde{\rho}_m \Delta p^{\text{int}} - \alpha_1 \alpha_1 \tilde{\rho}_g \rho_1 (v_v - v_i)^2}{\tilde{\rho}_m^2} \]  

(23)

\[ \tilde{\rho}_1 = \rho_1 + \beta \tilde{\rho}_m \]

\[ \tilde{\rho}_m = \alpha_0 \tilde{\rho}_g + \alpha_1 \tilde{\rho}_1 \]  

(24)

\( \beta \) is the added mass coefficient, \( c_1 \) and \( c_g \) are the sonic velocities in liquid and gas, and \( \Delta p^{\text{int}} \) is the pressure difference given by eq. (7).

The single pressure two fluid convective equations only are not hyperbolic without differential forces (\( F_v \), \( \Delta p^{\text{int}} \) \( \nabla \alpha \),…). Hyperbolicity is a condition of the well-posedness of the problem and a condition for the stability. However non hyperbolic equations are used in some system codes and remain stable through diffusion terms or numerical diffusion. If the stability only depends on numerical diffusion, instability may occur when the mesh size and time step are small enough to reduce the stabilizing effects of numerical diffusion.

Some important remarks about propagation phenomena in two-phase flow are necessary:

- Void waves and pressure waves are « dispersive » in two-phase flows: the propagation velocity depends on frequency or wave number due to the effect of source terms. This can be demonstrated by a linear stability analysis.
- The characteristic velocities are the high frequency limit of wave velocities which only depend on differential terms of the equations. Consequently the propagation velocity \( c_s \) in eq. (22) only represents the high frequency sound velocity, which does not play a significant role in two-phase flows since high frequency waves are the most rapidly damped by diffusion. Lower frequency wave propagation depend on the algebraic source terms and may play a dominant role in two-phase flow since they are much less damped by diffusive processes. In particular, chocked flow condition in two-phase flow is governed by low frequency sound waves.

### 3.6 Predicting Break Flow and Chocked Flow

Chocked flow or critical flow conditions may occur in reactor transients either at a break or in internal flow at flow restrictions or pumps. When sonic velocities are reached at a section (often at the smallest flow cross section area) the flow becomes independent from downstream conditions. Predicting critical flow is then of prime importance for all LOCA transients since the break flow rate controls the coolant mass inventory and consequently the core cooling capability.

System codes developed three methods for modelling break flow or chocked flow:

- 1) **Critical flow rate correlation**: the geometry of the flow restriction is simplified and a coarse meshing is used. The mass flow rate is predicted using an empirical correlation as a function of upstream flow conditions. This method may be used on option in CATHARE code [8].
- 2) **Use of characteristic velocity**: the geometry of the flow restriction is also simplified and a coarse meshing is used. A sound characteristic velocity is set to zero and simplified equations are used to predict flow evolution from upstream to the sonic section (e.g. Bernoulli equation, isentropic evolution,…). Such methods are available in TRACE [7] and RELAP5 [6] codes
- 3) **1D modelling of the flow through the nozzle**: the flow from upstream to sonic section is precisely calculated by 2-Fluid equations using a very fine meshing in the vicinity of the throat. This method is possible in the CATHARE code [8]. Such a method is only possible with an implicit numerical scheme which allows high velocity flow in small meshes without material Courant limitation.

When using the method 3) it was observed that the prediction of the mass flow rate is very sensitive to flashing in sub-cooled liquid upstream conditions and to interfacial friction in high quality flows. On the contrary a very small sensitivity to added mass which controls the characteristic velocity \( c_s \) is found and chocked flow is predicted for \( w < c_s \). This confirms that the sonic velocity which plays a role is not the characteristic velocity and is controlled by source terms associated with flashing and interfacial drag. A mechanistic modelling of interfacial friction in droplet flows guaranties good predictions of high quality critical flows. In sub-cooled upstream conditions, a semi-empirical
modelling of liquid to interface heat flux performs better than more mechanistic models and the modelling of the flashing delay due to heterogeneous nucleation plays a significant role. This method is able to capture the governing phenomena, which control the actual sonic velocity. The limitations are the following:

- Non established flow conditions make more difficult the derivation of mechanistic models
- Nucleation delay is difficult to model in a general way for all conditions
- 2D or 3D effects (such as vena contracta) in abrupt area change are not well described in a 1D model
- In the worst cases the uncertainty in predicting flow rate may be about 10%

The use of simplified modelling of chocked flow with coarse meshing in methods 2 and 3 has reached a similar accuracy as method 3. In semi-implicit numerical schemes such methods prevent from calculating high velocity in small meshes, which is CPU costly since the material Courant limit has to be satisfied. A simple coefficient may be added to take geometrical effects into account. The limitations are the following:

- Using characteristic velocity as speed of sound is not well based as explained above
- 2D or 3D effects in abrupt area change cannot be described
- In worst cases uncertainty in predicting flow rate may be about 10%

Whatever method is used, the prediction of break flow-rate or chocked flow remains a limitation for the accuracy of reactor transient simulations.

### 3.7 Predicting Stratified Flow

2-fluid 6-eq models can describe shallow water waves and predict both subcritical & supercritical flows or a hydraulic jump.

In CATHARE momentum equations, the term \( \Delta p_{\text{int}} \frac{\partial}{\partial x} \) controls level variation and waves propagation. In stratified flows the following expression of \( \Delta p_{\text{int}} \) can be derived assuming a hydrostatic transverse pressure field:

\[
\Delta p_{\text{int}} = \Delta p_{\text{strat}} = a(1 - a) \rho g L D
\]  

and a critical value can be defined:

\[
\Delta p_{\text{crit}} = a_g \rho_f v_f^2 + a_l \rho_g v_g^2
\]  

A criterion allows to distinguish sub-critical (or “fluvial”) and supercritical (or “torrential”) flow regimes:

- When \( \Delta p_{\text{int}} < \Delta p_{\text{crit}} \), the flow regime is supercritical (or “torrential”)
- When \( \Delta p_{\text{int}} > \Delta p_{\text{crit}} \), the flow regime is sub-critical (or “fluvial”)
- When \( \Delta p_{\text{int}} = \Delta p_{\text{crit}} \), it corresponds to the passage from “fluvial” to “torrential” or “torrential” to “fluvial” with a hydraulic jump

When only a mixture momentum equation is written, there is only a characteristic velocity for transporting the void fraction, whereas two momentum equation can predict two void wave velocities. Consequently, drift flux models can distinguish stratified flows but cannot capture waves nor distinguish sub-critical and super-critical flows.

### 4. INTERFACIAL TRANSFERS

The sources describing the interfacial mass, momentum and energy exchange processes between the phases, \( \sigma_i^\text{int} = \Gamma_i^\text{int}, F_i^\text{int}, Q_i^\text{int} \) are assumed to be algebraic functions of the form:

\[
\sigma_i^\text{int} = C_i^\text{fr} a_i^\text{int} X_i
\]  

with the interfacial area concentration (interfacial area per unit volume) \( a_i^\text{int} \), a driving force (in the sense of non-equilibrium thermodynamics) \( X_i \), and a corresponding transfer parameter \( C_i^\text{fr} \). The driving forces \( X_i \) are assumed to be linear (or higher order) functions of parameters describing the deviations from the thermal and mechanical
equilibrium between the phases: e.g. superheating of the gas phase, \( T_g - T_{sat} \) or liquid phase \( T_l - T_{sat} \), for the heat and mass transfer processes and the “slip” velocity, \( v_g = v_1 \), for interfacial friction (drag).

4.1 Interfacial Area

As indicated in equation (27) a key parameter for the interfacial transfer process is the interfacial area concentration which strongly depends on the (local) spatial phase distribution as characterized by the two-phase flow regime. Since the balance equations give no information on the two-phase flow structure, additional modelling is required to provide an at least approximate value for the interfacial area concentration.

The standard approach in nearly all TH-codes is to correlate algebraically the interfacial area concentration with major governing state and transport parameters like volumetric concentration of vapour and liquid, \( \alpha_g \) and \( \alpha_l \), phase velocities, densities, viscosity, surface tension, etc:

\[
a^{int} = f(\alpha_g, v_1, \rho_1, \mu_1, \sigma, ...) \quad i = g,l
\]

The interfacial area transport model, which consists of a balance equation for interfacial area concentration and related source terms

\[
\frac{\partial}{\partial t} (a^{int}) + \nabla \cdot (a^{int} \vec{v}^{int}) = \sigma^A
\]

seems rather attractive since it allows the description of interfacial processes as a dynamic, time-dependent process and is expected also to provide a more physically-based prediction of flow regime transitions.

4.2 The Use of Flow Regime Maps

Every flow regime has its internal structure and its transfer mechanisms. So it seems natural to use a flow pattern map in a code and to develop correlations for mass momentum and energy transfers which depend on the flow pattern. This is usually done via flow regime maps where specific two-phase flow patterns are identified as functions of input data such as superficial gas and liquid velocities, flow rates or more complex dimensionless parameters. The highly empirical flow regime maps are based on a large amount of measured data for different fluids, vertical and horizontal flow conditions and different pipe diameters. Typical examples are the flow pattern maps of Baker [12], Mandhane [13], and Taitel and Dukler [14]. Nevertheless, all these flow regime maps are valid only for steady state - or quasi-steady state - and fully developed – or quasi-developed - flow conditions although rapid transient and non-established flows also exist in nuclear reactors under accident conditions. Having these restrictions in mind it might be justified to further simplify the flow maps as is usually done in all the codes.

As an example, the flow regime map as used in RELAP5/MOD3 for horizontal flow is shown in Fig. 2. Major selection parameters are void fraction \( \alpha_g \) and total mass flow density \( G_m \).

Specifically modelled are the bubbly, slug annular-mist, mist and stratified flow regimes. Transition criteria are also algebraic relations between flow parameters. To avoid discontinuities, some transition regions are included where all parameters are interpolated from the adjacent flow regime boundaries. This only provides artificially smooth transitions but cannot take into account the relaxation time constant associated to the flow process responsible for the transition. Horizontal stratified flow conditions are assumed to exist for \( G_m < 3000 \text{ kg/m}^2\text{s} \) and \( |V_g - V_l| < V_{crit} \), with the critical velocity according to Taitel and Dukler [14].

\[
V_{crit} = \frac{1}{2} \left( \frac{(\rho_1 - \rho_g) g A}{\rho_g D \sin \theta} \right)^{1/2} \left( 1 - \cos \theta \right)
\]

where \( \theta \) represents the angle between the vertical symmetry axis and the intersection of the collapsed liquid level in the circular pipe with the pipe wall.
The criteria for bubbly to slug, slug to annular mist and annular mist to mist flow regimes are just given by limiting values of void fraction and mass flux:

Similar flow regime maps are applied for vertical flow including pre-CHF and post-dry-out conditions, high mixing flow conditions in pumps and for mixing of sub-cooled ECC water with near saturated steam and resulting condensation processes.

Flow regime maps provide a necessary information on interfacial structure and interfacial area based on experimental observation and with some theoretical basis and allows a mechanistic modelling of interfacial transfers.

The most important transitions correspond to interfacial area changing by several orders of magnitude, e.g. at the onset of droplet entrainment or when phase stratification occurs in a bubbly flow in a horizontal pipe.

The main limitations of using such flow regime maps are:

- **Range of validity**: no observations are available in high pressure steam water flows to validate the flow maps in such reactor conditions. Also data in large diameter pipes are very limited.
- **Flow geometry**: very limited observation of flows in complex geometry (rod bundle) is available although geometrical effects are likely to be significant. Also effects of some singularities are not taken into account.
- **Steady and established flows** are necessary to establish such maps, and they are extrapolated in transient or non-established conditions. History effects and relaxation time constants associated with regime transitions are not taken into account.

### 4.3 How to Develop Closure Relations

Physical models are required to close the system of equations. Closure relationships concern mass, momentum, and energy exchanges between phases and between each phase and the wall. In a first step code developers look in the scientific literature to find such models. When they exist, they are often developed based on theoretical work and from rather academic experimental data such as air-water flow in a circular pipe with a large length to diameter ratio. They must be later confronted to more industrial flow conditions, with steam-water data in larger ducts of various shapes. Specific separate effect tests were performed and analysed to investigate two-phase flows in conditions more representative of the reactor transients to be simulated. Based on these data new correlations were developed when existing models were not satisfactory. The degree of empiricism depends on the comprehension of the physical mechanisms involved. In the domain where experimental and theoretical knowledge was still missing, extrapolations were adopted by making simple assumptions.

Thermal and mechanical transfers are interconnected in steam-water two-phase flows. However it was often assumed in a first approximation that mechanical interactions do not strongly depend on thermal exchanges. A step-by-step method was then used. Mechanical terms were first derived from experiments where thermal non
equilibrium is negligible. Interfacial heat transfer terms were then derived. Finally wall to fluid heat fluxes are correlated.

One may distinguish four different approaches used to establish the closure relations:

- **The fully empirical approach**: representative experiments are carried out in the required range of parameters and a transfer term of the system of equations is measured together with the main variables. A correlation is then established between the transfer term and the main variables of the flow by using some fitting technique or interpolation technique. CATHARE and RELAP use CHF look up tables for the Critical Heat Flux prediction. This method may be the most accurate but it requires a lot of experimental data and it does not allow extrapolations out of the domain covered by data.

- **Empirical approach with dimensional analysis**: correlations are here expressed with dimensionless numbers. Since many dimensionless numbers exist in two-phase flows, a preliminary analysis may allow to identify the controlling physical processes and only the dimensionless parameters which play a role are used in the correlation. This method was used to establish “Full range drift flux correlations” by EPRI.

- **The phenomenological or mechanistic approach**: A governing physical mechanism is assumed and an expression of the transfer term is derived theoretically. Expressions of the added mass force, or of the interfacial friction for dispersed droplet flows are examples of this method. The validity is verified a posteriori by comparison with experimental data. The quality depends on the appropriateness of the basic assumptions made on the governing processes.

- **The semi-empirical approach**: Compared to the previous method, weaker assumptions are made, which allow to derive expressions theoretically with some free parameters to tune with experimental data. The interfacial friction or interfacial heat transfer expressions are often of this type.

First versions of the system codes used mainly the mechanistic approach but, after an extensive validation, the degree of empiricism of closure relations was progressively adapted to the lack of comprehension and to the complexity of the physical mechanisms involved in such a large variety of two-phase flows.

### 4.4 Interfacial Drag

Prediction of liquid water repartition in primary circuit is of paramount importance for predicting core cooling capability and the sensitivity of many accidental transients to interfacial friction was found to be very high. Consequently, a huge effort was paid to interfacial friction due to large variety of component geometries and flow regimes.

For dispersed flow a simple law is usually applied which can be written for mono-dispersed conditions assuming spherical particles as:

\[
F_{\text{int}} = -F_{\text{g}} = C_w \frac{1}{2} \rho_c \pi r_p^2 n_p |v_g - v_1| |v_g - v_1|
\]

(31)

with the density of the continuous phase \( \rho_c \), the particle radius \( r_p \), the particle number per unit volume \( n_p \), and a Reynolds-number dependent friction coefficient \( C_w = f(Re) \).

Introducing the interfacial area concentration \( a_{\text{int}} \), a more general form for the interfacial drag is obtained from equation (31)

\[
F_{\text{int}} = -F_{\text{g}} = \frac{1}{8} C_w \rho_c a_{\text{int}} |v_g - v_1| |v_g - v_1|
\]

(32)

More precisely, the interfacial drag or interfacial friction force correlates together the stationary part of both viscous and pressure interfacial forces.

For the estimation of the interfacial friction coefficient various modelling approaches are used ranging from simplified mechanistic models to largely empirical correlations. One of the major problems is that all these correlations can only be indirectly verified based on measured local flow velocities or mass flow rates. The second problem is the prediction of either the particle size \( r_p \) or interfacial area \( a_{\text{int}} \) since there is no possibility to describe dynamics of coalescence, break up, nucleation, collapse, vaporization, condensation, which are all phenomena
affecting \( r_p \) and \( a^{\text{int}} \). At last, although a mechanistic formulation of interfacial can be applied for dispersed bubbly or droplet flows, the effect of space averaging may also degrade the predictive capability.

The most difficult cases is the churn flow where no clear interfacial structure can be assumed resulting in more empirical correlations although churn flow is more likely to occur than slug flows in industrial geometry. Drift flux correlations were developed based on a large experimental data base. However, strictly speaking they are only valid for steady state - or quasi-steady state - and fully developed – or quasi-developed - flow conditions although rapid transient and non-established flows also exist in nuclear reactors under accident conditions. In some codes (e.g. RELAP5/MOD3, CATHARE) drift flux correlations were re-transformed into interfacial forces which are then used within the momentum equations for liquid and gas/vapour. In some codes, the geometry of the duct (pipe, rod bundle) is a parameter of the flow regime and separate set of interfacial friction correlations are used for each geometry.

For separate-phase flows like annular flows or stratified flows Eq 32 ,which is rather mechanistic, can also be applied replacing \( \rho \) by the density of the light phase \( \rho_g \). In separate-phase flows, the main difficulty is to express the interfacial area which depends on surface roughness due to interfacial waves.

No clear mechanistic formulation for annular-mist or stratified-mist flows since there are two liquid fields having very different velocities.

Extensive validation and tuning was necessary to reach reasonable predictions in the most sensitive situations such as the level swell in core. Many SET and IET were used to progressively improve the modelling. Initial mechanistic models due to academic investigations were often far from applicable to industrial problems. Most correlations use the semi-empirical approach and some difficult cases the empirical approach with dimensional analysis. Finally, satisfactory predictions are now obtained for most situations of interest.

### 4.5 CCFL Prediction

Counter-Current Flow Limitation (CCFL) is likely to occur in complex geometries within the reactor circuits:

- Upper Tie Plate in a core
- Steam Generator tube inlet
- Hot leg bend of a PWR

where standard drift flux models or interfacial friction models are no more valid due to local effects.

All system codes developed local treatment to obtain a pre-determined flooding curve (Wallis type or Kutateladze type). A drift flux model was made consistent with a flooding correlation in the ATHLET code. An interfacial friction model was made consistent with a flooding correlation in the CATHARE code.

System codes are not predictive for CCFL since the user must know the solution which depends on the actual geometry and representative experiments are necessary to establish the adequate flooding correlations. CCFL is for interfacial friction what form losses are for wall friction.

### 4.6 Interfacial heat and mass transfer modelling

Interfacial heat and mass transfers have to be modelled depending on the flow regime. Liquid to interface heat flux \( Q_{l}^{\text{int}} \) and gas to interface heat flux \( Q_{g}^{\text{int}} \) are expressed in the following way:

\[
Q_{l}^{\text{int}} = -q_{l}^{\text{int}} a^{\text{int}} = -H_{l}^{\text{int}} (T_{l} - T_{\text{sat}}) a^{\text{int}}
\]

\[
Q_{g}^{\text{int}} = -q_{g}^{\text{int}} a^{\text{int}} = -H_{g}^{\text{int}} (T_{g} - T_{\text{sat}}) a^{\text{int}}
\]

\( H_{l}^{\text{int}} \) and \( H_{g}^{\text{int}} \) being heat transfer coefficients. The interfacial mass transfer can be calculated from an energy balance through the interface:

\[
\Gamma_{g} = \frac{H_{l}^{\text{int}} (T_{l} - T_{\text{sat}}) + H_{g}^{\text{int}} (T_{g} - T_{\text{sat}})}{h_{l}^{\text{sat}} - h_{l}^{\text{sat}}} a^{\text{int}}
\]
In most cases no high sensitivity to interfacial transfers due to meta-stable superheated liquid or subcooled vapour was found except in flashing flows where superheated liquid affects break flow.

On the contrary a high sensitivity was found to transfers due to superheated vapour with post-CHF transfers in a core. Vapour to interface heat transfers are often modelled as follows:

\[ Q_{\text{g}^\text{int}} = \frac{6(1-a)}{\delta} \frac{\text{Nu} \cdot \lambda_{\text{G}}}{\delta} (T_{\text{sat}} - T_V) \]  

(35)

with:

\[ \text{Nu} = 2 + 0.57 \text{Re}^0.5 \text{Pr}^{0.33} \]  

(36)

Here the main difficulty is to predict the droplet diameter which may be strongly affected by the presence of spacer grids in a core. Many experiments were necessary to obtain adequate models of the core Reflooding situation.

A significant sensitivity of system pressure upon transfers due to sub-cooled liquid in case of Direct Contact Condensation due to ECCS injection was also found. The effect of liquid turbulence on liquid to interface heat transfer had to be correctly modelled before having reasonable predictions.

Most correlations use the semi-empirical approach and satisfactory predictions are now obtained for most situations of interest. However the occurrence of condensation driven flow instabilities after ECCS injection remains rather difficult to predict.

5. MODELLING WALL TRANSFERS

5.1 Wall Friction

The one-dimensional form of the momentum conservation equations requires expressions for pressure losses due to wall viscous and pressure forces which are usually split into two contributions (1) the wall shear losses continuously distributed over the pipe or duct length and (2) local “form losses” due to abrupt area changes, elbows and fittings or even more complicated flow passage geometries.

Wall shear forces are usually deduced from empirical correlations for wall friction pressure losses assuming steady state, fully developed flow conditions and constant pipe cross section.

For single phase flow of liquid \((i = l)\) or vapour \((i = g)\) the frictional pressure drop is modelled as

\[ \frac{(\Delta p)^\text{wall}}{\Delta x} = -\lambda_i \frac{G_i^2}{2 \rho_i D} \]  

(37)

where the wall friction factor \(\lambda_i\) is assumed to be a function of the Reynolds number and wall roughness

\[ \lambda_i = f(\text{Re}) \quad \text{with} \quad \text{Re} = \frac{G_i D}{\mu_i}. \]  

(38)

For the frictional pressure in two-phase flow various models and correlations have been proposed which are either directly or in a modified version implemented into present TH-code: Lockhart–Martinelli [15], Martinelli–Nelson [16], Chisholm [17], Baroczy [18] and others. All of these models follow the same approach to express the overall pressure drop in terms of the corresponding pressure drop for the liquid respectively vapour alone flow in the pipe:

\[ \left( \frac{dp}{dx} \right)^\text{wall}_{\phi} = \Phi_\phi^i \left( \frac{dp}{dx} \right)^\text{wall}_{\phi} \]  

(39)

and the corresponding empirical two-phase flow multipliers \(\Phi_\phi^i\) and \(\Phi_\phi^g\).
For the two-phase flow multipliers, the following correlations were developed:

\[ \Phi^2_g = 1 + \frac{c}{X} + \frac{1}{X^3} \]
\[ \Phi^2_l = 1 + cX + X^2 \]  

(40)

with the Lockhart-Martinelli ratio:

\[ X^2 = -\frac{\left(\frac{dp}{dx}\right)^{wall}}{\left(\frac{dp}{dx}\right)^{gwall}} \]

(41)

The empirical parameter \( c \) depends on the liquid and gas mass flow, void fraction and Reynolds numbers. From equations (37) to (41) only the overall frictional pressure drop can be estimated. The two fluid model requires a partitioning of the pressure drop and the determination of related shear forces for liquid and vapour. As an example the phasic wall shear forces as used in RELAP5/Mod3 are given below which have been derived following the approach of Chisholm [13]:

\[ F_{wall}^{l} = \frac{a_l}{D} \left(\frac{dp}{dx}\right)^{wall} \left(\frac{Z^2}{\alpha_g + a_l Z^2}\right) \]
\[ F_{wall}^{g} = \frac{a_g}{D} \left(\frac{dp}{dx}\right)^{wall} \left(\frac{1}{\alpha_g + a_g Z^2}\right) \]

(42)

with the parameter

\[ Z^2 = -\frac{\lambda_l \rho_l b_l^2 a_{lw}^2}{\lambda_g \rho_g v_g^2 a_{gw}^2} \]

(43)

The newly introduced volumetric fraction in equation (43) for liquid and vapour at the wall region \( a_{lw} \) and \( a_{gw} \) respectively might differ from the corresponding bulk values and as such allow a certain correction for inhomogeneous phase distribution for annular, annular-mist, inverted annular or slug flow regimes.

In addition to the wall shear the wall friction force \( F_{wall}^{l} \) also includes pressure losses due to geometrical effects like abrupt area changes, elbows and fittings or even more complicated flow passage geometries. For abrupt area change or orifice, mechanistic models are often used (e.g. RELAP or TRACE) based on a steady state formulation for Borda-Carnot shock losses and related vena-contracta effects for a sudden contraction. More complex geometries are described as

\[ (\Delta p)^{FL} = a_g \frac{G_g^2}{2 \rho_g} K_g \]
\[ (\Delta p)^{FL} = a_1 \frac{G_1^2}{2 \rho_1} K_1 \]

(44)

where the corresponding energy loss coefficients \( K_g^{FL} \) and \( K_1^{FL} \) respectively are user input parameters. Due to the lack of systematic data bases for form losses (in particular under two-phase flow conditions) the energy loss coefficients are often used as tuning parameter to represent measured data for initial steady state and (where available) for transient conditions.

Friction pressure losses and form losses are examples of the use in system codes of very empirical correlations. The predictive capability of these models remains rather limited without any big consequences on accidental
transient simulations. Indeed, in many two-phase situations, due to rather low velocities, the gravitational pressure loss is larger than the friction pressure loss.

5.2 Wall Heat Transfer Modelling

All system codes use a rather complex heat transfer regime map with many cases and a complex logic. Wall to fluid heat transfers are generally splitted into three components:

- Wall to liquid heat flux $q_{\text{wall}}^{\text{wall}}$ with possibly sensible heat plus radiation heat fluxes
- Wall to gas heat flux $q_{\text{g}}^{\text{wall}}$ with possibly sensible heat plus radiation heat fluxes
- Wall to interface heat flux $q_{\text{int}}^{\text{wall}}$ when energy is directly used for interfacial mass transfer in case of boiling or condensation

In the case of a heating wall, the whole boiling curve has to be described with successive regimes:

- Convection to liquid
- Nucleate Boiling with models for Onset of Nucleate Boiling (ONB), Net Vapor Generation, (NVG), and partition of the fluxes to liquid and interface in sub-cooled boiling, up to saturated boiling
- CHF : Critical Heat Flux
- Transition Boiling
- Film Boiling with several regimes for Inverse annular, Inverse Slug and Dispersed Flow film boiling

Some examples are given below to illustrate the differences between codes.

**Forced convection heat transfer**
RELAP, TRAC & CATHARE use the Dittus & Boelter [19] correlation developed for single phase flow in pipes. TRACE V5.0 implemented a more complex model with laminar & turbulent regimes, effect of film temperature, effect of geometry (rod bundle vs tube), effect of void on convection to liquid and of droplets in convection to gas. Since no experimental data were available to quantify these effects, only an indirect validation was possible. However, most transients of interest are not very sensitive to heat transfer coefficient in single phase flow.

**Critical Heat Flux**
RELAP, TRACE & CATHARE codes use some version of lookup tables by Groeneveld et al. [20] for CHF. TRACE uses critical quality correlations in the Dry-Out domain at high quality. This is the most empirical approach for closure relations. It illustrates the lack of understanding of the various physical processes governing the critical heat flux in such a wide range of flow parameters.

**Nucleate Boiling**
TRAC & RELAP use Chen [21] heat transfer coefficient correlation of nucleate boiling whereas CATHARE uses the Thom et al. [22] heat transfer coefficient Saha-Zuber for NVG. TRACE uses the Gorenflo model with the Basu model for ONG and Saha-Zuber for NVG.

**Film Boiling**
RELAP uses a modified Bromley [23] film boiling correlation whereas CATHARE uses a modified Berenson [24] film boiling correlation. TRACE V5.0 developed a new model including laminar convection through the vapour film, and an enhancement due to waves in a vaporisation term boiling. It should be mentioned that in such film boiling regime, the 5 heat transfers of the 6-equation model may play a role in the global efficiency of the heat transfer: wall to vapour heat flux $q_{\text{wv}}$, wall-to interface vaporisation heat flux $q_{\text{wi}}$, wall to liquid direct heat flux $q_{\text{wl}}$, vapour to interface heat flux $q_{\text{vi}}$ and liquid to interface heat flux $q_{\text{li}}$ may play a role.

**Concluding remarks**
From the few examples above one can draw some conclusions:

- Codes rarely selected the same correlation. First versions of the codes used mechanistic models which were often modified through empirical corrections after comparison with experimental data of the validation.
- Even when codes do not use the same basic correlation, they often converged to the same degree of empiricism in the selected final correlation. The level of empiricism corresponds to the understanding of basic processes.
- A high accuracy is not required for all models and more attention was paid to the most sensitive models in accidental transients and an effort was made to improve them using corrections based on experimental validation. The sensitivity to the models being not uniform, it results that the accuracy and reliability of the models is not uniform.
6. 3D MODELLING IN SYSTEM CODES

System codes like RELAP and ATHLET codes first developed “cross-flow junctions” between 1D modules to represent some multi-dimensional flow features. 1D equations are written in the main direction of the flow and simplified momentum equation are used in the transverse direction to allow mass exchanges between several parallel channels. This simplified approach of 3D flows may be sufficient in some cases particularly for porous body like a reactor core when only small cross flows exist due to high resistance to transverse velocity.

Explicit 3-dimensional modules exist as an option in the codes TRACE and CATHARE for the reactor pressure vessel. They represent a straightforward extension of the one-dimensional modules for cylindrical or Cartesian coordinates. The main objective of such 3D modules is the modelling of large scale 3D effects in a pressure vessel during LBLOCA such as downcomer penetration of ECCS water, Reflooding of the core with transverse power profile effects.

Due to the heavy computational effort needed, the 3-dimensional modules are being used mainly for fast (short) transients like large break LOCA but with the increasing computer power CATHARE 3-D Module is now also used for Small Break LOCA. In most applications, rather coarse nodalization schemes (about 1000 nodes for a CATHARE Pressure Vessel 3D nodalization) are applied and consequently the advantage of a 3-dimensional modelling of the flow processes might be offset to a certain extent. However, large scale 3D effects can be better modelled than with 1D models.

Using such coarse nodalization is far from being converged in space. Then these 3D modules must validate together the physical model, the numerical scheme, and the reference vessel nodalization using scale 1 experiments such as UPTF tests. This does not prevent from compensating errors but such 3-dimensional modules are a progress compared to parallel channel representation with cross-junction connections.

7. NUMERICAL SCHEMES

Numerical methods in system codes are mostly similar. Most codes use first order finite volume schemes with a staggered mesh and the donor cell principle. The time discretization varies from the semi-implicit scheme used in first versions of RELAP and TRAC to nearly implicit and multi-step schemes used in more recent versions, or the fully implicit discretization used in CATHARE and ATHLET. These methods are known for their stability and robustness but they are rather diffusive and dissipative.

Mass and energy equations use a conservative form and discretization is designed in order to yield the best mass and energy conservation. The wall conduction is implicitly coupled to hydraulic calculations.

The CATHARE code takes care of the hyperbolicity of the system in order to warrant stability even for very small time steps and meshes. The two-fluid model is not an unconditionally hyperbolic system of equations if differential terms such as added mass force or interfacial pressure term are not added. Codes which did not satisfy hyperbolicity remain stable and extremely small mesh size would be necessary to produce instability. Theoretically all calculations should be converged in space and time. In practice convergence tests are easily performed for simple analytical tests and some recommendations can be deduced for system tests or reactor calculations.

The problem of convergence in meshing is somewhat different for three-dimensional Pressure Vessel models. Convergence tests cannot reach the exact solution since it would require unaffordable CPU cost. So the physical model including the equation and the closure laws must be validated together wit the numerical scheme for a given meshing and possibly with scale 1 experiments. This is possible using the UPTF data or large scale data of the 2D-3D program. The same meshing can then be used for reactor calculations.

The numerical schemes were evaluated by calculating numerical benchmark tests cases.

7.1 The RELAP 5 Code

Sum and difference formulations of the phasic balance equations for mass and momentum are used in order to facilitate the transfer between two-phase and single-phase flow conditions (phase appearance/disappearance).

The standard option in RELAP5 is a semi-implicit scheme which might be seen as an extension of the ICE method of Harlow and Amsden [25]. Only those governing parameters are evaluated at new time values which are needed to
maintain stability and accuracy of the method or to avoid Courant time-step size limitations based on pressure waves. The implicitly evaluated parameters include mass and energy inventories, interfacial mass, momentum and energy transfer terms, pressure gradients in the momentum equations and phasic velocities in the mass and energy fluxes. In order to maintain linearity within the dependent parameters, the state relationships for the phasic densities as well as the new-time products of dependent parameters are linearized around old-time values.

With the restrictions described above, a linear system of equations is obtained which can be further reduced to N linear equations for N unknown pressure values in N computational cells which can be written in matrix form as:

$$ A p^{n+1} = b $$  \hspace{1cm} (45)

where the matrix $A$ and the vector $b$ are determined only by old-time values. For a strictly one-dimensional case, the resulting matrix $A$ is of tri-diagonal structure; for a hydraulic network with multiple connections a sparse matrix is obtained which is solved by a standard sparse matrix solver. A back-substitution is then used to calculate the remaining parameters like phasic velocities, void fraction, phasic internal energies, etc.

There are various criteria to check whether a time step advancement was successful:

- a “mass error” check comparing the numerical mixture density with the density calculated from state equations:

$$ E_{m} = \max \left( \frac{|\rho_{m}^{i} - \rho_{I}^{i}|}{\rho_{I}^{i}} \right) \leq 0.8 \times 10^{-4} \quad \text{for } i = 1, 2 \ldots \text{N} \hspace{1cm} (46) $$

- a material Courant limit check at each cell:

$$ \Delta t < \left( \frac{\Delta x}{c_{max}} \right) = \frac{\max \left( \alpha_{f}^{n}, \alpha_{g}^{n} \right)}{\max \left( \alpha_{f}^{n}u_{f}^{n}, \alpha_{g}^{n}u_{g}^{n} \right)} \quad \text{for } i = 1, 2 \ldots \text{N} \hspace{1cm} (47) $$

- a check whether material property values are found to be outside their range of validity.

In case of violation of at least one of these conditions, the time step is repeated with halved time step. If a user-defined minimum time step is reached, the calculation is aborted and a diagnostic dump is printed.

As described above, the numerical technique used in RELAP5 is single-step approach where the new time advancement is solved directly without any iteration.

As an option, RELAP5 includes a “nearly implicit” scheme based on a 'fractional step' approach. The method is claimed to provide a sufficient degree of implicitness to eliminate the material Courant-type stability limit. The first step uses the same finite difference equations as the semi-implicit method with the difference that the momentum flux terms are treated (linearly) implicitly and that the numerical viscosity terms have been cancelled. This results in a block tri-diagonal matrix which is solved for the phase velocities by a sparse matrix solver. The other new-time values are then calculated by back-substitution.

The second step involves only the conservative form of the mass and energy equations. From the updated values of the phasic mass and energy, the final new-time values for the scalar quantities are calculated with the help of the linearized state equations.

The success of the time advancement is controlled in the same manner as for the semi-implicit scheme with the only difference that the maximum material Courant number is set to 10 for transients and to 20 for steady-state calculations.

### 7.2 The TRAC Code

The numerical method applied is based on a staggered grid/donor cell approach with a partial implicit time integration to eliminate the material Courant stability limit condition. The solution methods applied in TRAC is the Stability-Enhancing Two Step (SETS) method of Mahaffy [26] which comprises the following five steps:
First guesses for the phasic velocities are calculated from the simplified separate momentum equations (equations of motions) using only new-time velocities in the linearized expressions for the drag force. All other values (including pressure) are taken from previous time level.

In this step updated (intermediate) phasic velocities are calculated again from the momentum equations (“stabilizer equations of motion”). Intermediate term values include only phasic velocities in the momentum flux and wall friction terms. Using the velocities from step 1 in the drag forces, the two momentum equations are decoupled with respect to each other. The resulting two sets of tri-diagonal systems are directly solved for the phasic velocities.

Steps 3 and 4 are the semi-implicit part of the SETS method. In step 3, the momentum equations are used again, now with new-time values for the phasic velocities in momentum flux, wall friction, interfacial drag and interfacial momentum transfer terms associated with mass interfacial transfer (evaporation/condensation). In addition first estimates for the new-time pressure values are introduced in the spatial pressure derivative terms. As a result, linear expressions are obtained for the new-time velocities as functions of the intermediate pressure values.

This step provides intermediate new-time values for the scalar (thermodynamic) variables from the expanded form of the mass and energy balance equations using the phasic velocities as calculated by step 3. The resulting non-linear set of equations for the void fraction, pressure and phasic temperatures is solved by a Newton-Raphson iteration method keeping the phasic velocities and wall temperatures constant.

In this final “stabilizer step” the mass and energy equations are used in conservative form to calculate new-time macroscopic values for the phasic mass and energy \((\alpha_1 \rho_1)_{n+1}, (\alpha_1 u_1)_{n+1}\), using the phasic velocities from the semi-implicit steps 3 and 4. The expressions for the macroscopic mass and energy are then linearized by a first-order Taylor expansion. The resulting system of 4x4 (or respectively 5x5 with non-condensable gases) linear equations are separately solved for each cell by a direct Gauss elimination to obtain final values for phasic temperature, pressure and void fraction.

The time step control is based on convergence criteria for user-specified threshold values for local pressure and void fraction increments.

One might note from the description above that in the SETS method the phasic flow velocities and wall temperature are not directly involved in the iteration process for the semi-implicit solution (apart from the linear relationships between local velocities and pressures of step 3).

Recently, US-NRC decided to merge RELAP 5 and TRAC code into a single consolidated code (called TRACE). It was envisaged to keep only the semi-implicit time discretization where the material Courant limit has to be respected.

### 7.3 The CATHARE Code

The numerical method in CATHARE is based on a standard staggered grid donor spatial discretization combined with a fully-implicit time integration scheme for 0-D and 1D modules. This leads to a highly non-linear system of equations which is solved iteratively by a standard Newton-Raphson method.

Introducing the Jacobian \(J\) a linear system of equations is obtained which can be written in the following compact form

\[
J \delta X = S
\]

with the vector of variable increment \(\delta X\) and the residual vector \(S\) respectively. Convergence is assumed to be reached if the increments for pressure, void fraction, phasic enthalpies and flow velocities are below pre-defined limiting values. The time step control is done automatically based on the time step history:

\[
(dt)^{n+1} = k(dt)^n
\]

with the factor \(0 \leq k \leq 3\) depending on the number of iterations needed in previous time steps.

The fully implicit time integration represents a relatively large computational effort, however this might be compensated by the increased numerical stability and the possibility to choose greater time steps for steady state, for slow and long transients and in case of small local cell sizes and large velocities. The latter will be of specific importance in calculations where the internal choking option in CATHARE is used which requires a fine nodalization upstream of the critical section. For the 3-D module of CATHARE, which may be used to model the Pressure Vessel, a, semi-implicit technique is used.
7.4 The ATHLET Code

The spatial discretization is based on a “Lumped Parameter” approach where the whole flow domain is substituted by a network of control volumes connected by one-dimensional flow paths. Mass and energy balances are formulated for the individual control volumes using pressure, phasic temperatures and vapour mass fraction (vapour quality) as the major dependent parameters. Mass, momentum and energy fluxes crossing the volume boundaries are determined from an (algebraic) integration of the momentum balance equations over flow path connections where some scalar quantities in the junctions are taken from the donor volume. The Lumped Parameter approach is, to a certain extent, equivalent to the staggered grid donor techniques of RELAP or CATHARE. However, it might offer some higher degree of flexibility to construct hydraulic networks: (1) theoretically, a control volume can be connected to any other volume in the cluster and (2) the flow conditions within a control volume do not have to be homogeneous. The latter feature is optionally used in the ATHLET code to predict a moving mixture level in a vertical pipe represented by a stack of control volumes. Nevertheless, it is worth mentioning that the lumped parameter approach cannot provide true multi-dimensional modelling in cases where the whole flow domain is represented simply by a two- or three-dimensional distribution of control volumes.

With the Lumped Parameter spatial discretization, a system of first-order, non-linear Ordinary Differential Equations (OED) in time is obtained

\[
\frac{d\mathbf{Y}}{dt} = f(\mathbf{Y}, t) \quad \text{with initial values } \mathbf{Y}(t_0) = \mathbf{y}_0
\]

where the solution vector \(\mathbf{Y}\) contains the major dependent variables in all network elements (control volumes and junctions). The initial value problem is solved by the general purpose Forward-Euler Backward-Euler (FEBE) ODE solver as described by E. Hofer [27].

For the implicit time integration, the solution vector is discretized in time as follows:

\[
\frac{\mathbf{Y}^{n+1} - \mathbf{Y}^n}{\Delta t} = f(\mathbf{Y}^{n+1}, t^{n+1})
\]

or, after linearization

\[
\frac{\mathbf{Y}^{n+1} - \mathbf{Y}^n}{\Delta t} = f(\mathbf{Y}^n, t^n) + A^n(\mathbf{Y}^{n+1} - \mathbf{Y}^n) + \Delta t \frac{\partial f}{\partial t}
\]

with the Jacobian matrix \(A\) having the elements \(A_{i,k} = \frac{\partial f_i}{\partial y_k}\).

In its basic form, the linear implicit integration is only of first-order in time. In order to obtain higher order accuracy, the linear Euler solver is repeated with different downscaling (multiple) time-step sizes \(\Delta t/n\) with \(n = 1,2,3\). The resulting sequence of solutions for \(t = \Delta t\) is then used for an extrapolation of the solution and an estimate of the corresponding time discretization error.

During the whole procedure a block sparse matrix package (FTRIX) is applied to handle the repeated calculations of the Jacobian and to facilitate the solution of the resulting linear system of equations.

The method described above has two major advantages: (1) it provides a “clean” interface between the modelling and the numerical method applied. This makes it relatively easy to handle models having a different degree of sophistication within the framework of the same code, and (2) it allows a rigorous error control and related time-step selection. A problem of the technique can be related to the multiple calculations of the Jacobian matrix which might affect the efficiency and robustness of the method. This might be strongly affected by the general stiffness of the system of equations (e.g. due to the wide spectrum of characteristic times for various two-phase flow processes) and the presence of (quasi) discontinuities in the governing system of equations resulting from poor or insufficient modelling. These are challenging conditions in any numerical simulation of two-phase flow; however, their effects might be strongly amplified in any time-extrapolation techniques based on numerical differentiation of the flow equations.
8. SUMMARY OF MAIN CAPABILITIES AND LIMITATIONS OF PRESENT SYSTEM CODES

After 30 years of validation and improvements, system codes are able to predict the main phenomena of most accidental transients of PWR & BWR with a reasonable accuracy and allow reliable conclusions on safety issues.

8.1 Why the Two-Fluid Model was the Right Choice?

The present generation of system codes is based on the two-fluid model. This choice was made after having identified unacceptable drawbacks of previous models based on improvements of the Homogeneous Equilibrium Model. In the domain of simulation of system codes for reactor accidental transients, all kinds of thermal and mechanical non equilibrium may exist.

Sub-cooled liquid with direct contact condensation after ECCS injection have to be modelled. Superheated vapour has to be modelled mainly when Post-CHF heat transfer occurs in the core. Meta-stable superheated liquid and sub-cooled vapour exist in flashing flows with a small relaxation time constant (of the order of $10^{-3}$ second). Only models with two mass balance equations plus two energy balance equations can model all these situations.

Mechanical non equilibrium is also encountered in most situations with possible weak coupling between phases. Using only one momentum equation with a drift flux models is sufficient for many situations, particularly when the coupling between phases is rather strong. Two momentum equations are necessary in other cases particularly when inertial forces play a role in the slip ratio. Droplets created in a Core during a reflooding are entrained by steam but do not reach the equilibrium slip velocity before leaving the core due to high inertia. Better capabilities are also found for stratified flows by writing two momentum equations, which allow to represent wave propagation phenomena.

The choice of the two-fluid model promoted a very extensive experimental program required for validating all closure relations. Due to this effort, the choice of the two-fluid model was a success.

Multi-field models are expected to have better capabilities for annular-dispersed flows or stratified-dispersed flows but a higher CPU cost and a lack of experimental data for validation temporarily disqualified them.

8.2 The Capabilities of System Codes seen from the Validation

An optimistic view of the code capabilities results from the assessment calculations including both Separate Effect Tests and Integral Effect Tests. Separate Effect Tests with well known boundary conditions and many measured flow parameters provided enough data for validation of closure laws. Integral Effect Tests are then necessary to check the general consistency of all models, to verify that all important phenomena are modelled, to validate the nodalization in conditions close to reactor application.

Separate Effect Tests are generally well calculated since they were extensively used for the development or the improvement of the closure relations. SET with well known boundary conditions are the only way to determine the validity and accuracy of each closure relation.

Many Integral Effect Tests are also well calculated. One can consider three main cases:

- All the important phenomena are well predicted with a good timing and a good accuracy.
- The most important phenomena are predicted. The timing and the accuracy are not perfect but it does not prevent from clear conclusions on safety issues.
- Some important phenomena are not predicted or are predicted with a very bad accuracy.

The first type of calculation is still exceptional but the second type became more and more frequent and the third type was progressively eliminated.

Calculations of many integral tests with system codes allowed to point out and to solve some problems:

- Problems in modelling a component: Problem may come from an inadequate choice of module to model a component. All codes have several types of modules with specific capabilities. The user must select a module depending on the transient to calculate. For example a common modelling of a pressurizer with the CATHARE code uses a simple two node Volume module which is sufficient for all LOCA transients where the pressurizer is rapidly empty. For transients such as a Loss of Feedwater with PORV opening or a
Multiple SGTR with pressurizer regulations, a more sophisticated module is available and must be selected. Such errors were minimized after having developed precise User’s Guidelines.

- **Physical process not predicted by the code**: physical mechanisms can be encountered which cannot be quantitatively predicted by system codes. For example, when CCFL occurs in a zone with a complex geometry the corresponding flooding correlation must be established first from experimental tests having the exact geometry before it is implemented in the code. A specific CCFL model is implemented in system codes where the equations are forced to find a prescribed flooding limit correlation given by the user on option for a particular node. Experimental data now exist for most important reactor components where CCFL is expected to occur and User’s Guidelines may recommend adequate flooding correlations.

- **Transients highly sensitive to a certain physical process**: The loop seal clearing in some small break transients is an example of possibly sensitive process. The formation of a liquid slug in the intermediate legs of a PWR is a common feature of many small break LOCA transients. This occurs in particular after the rupture of the natural circulation when the reflux condenser mode starts. The primary circuit steam is divided into two parts separated by two liquid slugs in loop seals and in the vessel bottom. These two subsystems are practically uncoupled from the energetic point of view. They follow a thermal equilibrium defined by a saturation temperature resulting from all the energy exchanges. In the hot subsystem - core, hot legs, SG tubes - the temperature level is generally controlled by the secondary side except if a break in this part is able to discharge more vapour than the core produces. In the cold subsystem the energy balance generally depends on the presence of a cold leg break and of safety injections, both of them giving a trend to a temperature decrease. Most of the time a depression of the cold part is induced which shifts water from core to downcomer. This dangerous situation stops when the liquid level in the descending part of the intermediate leg reaches the horizontal part. The loop seal clearing occurs either simultaneously or at different times in the different loops. After loop seal clearing the pressure in hot and cold legs are nearly equal again. When the process is slow it is very sensitive to small asymmetries between the loops. Once it happens in one loop a stable state may be reached preventing from its occurrence in the other loops. Codes have many difficulties to predict these non symmetrical behaviour correctly. It is a typical problem of predictability with small causes having big effects. Such situations were identified by the extensive application of system codes to integral tests.

### 8.3 Main Limitations of the Present System Codes

As illustrated in this paper, the accuracy of the various models is not uniform since more attention was paid to the most sensitive models in accidental transients.

A rather high degree of empiricism was necessary for some models reflecting a lack of understanding of all governing physical processes.

These two reasons make system code application to new transient situations or new geometry somewhat hazardous. New experimental programs with industrial configuration are still required for any new reactor design.

The intrinsic limitations of the two-fluid 6-equation model were reached in this generation of system code. Further progress would require additional equations, such as transport equations for interfacial area or for turbulent scales, or a multi-field modelling. These were the conclusions of several OECD-CSNI specialist meetings [3,4,5].

Another important limitation of present system codes is related to the coarse nodalization of reactor circuit required by CPU cost. The geometrical complexity is then very simplified and specific effects cannot be predicted.

### 8.4 Looking for more Predictive Tools

Progress of computer efficiency will allow to go beyond the limitations of present system codes in two ways:

- Use of more advanced modelling for reducing the degree of empiricism: multi-field models, turbulence modelling and dynamic interfacial area are the main directions for improving the predictive capability of the codes

- Refining the space resolution of simulations by a more systematic use of 3D modelling with finer meshing and by coupling with two-phase CFD.

New SET with advanced instrumentation are necessary to develop and validate the new models. Direct Numerical Simulation techniques can also be used as a complement to experiments to investigate flow phenomena at a very small scale. A multi-scale approach to two-phase reactor thermalhydraulics is planned for future [28,29].
8.5 Benefits of having Developed System Codes

The development of the present generation of system codes promoted an extensive SET & IET experimental program which provided a much better understanding of basic flow processes and an exhaustive knowledge of the phenomenology of reactor accidental transients. The most sensitive flow processes were clearly identified and the complexity was reduced to some safety relevant phenomena.

The scientific knowledge was extended and a good compromise was found between mechanistic and empirical approaches. An extensive code validation allowed improvement of predictions and later the evaluation of code uncertainty. The two main products of this effort are:

- An improved reliability of Safety Analyses
- The success of the challenge of Best-Estimate methodology for licensing

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>cross section area</td>
</tr>
<tr>
<td>B</td>
<td>buoyancy force</td>
</tr>
<tr>
<td>c</td>
<td>propagation velocity</td>
</tr>
<tr>
<td>D</td>
<td>drag force</td>
</tr>
<tr>
<td>F</td>
<td>force</td>
</tr>
<tr>
<td>g</td>
<td>gravity acceleration</td>
</tr>
<tr>
<td>G</td>
<td>mass flux</td>
</tr>
<tr>
<td>h</td>
<td>enthalpy</td>
</tr>
<tr>
<td>j</td>
<td>superficial velocity</td>
</tr>
<tr>
<td>p</td>
<td>pressure</td>
</tr>
<tr>
<td>Q</td>
<td>energy source</td>
</tr>
<tr>
<td>r</td>
<td>radius</td>
</tr>
<tr>
<td>T</td>
<td>stress tensor</td>
</tr>
<tr>
<td>t</td>
<td>time</td>
</tr>
<tr>
<td>u</td>
<td>internal energy</td>
</tr>
<tr>
<td>v</td>
<td>velocity</td>
</tr>
<tr>
<td>x</td>
<td>curvilinear abscissa</td>
</tr>
</tbody>
</table>

Greek symbols

- $\alpha$: volume fraction or void fraction
- $\beta$: added mass coefficient
- $\delta$: size or diameter
- $\Gamma$: interfacial mass transfer
- $\mu$: viscosity
- $\sigma$: surface tension
- $\theta$: angle
- $\rho$: density
- $\rho$: density

Subscripts-superscripts

- ext: external
- g: gas phase
- i: phase i
- int: interface
- l: liquid phase
- m: mixture
- vm: virtual mass
- wall: wall

ACKNOWLEDGMENTS

The author is grateful to the Programme Committee of THICKET-2004 for allowing this attempt to represent the community of the system code developers and users, and is fully indebted to all participants to the OECD-CSNI specialist meetings and to the members of the EUROFASTNET project who produced the material used in this synthesis.
REFERENCES


