YDINVOIMATEKNIIKAN LABORATORIO

TYÖRAPORTTI
TECHNICAL REPORT

Projekti/työ - Project identification	Sivuja - Pages	Päiväys - Date	Raportin nro - Report No.
HOHTI	26	18.12.1992	HOHTI-5/92

Otsikko ja tekijä - Title and author

APROS FIVE EQUATION THERMOHYDRAULICS
AFET

Jaakko Miettinen, Markku Hänninen

Hyväksynyt - Approved by

Julkisuus - Availability statement

Free

Jakelu - Distribution

Abstrakti, sisällysluettelo tms. - Abstract, list of contents etc.

This report documents the five equation model AFET of the computer program APROS intended for the simulation of nuclear and conventional power plants.

Valtion teknillinen tutkimuskeskus
Ydinvoimatekniikan laboratorio
PL 208 (Tekniikantie 4)
02151 Espoo

Technical Research Centre of Finland
Nuclear Engineering Laboratory
P.O.Box 208 (Tekniikantie 4)
SF-02151 Espoo, Finland

Tel.. +358 0 4561
Telefax +358 0 456 5000
Telex 122972 vttha sf

THE FIVE EQUATION THERMOHYDRAULICS MODEL AFET FOR THE APROS SIMULATION ENVIRONMENT

The five equation thermohydraulics model AFET describes two-phase flow using separate equations for the mass of mixture and vapour, for the energy of liquid and vapour and for the momentum of mixture. The phase velocities are solved using the drift flux model. For the containment calculation the conservation equations of the non-condensable air mass and energy are solved. This kind of solution method can handle small break LOCA (SBLOCA), large break LOCA (LBLOCA) events and all kinds of transients in the pressurized water reactor (PWR) and boiling water reactor (BWR).

The decision to develop the AFET model was based on the good experiences obtained from the fast-running code SMABRE developed at VTT after the Three Mile Island accident (1979). The five equation formulation has been shown to describe properly different size of LOCA events and different types of plant transients in all type of LWR plants.

The present AFET formulation has been originally developed as an integral part of the APROS simulation environment using the database and solution systems of APROS and it contains many extended features such as models for containment description. Thus the verification work done for SMABRE is not to be considered directly applicable to the AFET model.

1. GENERAL DESCRIPTION

1.1 Basic modelling principles

The five equation model includes field equations for the mass of mixture and vapour, for the energy of liquid and vapour and for the momentum of mixture. The mathematical modelling is based on the lumped parameter approach describing the geometry with nodes, junctions and heat structures. The definition having state variables (pressure, enthalpy, void fraction) in the node midpoints and flow variables (mass flow velocity) in junctions connecting nodes means the staggered mesh modelling principle, but applied to the real geometry. The original field equations are expressed in the discretised form as equations for the pressure distribution, volumetric flow distribution, void fraction transport, liquid enthalpy transport and vapour enthalpy transport and these equations are solved using the sparse matrix solution method. The tracking of non-condensable gas fraction (in-vessel) and boron in water is calculated separately using the solved mass flow field.

For containment application the vapour includes two gas fractions, condensable steam and non-condensable air. The components are handled like vapour in the original model, only the condensation characteristics are different. In the energy equation the steam cannot be subcooled, however, but for air all temperatures are possible.

The heat structures like pipe walls, vessel walls, fuel elements, steam generator tubes and heaters are significant heat sources and sinks for the TH system. The integration includes the calculation of structure temperatures based on the discretisation of heat conduction equations. In principle all different types of heat structures may described individually. The fast-running heat structure model is based on volume averaged temperatures through the structure. The heat conduction from the structure midpoint is considered during the definition of wall heat transfer, however. Optionally in essential sections a more detailed radial mesh structure may be used. In the fuel rods different averaged temperatures for the fuel and the cladding are defined even in the most fast-running approach.

The computation time and the accuracy of the calculated results depend on the number of fluid volumes and heat transfer structures and on the rate of change of the transient compared to the selected time step. The numerical method allows the use of a constant time step. For a coarse nodalisation the time step up to 1.5 s may be used, for a fine nodalisation the maximum stable time step 0.5 s may be recommended.

1.2 Constitutive correlations

The constitutive correlations of the TH model describe the physical relationships in the field equations. The most significant correlations are related to the heat transfer between fluid and structure, heat transfer between phases, phase separation and critical flow rate.

The heat transfer correlations for the fluid-structure interaction cover all possible heat transfer modes like single phase convection to liquid, nucleate boiling, transition boiling, film boiling, convection to droplet dispersed flow and convection to single phase vapour. As transition criteria the critical heat flux and Leidenfrost temperature are defined.

The interfacial heat transfer between vapour and liquid incudes the condensation caused by the subcooled liquid and cold surfaces as well as the flashing caused by the superheated liquid. Especially in the containment the condensation of the subcooled vapour is considered, too.

The phase separation caused mainly by gravitational forces is modelled using the drift flux formulation. The drift flux formulation is an efficient way for describing the gravitational separation in vertical flow channels and even the experimental results are most easily presented using this formulation. In the five equation model the drift flux separation is used even for the horizontal channels, but then a relatively short length for the channel is assumed. It is assumed that the model is not used for long pipelines.

Critical break flow modelling in the breaks and valves is based on the separate break flow model by Moody combined with contraction coefficients of the break geometry. From many calculations of test cases best proposals for contraction coefficients have been found, but the values are different for the subcooled liquid and two-phase mixture. In normal junctions the limitation caused by the critical flow rate may tested optionally.

The steam and water property package is derived as rational function fittings to the table data in the package.

1.3 Coupling to external systems and special components

The TH model is coupled to the external systems via different type of source contributions increasing fluid mass, decreasing fluid mass, adding heat, adding fluid momentum and affecting on the valve status. The assumption is that the TH model is used in a PWR for the whole primary loop and on the secondary side from the feedwater inlet nozzle to the turbine control valve. In a BWR the TH model is used for the vessel, steam lines and steam header.

The core TH model is typically coupled with an external core or neutronics model. In both cases the heat generated in different fuel heat structures is given as a heat input into the fuel pellet structure. The fast running fuel model calculates only the average fuel temperature and cladding temperature. For specific purposes even the maximum fuel temperature may be easily generated from these. Between the fuel pellet and cladding the gas gap conductance is calculated as well. For the external neutronics model the variables needed for the reactivity calculation may be picked directly (coolant temperature, void fraction, mixture density and fuel temperature).

Other heat inputs may be located in the pressurizer as heaters. This heating effect may be given as a thermal power directly to the heat structure describing heaters.

Any junction in the nodalisation may include a control valve or a stop valve. It is assumed that these valves are controlled externally. The pressurizer spray system is a

special type of line connected to the cold leg after reactor coolant pumps. This line with control valves may be defined in the nodalisation.

The accumulator may be described with normal nodes assuming vapour containing air. As a fast running alternative a special accumulator model may be used as well. The model assumes a invariable polytropic constant for the nitrogen expansion.

The reactor coolant pumps in a PWR and circulation pumps in a BWR accelerate the circulating coolant flow by adding momentum into the fluid. The jet-pump includes a special momentum interaction, where one flow stream affects on the another flow stream. All different pump alternatives may be described by the model. The pump model based on the two quadrant homologous curves for the head and torque. For two-phase conditions a two-phase multiplier is included.

In many reactor geometries the fluid circulates without any pumps like on the secondary side of steam generators. With proper loss coefficients the circulation results automatically from the nodalisation.

In large simulator systems the external injections like the feedwater, charging flow, high pressure injection and low pressure injection result from the calculation of a complicated pipe network. It is practical that this kind of systems are described with an external pipe network solver. Then usually one node from the five equation system is used as an external node for the flow calculation. The externally calculated flow rate can then directly be given as a mass, enthalpy and boron source. The source may be in a vapour form as well.

The flow rates out from the major primary and secondary systems consist typically the letdown as liquid and turbine (bypass) flow as vapour. Usually external systems have calculated these flow rates by using the nodes in the main systems as external nodes. This kind of flow rates may be directly given as an input for the model. The model includes an automatic readjustment of the flow rate, which takes into account the possible vapour content in the letdown node and liquid content in the turbine flow node.

For relief valves and safety valves the valve position is typically given based on the external control system. The thermohydraulic model calculates then the flow rate out by assuming that the pressure outside the valve is given. In an alternative coupling the valve loss coefficient may defined as a constant and the external pressure may be varied.

1.4 Assessment of the model

The assessment of the model is being performed internally by calculating e.g. international standard test problems. Quite large database with experimental data and input decks for RELAP5 and SMABRE exist already now from nearly all significant experiments performed during the last ten years. As a first step the five equation model will be assessed against these data, which is rather quickly available for testing.

2. SOLUTION OF CONSERVATION EQUATIONS

The basic thermodynamic and fluid dynamic equations are written in an Eulerian form /1/ as one-dimensional conservation equations. The mass conservation equations are solved for vapour and vapour-liquid mixture, the energy equations for vapour and liquid separately and the momentum equation for vapour-liquid mixture. The energy equations are expressed using enthalpies. The basic flow equations are one-dimensional. The discretised equation is based on the staggered mesh description of the fluid flow geometry. The state parameters like pressure (p), void fraction (α), density (ρ) and enthalpy (h) are averaged values for a node. The flow parameters like velocity (u) are defined in a junction.

2.1 Mass conservation equations

The formulation of the numerical solution for mass conservation equations is based on the consideration of phase conservation equations for vapour (steam) and liquid (water). For specific purposes the conservation of the non-condensable mass is considered as well. The vapour mass balance (includes (1-X) if non-condensables are described) in an Eulerian form by neglecting the diffusion term is written as

$$\frac{\partial}{\partial t}(A\alpha\rho_g) + \frac{\partial}{\partial z}(A\alpha\rho_g u_g) = A\gamma + As_g \qquad (2.1)$$

The equation for the liquid mass balance may be written as:

$$\frac{\partial}{\partial t}(A(1-\alpha)\rho_l) + \frac{\partial}{\partial z}(A(1-\alpha)\rho_l u_l) = -A\gamma + As_l \qquad (2.2)$$

The source term (s) means injections and leakages. The boiling term (γ) means evaporation and condensation.

The mass conservation for the non-condensable mass may be written as

$$\frac{\partial}{\partial t}(A\alpha\rho_g X_n) + \frac{\partial}{\partial z}(A\alpha\rho_g u_g X_n) = As_n \qquad (2.3)$$

The mixture equation is derived by summation of phase equations.

2.2 Momentum conservation equation

The gas momentum balance equation for a separated two phase flow is defined by

$$\frac{\partial}{\partial t}(A\alpha\rho_g u_g) + \frac{\partial}{\partial z}(A\alpha\rho_g u_g^2) =$$

$$- A\,\alpha\,(\frac{\partial p}{\partial z}) - A\,\alpha\,(\frac{\partial p}{\partial z})_e - A\,(\frac{\partial p}{\partial z})_{fg}$$

$$- A\,\alpha\,\rho_g g\cos\Theta + A\,(\frac{\partial p}{\partial z})_i \qquad (2.4)$$

The term inside the time derivative is equal to the gas flow rate. The term inside the space derivative expresses the changes of flow momentum along the integration axis. The first pressure derivative expresses the effect of the axial pressure distribution on the momentum conservation. The subscript "e" on the right hand side means an external pressure force, like circulation pumps. The subscript "f" refers to the wall friction and the last term on the gravitational field. The subscript i means the interfacial shear and it is included into initial formulas but it is vanishing when mixture mass conservation equations are derived.

The liquid momentum balance equation for a separated two phase flow is defined by

$$\frac{\partial}{\partial t}(A(1-\alpha)\rho_l u_l) + \frac{\partial}{\partial z}(A(1-\alpha)\rho_l u_l^2) =$$

$$- A\,(1-\alpha)\,(\frac{\partial p}{\partial z}) - A\,(1-\alpha)\,(\frac{\partial p}{\partial z})_e - A\,(\frac{\partial p}{\partial z})_{fl}$$

$$- A\,(1-\alpha)\,\rho_l g\cos\Theta - A\,(\frac{\partial p}{\partial z})_i \qquad (2.5)$$

The corresponding term like for the gas momentum equation may be found in this equation as well.

Summation of gas and liquid momentum equations gives the mixture momentum conservation equation as a result.

In the numerical solution the volumetric flow rates in the mass conservation equation are replaced by formulas derived from the momentum conservation equation. Finally only the equation for the pressure field remains. The matrix equation has a form

$$\underline{\underline{A}}\ \underline{p}^{n+1} = \underline{r} \qquad (2.6)$$

Here A means the matrix (sparse) to be inverted in the calculation, p means the pressure vector over all nodes and r means right hand side resulting from the formulations. This matrix equation is solved with numerical methods. When the new pressure field is known, new mixture flow rates in the volumetric form may be calculated from the momentum equation as

$$J^{n+1} = J^{n} + f(p^{n+1}) \qquad (2.7)$$

for all junctions between nodes.

2.5 Phase separation with the drift flux model

The drift flux model couples the superficial velocity of the gas (jg) to the superficial velocity of the mixture (j), void fraction in the junction (α), drift flux velocity (ugj) and distribution parameter (Co) by

$$j_g = \alpha\,(C_o\, j + u_{gj}) \qquad (2.8)$$

The distribution parameter Co is defined by the averaging procedures for the local gas velocity, void fraction, mixture velocity as

$$C = \langle \alpha\, j \rangle \,/\, \langle \alpha \rangle \langle j \rangle \qquad (2.9)$$

The expression within the angle brackets (<>) means a cross section averaging. The symbol j means superficial velocity.

After the volumetric flow distribution has been solved, the drift flux parameters are calculated from the constitutive correlations. Using updated separation parameters the new void fraction distribution is solved with sparse matrix solution methods from the equation

$$\underline{\underline{A}}\ \underline{\alpha}^{n+1} = \underline{r} \qquad (2.10)$$

where A means the inverted matrix, α void fraction vector and r right hand side vector resulting from the formulations.

After solving the new void fraction new volumetric flow rates for vapour are updated using the equation 2.8 and relationship between the superficial velocity and volumetric flow (J = j*A).

The gas mass flow rate is calculated from the volumetric flow by

$$W_g = J_g \rho_g^{n+1} \tag{2.11}$$

Typically the nodes are assumed homogeneous with respect to the phase separation, but for specific purposes s.c. separated nodes may defined. The phase separation condition in these nodes is described by the average void fraction (α) and the void fraction below water level.

2.3 Energy conservation equation

The enthalpy balance equation for liquid may be written as

$$\frac{\partial}{\partial t}((1-\alpha) A \rho_l h_l) + \frac{\partial}{\partial z} (W_l h_l) = A (1-\alpha) (dp/dt)$$

$$+ A q_{wl} + A q_{il} - A \Gamma h_s + A s_l h_{S,l} \tag{2.12}$$

The numerical solution may be conducted from this equation, because the enthalpy equation is solved independently after the flow rates, new pressures and updated heat flux rates have been solved. The principle of the matrix equation solved with sparse matrix system may be written as

$$\bar{\bar{A}} \bar{h}_l^{n+1} = \bar{r} \tag{2.13}$$

The enthalpy balance equation for gas may be written as

$$\frac{\partial}{\partial t} (\alpha A \rho_g h_g) + \frac{\partial}{\partial z} (W_g h_g) = A \alpha (dp/dt)$$

$$+ A q_{wg} + A q_{ig} + A \Gamma h_{gs} + A s_g h_{S,g} \tag{2.14}$$

Like for the liquid enthalpy equation, the numerical solution of the equation may be conducted from this equation and the principle formulation may be expressed similarly like in the equation 2.11. The equation is solved with the sparse matrix solver.

The heat flux rates on the right hand sides are defined separately for the heat transfer from wall to liquid, wall

to vapour and between phases. The heat transfer coefficient is typically defined via the transfer correlations and for convective heat transfer typically

$$q''_{wl,k} = h_{wl,k} (T_{w,k} - T_{l,i}) \quad (2.15)$$

$$q''_{wg,k} = h_{wg,k} (T_{w,k} - T_{g,i}) \quad (2.16)$$

For the boiling heat transfer typically (n = 0.9 ... 3)

$$q''_{wb,k} = h_{wb,k} (T_{w,k} - T_{s,i})^{n} \quad (2.17)$$

The interphasial flashing heat transfer is typically written

$$q_{li,i} = h_{li,i} \max(T_{l,i} - T_{s,i}, 0) \quad (2.18)$$

The interfacial condensation heat transfer is typically written

$$q_{gi,i} = h_{gi,i} \max(T_{s,i} - T_{l,i}, 0) \quad (2.19)$$

The enthalpy is related to the internal energy through the relationship

$$h = e + p/\rho \quad (2.20)$$

In the described formalism for the solved energy equation the effect of kinetic energy on the enthalpy has been neglected. This contribution is significant for the detailed calculation of the critical break flow. When critical flow is solved using a special model based on the stagnation pressures, the kinetic energy has a negligible effect elsewhere.

2.4 Mass conservation of boron in liquid and non-condensables in gas

The boron concentration is one example of substances following water. The conservation equation for the boron concentration is integrated using the transport equation

$$\frac{\partial}{\partial t}((1-\alpha)A\rho_l C_B) + \frac{\partial}{\partial z}(W_l C_B) = S_{l,i} C_{B,S} \quad (2.21)$$

The integration can be done with a simple method based on the donor cell discretisation of the equation and no sparse matrix solution is needed.

The conservation equation for the concentration on non-condensables in vapour (in-vessel) is integrated with the transport equation for the concentration of non-condensable fraction

$$\frac{\partial}{\partial t}(\alpha A \rho_g C_N) + \frac{\partial}{\partial z}(W_g C_N) = S_g C_{N,S} \qquad (2.21)$$

The integration may be done with a simple method based on the donor cell discretisation and no sparse matrix solution is needed.

For the containment the integration of non-condensable air mass was described earlier. In the enthalpy equation the existence of a massive non-condensable gas must be considered as well and the total gas enthalpy is defined by

$$h_g = (1 - X_n)\, h_n + X_n c_p T_g \qquad (2.22)$$

The gas density can be calculated by

$$\rho_g = \rho_s + \rho_n \qquad (2.23)$$

as a sum of steam partial density and non-condensable gas partial density.

The density of non-condensables can be calculated using, for instance, the ideal gas law

$$\rho_n = (p_n M_n) / (R\, T_g) \qquad (2.24)$$

Like for the gas density, the total pressure is the sum of partial pressures

$$p_{tot} = p_s + p_n \qquad (2.25)$$

The heat transfer to the wall must be calculated with the special correlation developed in the containment tests. Even in the heat and mass transfer between gas and liquid the effect of non-condensable gases must be taken into account.

2.6 Temperature equations for structures

The transient heat conduction equation for the radial conduction is solved in an integrated form for core fuel and steam generator wall structure. The method allows the computation of the radial temperature distribution described with key values, like fuel center temperature, gas gap temperature, cladding surface temperature, outer and inner steam exchanger tube temperature.

For the ordinary heat structure only average temperature is used. For thick wall structures a radial mesh may be defined, however.

One dimensional transient heat conduction equation in the slab geometry assuming heat conduction, material density and specific heat of the material constant in the structure is described by

$$(\rho c_p)\frac{\partial T}{\partial t} = \lambda \frac{\partial^2 T}{\partial y^2} + q''' \qquad (2.26)$$

By multiplying the both sides of the equation with the surface area A the equation is expressed in the more suitable form for the integration.

One dimensional transient heat conduction equation in the cylindrical geometry assuming material density and specific heat of the material constant in the structure is described by

$$(\rho c_p)\frac{\partial T}{\partial t} = \frac{1}{r}\frac{\partial}{\partial r}(\lambda r \frac{\partial T}{\partial r}) + q''' \qquad (2.27)$$

By multiplying the equation with the rod length and rod perimeter the form of equation becomes more suitable for the integration.

The slab geometry equation is practical for slab like of slightly curved slab geometries. The cylindrical geometry equation is needed for rods and small tubes, which cannot be approximated any more with a single slab geometry.

The new fuel and cladding temperatures can be solved from the conduction equations using a simple elimination process, because no axial heat conduction has been modelled.

2.7. Special component handling

2.7.1 Main coolant pumps

The pump head and hydraulic momentum are calculated from the numerical expressions as a function of the relative pump speed and volumetric flow

$$H = f (n, J) \tag{2.28}$$

$$T_{hy} = f (n, J) \tag{2.29}$$

$$n = \omega / \omega_{ref} \tag{2.30}$$

In the case of two-phase flow a specific two-phase multiplier is added to the head and torque formulations. For the stable pump speed the electrical momentum and hydraulic momentum are coupled by the difference caused by the frictional loss and pump inefficiency

$$T_{el} = T_{hydr} / (1 - C_0 - C_1 n - C n^2) \tag{2.31}$$

The coefficient C mean friction and inefficiency coefficients.

2.7.2 Accumulator

A single polytropic expansion law may be used for the accumulators

$$p \left(1 / \rho_{N_2}\right)^{\gamma} = \text{constant.} \tag{2.32}$$

2.8. Integration of conservation equations

During a single time step next solution procedures are performed:

1. The trip signals and their consequences to the reactor power, injection start, main coolant pump trip and valve positions are updated earlier.

2. Update mass and energy source terms. The mass source terms include the time dependent injection and leak, the valve leak, the break flow rate and the pressure dependent injection. The energy sources include the core power and the heater input.

3. Calculate the wall heat transfer and its derivative with respect to the wall temperature, the saturation temperature and the fluid temperature. For boiling calculate the mass transfer and the derivative with respect to the saturation temperature and wall temperature.

4. Calculate the interphasial heat and mass transfer and their derivative with respect to the saturation temperature.

5. Solve the pressure distribution using the sparse matrix method.

6. Calculate the main coolant pump characteristics. Integrate the pump speed.

7. Solve the volumetric flow rates in all junctions.

8. Update new heat transfer and boiling rates with new pressures. Update the leak flow rates with new pressures.

9. Integrate the new temperatures in heat structures. Update the fluid heating fluxes and boiling with new wall temperatures.

10. Find the phase separation velocity using the drift-flux model.

11. Solve the liquid and gas flow distributions for all junctions. For the prediction of the new void fraction use the sparse matrix solution procedure.

12. Using new flow rates and mass sources solve the new liquid and gas mass for all nodes. Calculate the void fraction from the new liquid and gas mass.

13. Solve new liquid enthalpies using the sparse matrix procedure.

14. Solve new gas enthalpies using the sparse matrix procedure.

15. Solve the boron concentrations.

16. Solve the concentration of non-condensables.

3. CONSTITUTIVE CORRELATIONS

3.1 Wall friction calculation

For wall shear and form losses the friction coefficients are calculated only once and during the transient turbulent flow conditions are assumed resulting in a constant wall friction coefficient. The two-phase multiplier is calculated from the local fluid condition during the transient. The turbulent wall friction correlation is based on the Blasius correlation and the homogeneous flow approach is used for the two-phase multiplier.

The wall friction in the fast-running formula is calculated from the expression

$$f_{wm} = (1 / D_e)\ 0.375\ Re^{0.25} \qquad (3.1)$$

where the Reynolds number is defined for the mixture.

The two-phase multiplier is defined by

$$\Phi^2 = \rho_l / \rho_m \qquad (3.3)$$

Using these the frictional pressure drop over a junction may be calculated from

$$\Delta p_j = 0.5\ \rho_m\ (f_{wl}\ \Delta z_j + X_{k,j})\ u_m\ |u_m|\ \Phi_j^2 \qquad (3.4)$$

3.2 Critical break flow rate

Initially four different library packages have been developed for the calculation of critical break flow rate. The same correlations are possible to be used in a limited amount of junctions inside the geometry. The present possibilities for the break flow calculation are:

a) a rational function fitted against 272 data points calculated with the Moody model in RELAP4/Mod6,

b) a rational function derived from data points calculated with the RELAP5/Mod1 correlation.

c) a fast running method based on the experiences with Marviken experiments.

d) a rational function fitting to the Henry-Fauske model.

The appropriate model can be activated with an option.

For subcooled and saturated conditions the user defines the contraction coefficients separately, typical values are 0.45 ... 0.60 for the saturated conditions and 0.8 ... 1.0 for the subcooled conditions, when the Moody model is selected. Both parameters are given in the input and the code calculates the final multiplier on the basis of the process data.

5.3 Interphasial heat and mass transfer

Interphasial heat transfer correlations are used for the flashing and condensation. In reality the time constant of the flashing process is in a class of few milliseconds. In the fast-running model the time constant in a class of a time step is used. For usual volumes the time constant of the condensation process is short, too, a slightly longer than that of the flashing process. A special condensation model is used for volumes, where the liquid level is defined separating the vapour and liquid.

For the superheated liquid the flashing heat transfer is calculated from a practical formula derived by simplifying from a more detailed model. The liquid cooling heat flux for a node is calculated using the flashing heat transfer correlation from the expression

$$Q_{flash,i} = h_{f,i} M_{l,i} (T_{l,i} - T_{s,i}) \qquad (3.5)$$

The coefficient h means flashing heat transfer coefficient. The simple model neglects the effect of the bubble growth. The liquid superheat used in the calculation is limited due to stability reasons.

The condensation model for homogeneous nodes is based on the analytical modelling of the heat conduction in a droplet having the diameter about 0.003 ... 0.02 cm. The heat conduction is assumed laminar. The fast-running expression is formulated as

$$Q_{cond,i} = h_{c,i} M_{l,i} \alpha_i \rho_{g,i}^{X} (T_{l,i} - T_{s,i}) \qquad (3.6)$$

The coefficient h means condensation heat transfer
The condensation heat flux is used for the liquid heating. The user can dial the correlation with the parameter $0 < X < 1.0$. The liquid subcooling is limited due to stability reasons.

For separated nodes the condensation through the water level is calculated from a model assuming a diffusion speed (vdiff) for the mixing below the water level. The fast running correlation is formulated as

$$Q_{cond,i,sep} = v_{diff} A_i \rho_{l,i} c_{pl,i} (1 - \alpha_i) X_{cond} \quad (3.7)$$

The typical rate for the diffusion speed in 0.001...0.1 m. Both for flashing and condensation the derivatives with respect to the saturation temperature and liquid temperature are needed during the solution process.

3.4 Wall heat transfer and boiling or condensation

The wall heat transfer package includes the correlation for all heat transfer regimes. Most of them are simplified formulas from more detailed correlations which due to several real exponents are too time consuming to be used in real time simulators.

The forced convection to single phase liquid is calculated with the Dittus-Boelter correlation

$$h_{c,k} = (\lambda_{l,i} / D_{e,i}) \, 0.028 \, Re_{m,i}^{0.8} \quad (3.8)$$

where the mixture Reynolds number is defined as

$$Re_{i,m} = W_{m,i} D_{e,i} / (A_i \mu_{l,i}) \quad (3.9)$$

The convective heat flux for the heat structure is calculated as

$$Q_{w,k} = A_k h_{c,k} (T_{w,k} - T_{l,i}) \quad (3.10)$$

The boiling of subcooled and saturated liquid is calculated with a fit to the Chen boiling correlation, Thom's correlation or a fast running correlation proposed by J. Rasmussen. The selected correlation may be selected optionally.

The boiling heat flux for the heat structure is calculated as

$$Q_{b,k} = A_k h_{b,k} (T_{w,k} - T_{s,k})^2 \quad (3.11)$$

The steam generation due to the boiling is calculated from

$$\Gamma_i = Q_{b,k} / h_{fg} \quad (3.12)$$

The maximum for the boiling and convective heat fluxes is defined by the critical heat flux correlation. The selected model is a a fit to the Griffith-Zuber correlation. An optional correlation a fit to the Biasi correlation may be used.

If the total wall heat flux exceeds the critical heat flux, the convective and boiling heat fluxes are limited so that the total heat flux is equal to the critical heat flux.

The film boiling heat transfer for the surfaces between the critical heat flux and minimum film boiling point is calculated from a fit to the Bromley correlation. The transition boiling between the critical heat flux point and the minimum film boiling point is added to the film boiling rate.

The single phase heat transfer to the superheated steam is calculated using the Dittus-Boelter correlation

$$h_{c,k} = (\lambda_{g,i} / D_{e,i}) \, 0.028 \, Re_{m,i} \qquad (3.13)$$

The correlation for the wall condensation is derived from the correlation proposed in the General Electric's Heat Transfer Data Book and the fast-running model has the following formulation

$$Q_{wcon,i} = A_k \, \alpha_i \, h_{wc} \, (T_{w,k} - T_{s,i}) \qquad (3.14)$$

A constant heat transfer coefficient given in the input is used for the radiation to the surroundings. It is assumed that the surroundings has the constant temperature.

The results of the Nordic heat transfer correlation study have been used for the definition of alternative correlations and for the development of the heat transfer logic. The alternative correlations are needed, when code is used for a detailed study of well instrumented experiments or when special correlations are available for a specific nuclear plant.

3.5 Drift flux correlation

The drift flux correlation is based on Zuber's bubble rise velocity, on Nicklin's results for large tube diameters and on Wallis type of correlation for the counter-current-flow-limitation (CCFL). An optional model is quite a complicated formula published by EPRI as a full range model for the vertical flow. This can be used only for some demonstrative calculation. A simplified version of this is useful for simulator application, too.

The drift flux velocity for the vertical flow is defined by the simple expressions

$$u_{gj,j} = v_{base} + (D_{e,i1} + D_{e,i2}) X_{mult} \quad (3.15)$$

The velocity vbase means the base velocity for small geometries Xmult the effect of geometry size. For CCFL limitation is modelled by the factor F which is defined as

$$F_j = (0, (1.000 - \alpha_j) / \alpha_{limit}, 1.0 > \quad (3.16)$$

For horizontal flow channels the drift flux parameters are defined based on the inclination of the water level as follows

$$\Theta_j = (2 D_j(\alpha_{i1(j)} - \alpha_{i2(j)}) / (z_{i1(j)} + z_{i2(j)}) \quad (3.17)$$

In addition to the drift flux velocity the distribution parameter C is defined as well.

3.6. Material properties

The material properties calculated by the code include the following:

Liquid density as a function of liquid enthalpy (in the accurate version enthalpy and pressure).

Vapour density as a function of integrated pressure and vapour temperature (vapour enthalpy and pressure).

Saturation temperature as a function of pressure.

Liquid and vapour viscosities as a function of temperature (enthalpy and pressure).

Liquid and vapour conductivity as a function of temperature.

Liquid and vapour temperatures as a function of enthalpy (enthalpy and pressure).

4. REPRESENTATION OF REACTOR GEOMETRY

In Figure 1 a nodalisation for a generic VVER-440 PWR is presented. The nodalisation scheme is chosen depending on the phenomena of interest. Typically, some details are essential for the transient, but large parts of the system may be described in a simple way. For the simulator application the nodalisation has to be a compromise between the accuracy of the calculation and the computer resources available. It is very advisable to build the simulator model with few alternatives.

In a small or medium break transient the liquid level build-up in loop seals, upper plenum or core, downcomer and pump inlet contribute strongly to the transient and the phase separation needs a careful description. A small bypass connection between upper plenum and downcomer may totally change the nature of the natural circulation process. The liquid level height in the steam generator primary side is significant, if a steam binding effect caused by the high secondary temperature is possible. During the reflux cooling the model has to be capable to calculate the CCFL for liquid downflow. During single phase natural circulation the primary nodalisation should be fine enough, to allow seeing the effect of a variable water density on the circulation flow rate. In the case of a non-symmetric coolant injection the circulation loops behave in different ways. The loop seals may be water filled during primary loop voiding and then the model has to be able to predict the decreased natural circulation rate.

Usually the thermohydraulic phenomena encountered in tests are so complex, that the capability of the code to predict the transient must be discussed separately for each test case.

The features included into the reference nodalisation will be discussed closer here.

For the VVER-440 plant simulation (see nodalisation, Fig 1.) as for any other PWR plant each primary loop is presented separately because of non-symmetric injection connections, steam generator operations and pump characteristics. The loop itself includes several nodes for description of temperature transport phenomena around the loop, for presentation of local temperature and pressure measurement and especially in the case of LOCA accidents for proper modelling of the void fraction distribution around the loop. In a VVER-440 plant the loop geometry is very significant due to loop seals in both legs, hot and cold. In this nodalisation the number of nodes in loop seals is quite small, but the nodalisation describes well the loop seal phenomena, because special

separable node types have been used in these sections. In a PWR with vertical steam generators typically the hot leg is horizontal and only the cold leg includes loop seals. Even in this case the similar modelling principles may be used.

The vessel model includes only two nodes in the downcomer section and two nodes in the lower plenum. For downcomer the application of the separable node type allows the coarse nodalisation. In fact in this case the steam flow between different cold leg loops will be described in the best way during a LOCA accident.

The core is modelled with five nodes for the average core. Five nodes is enough for the demonstrative simulation. For a more precise calculation more axial nodes may be used in the core. The core may include even several flow channels in the radial direction. This is essential specially in the BWR core with isolated flow channels. In some reactor constructions the core by-pass may be significant, and this feature must be described with an additional parallel node.

The upper plenum is described with three nodes and due to limited node number the node at the hot leg elevation includes a special separation model. The upper plenum description describes realistically the void fraction separation above the core, but the correct dynamic behaviour for the water temperature requires the use of a specific mixing model for the upper plenum.

The pressurizer is described using a single separated node. The model is sufficient to describe all significant subprocesses in the pressurizer like condensation on the water level, bubbling during the pressure decrease, steam superheating, condensation caused by the pressurizer spray, pressure increase caused by the heaters and effect of structures on the pressurizer dynamics.

For the steam generators only a single node for the tubing is shown. The description is possible if a specific enthalpy transport model is used for the tubing. In the generic simulator about three nodes for the steam generator primary side may be needed. Even for a U-tube steam generator three nodes for the tubing is a minimum, one describing ascending tubes, one descending tubes and one U-bend at the top. For a U-tube steam generator the geometry is very significant during a LOCA.

In Fig. 1 no nodalisation for the secondary side in shown. For horizontal steam generators the minimum nodalisation describes the liquid volumes to the separators with one node, steam section above separators with one node, each steam line with one node and steam collector with one common node. With this nodalisation a node with a special separation option must be used for the water filled part. For U-tube steam generators the riser, downcomer and steam

separator are typically described with separate nodes allowing the two-phase circulation on the secondary side.

The heat structures are not shown in the drawing. All significant heat structures are included into the nodalisation like pipe walls, thick walls in the vessel, thin walls in the vessel, fuel in the core, other internals in the core, pressurizer wall, pressurizer heaters, steam generator tubing, structural parts in the steam generator and walls in the steam line.

The reactor coolant pumps are modelled in each loop with pump characteristics. Heat generated in the pumps is added into the node just after the pump.

The heating power in the fuel elements is added into the fuel pellets of the core heat structure. In the pressurizer the heater effects are described as well. In the primary loop all significant injections and letdown flows are related to the nodes best presenting the real geometry. The injection include the charging flow, high pressure injection and low pressure injection. The hydroaccumulators are even coupled to the related nodes.

On secondary side the injections include the feedwater and emergency feedwater.

For pressurizer the PORV and safety valves are located to the top of the vessel.

On the secondary side the safety valves are located in each steam generator. The steam header includes a relief valve. The turbine bypass and turbine flow itself are described using a given valve position and external pressure outside the valve.

The main steam isolation valves are located in the junctions connecting the steam line nodes to the steam header and the valve positions are controlled externally. The spray line from the cold leg to the pressurizer top is modelled with a special junction coupled with an externally controlled valve. Then spray line takes into account the effect of operating pumps on the spray flow rate.

The thermohydraulic data base includes all necessary variables for modelling the measurements in instruments like pressure around the loop, local temperatures, flow rates and water levels. It even includes all necessary variables for the coupling to external systems, like local temperatures, local enthalpies, boron concentration, non-condensables, flow rates in valves. For the code the injections and leakages may be given externally as well.

REFERENCES *)

/ 1/ Ishii M., Thermo-Fluid Dynamic Theory of Two-Phase Flow, Eyrolles, 1975.

/ 2/ Wallis G. B., One-dimensional Two-phase Flow, McGraw-Hill Book Company, 1969.

/ 3/ Collier j.g., Convective Boiling and Condensation, McGraw-Hill Book Company, 1972.

/ 4/ Coarse material of Bases of Two-Phase Flow, Swiss Federal Institute of Technology, 1984.

/ 5/ Rathman O. et.al., "Computer Codes for Small-Break Loss-of-Coolant Accidents", A Nordic Assessment, Final report of the NKA project SÄK-3, Nord report, Riso, Denmark, September 1985.

/ 6/ Abel-Larsen H. et.al., "Heat Transfer Correlations in Nuclear Reactor Safety Calculations", Final report of the NKA project SÄK-5, Nord report, RISO-M-2504, Riso, Denmark, June 1985, Volumes I and II.

*) References are not referred in the text

NOMENCLATURE

A	flow area m^2
α	void fraction, cross sectional or volume average density kg/m^3
u	velocity m/s, phase average quantity
f	friction force N/m^3
F	friction force into the calculation cell N
j	superficial velocity m/s, local volumetric liquid or gas flux
J	volumetric flow in a junction m^3/s
V	volume m^3
γ	steam generation rate $kg/m^3/s$
Γ	steam generation rate in a node kg/s
s	mass source $kg/m^3/s$
S	mass source into a node kg/s
W	mass flow kg/s
f	friction N/m^3
λ	heat conduction W/m/K
g	gravity constant (9,81 m/s^2)
Θ	inclination from vertical direction
h	enthalpy J/kg, phase average quantity
e	internal energy J/kg
p	pressure N/m^2
q	heat source W/m^3
Q	heat input W
m	total mass kg
M	integrated momentum
t	time s
z	axial coordinate m
δt	time step, s
δ	Kronecker's delta
∂	derivative
C_o	drift flux factor (1...1.3)
T	temperature K,C
X	fraction of non-condensables

Subscripts:

g	steam, gas
l	water, liquid
m	mixture of steam and water
r	relative
gj	drift-flux (empirical model)
w	wall
i	node
j	junction
k	heat structure

Superscripts:

n	old time step
n+1	new time step

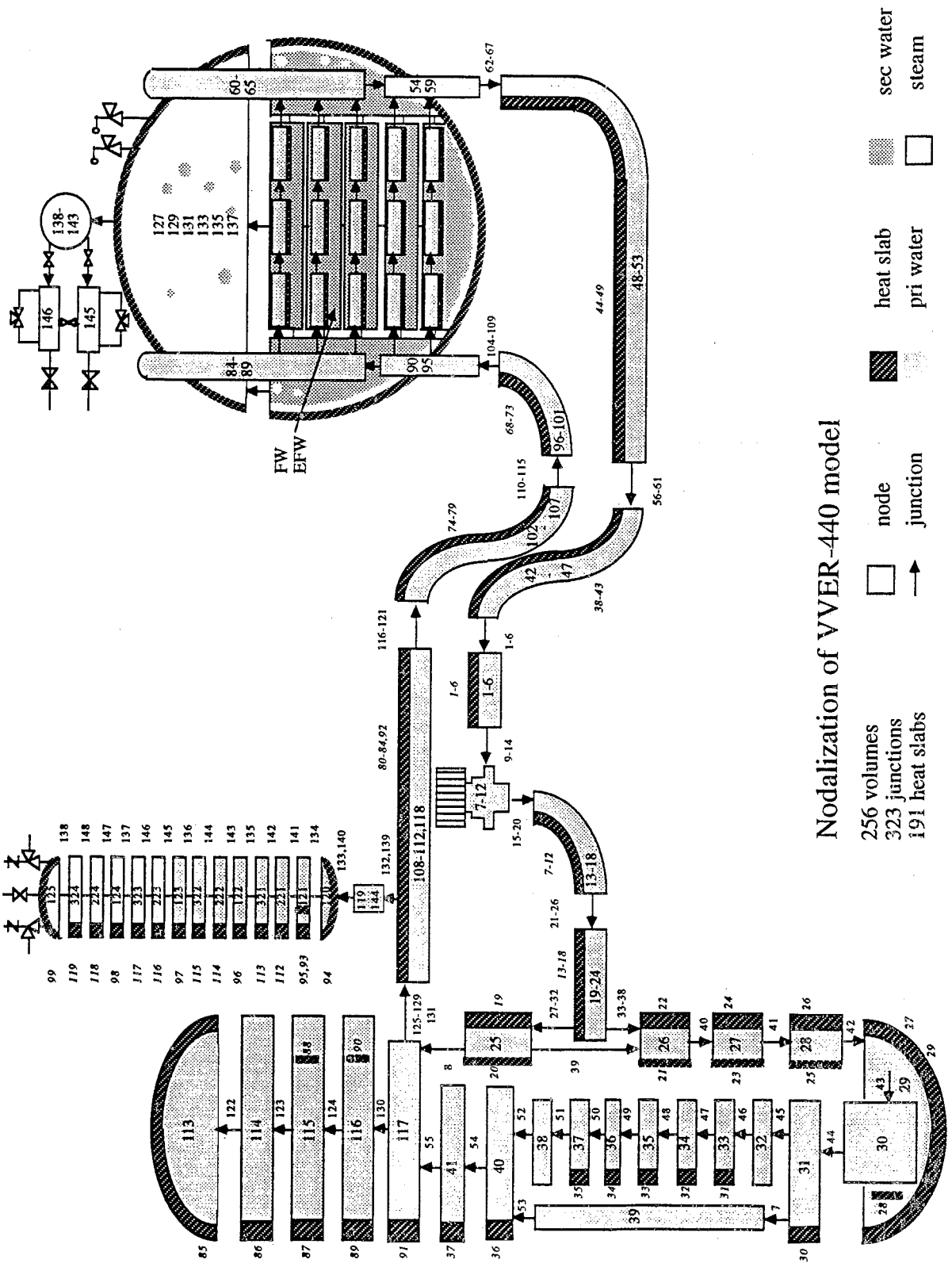

Figure 1. A nodalisation scheme for the generic VVER-440 plant