

Simulation and analysis of pipe and vessel blowdown phenomena using RELAP5 and TRACE

Sebastian Gurgacz*, Michał Pawluczyk, Grzegorz Niewiński, Piotr Mazgaj

*Institute of Heat Engineering, Warsaw University of Technology
21/25 Nowowiejska Street, 00-665 Warsaw, Poland*

Abstract

This paper presents the results of a pipe and vessel theoretical blowdown analysis in order to identify and assess rapid depressurization phenomena occurring during a Loss of Coolant Accident in a nuclear power plant. Calculations were performed in conditions similar to those in a reactor pressure vessel. Due to critical two-phase flow occurrence, RELAP5 mod 3.3. and TRACE v 3.0 system codes were used for this purpose and a computational model was built on the basis of available literature and implemented in MATLAB[®] code. Finally, pipe depressurization results were compared to experimental results taken from the literature.

Keywords: Two-phase flow, Two-phase flow simulation, Nuclear engineering

1. Introduction

The critical flow phenomenon has been extensively studied in single-phase and two-phase systems. It plays an important role in the design of two-phase power plant bypass systems with steam turbines, venting valves in chemical and power industries as well as in nuclear power plant safety calculations. Critical flow conditions during a Loss of Coolant Accident, part of what is called a Design Basis Accident, have motivated theoretical and experimental studies of two-phase critical flows. The aim of this article is to present the differences in modeling two-phase flow between system codes such as RELAP5, TRACE and analytical models implemented

by the authors. In order to compare different approaches, experimental data has been presented in the form of the Edwards pipe experiment and a theoretical model of a Reactor Pressure Vessel (RPV).

2. Two-phase critical flow phenomenon

There are various two-phase flow models, which can generally be divided into two types [1]:

- Equilibrium models (homogenous and non-homogenous)
- Non-equilibrium models.

The main assumption in the homogenous equilibrium model is that both gas and liquid are in thermodynamic equilibrium, which implies that their pressures and temperatures are equal. The mixture is described as one fluid with equal velocities of both phases. The flow is assumed to be steady and isentropic. This model is used only when high pressures and large mass-flow rates occur.

*Corresponding author

Email addresses:

sebastian.gurgacz@itc.pw.edu.pl (Sebastian Gurgacz*), michal.pawluczyk@itc.pw.edu.pl (Michał Pawluczyk), grzegorz.niewinski@itc.pw.edu.pl (Grzegorz Niewiński), piotr.mazgaj@itc.pw.edu.pl (Piotr Mazgaj)

Non-homogenous equilibrium models are used when relatively low pressures and mass-flow rates are involved. What diversifies the non-homogenous model from the homogenous model is the difference in velocity for each phase [1]. Flow is assumed to be adiabatic.

The non-equilibrium model does not assume thermodynamic equilibrium between phases. Each phase is described by three independent conservation equations: mass, energy and momentum. Such a model is used to describe the quickly changing flow parameters and when the non-equilibrium effects may play a significant role in the flow, for example, when the pipe length to diameter ratio is in the range of 3 to 12.

There are two distinct groups of non-equilibrium models:

- frozen models
- non-homogeneous, non-equilibrium models.

In frozen models the velocity ratio of phases is used. No heat and mass transfer between phases is assumed. Frozen flow is characterized by its constant quality throughout the expansion [1]. Transformations of vapor and liquid fractions are independent.

In non-homogeneous, non-equilibrium models there are no assumptions regarding the relationship between temperature and pressure of phases. Most of the models are empirically developed instead of growing out of theoretical considerations.

The critical flow rate is the maximum rate that can be reached by a compressible fluid during its flow from a volume at high pressure to a volume at low pressure. This low pressure is called the critical pressure (determined experimentally) and the velocity of the fluid embarking on the low pressure volume is equal to the speed of sound in those conditions. Such a flow is called a critical flow. The change of parameters in the pipe connecting both volumes during the flow or critical flow are shown in Fig. 1. One should ensure that the flow velocity is kept constant if pressure is higher than critical pressure; if not the flow velocity changes [2]. This phenomena can be observed in both single-phase and multiphase flows, although one needs to use various calculation models due to the differences in any single component properties against mixtures [2].

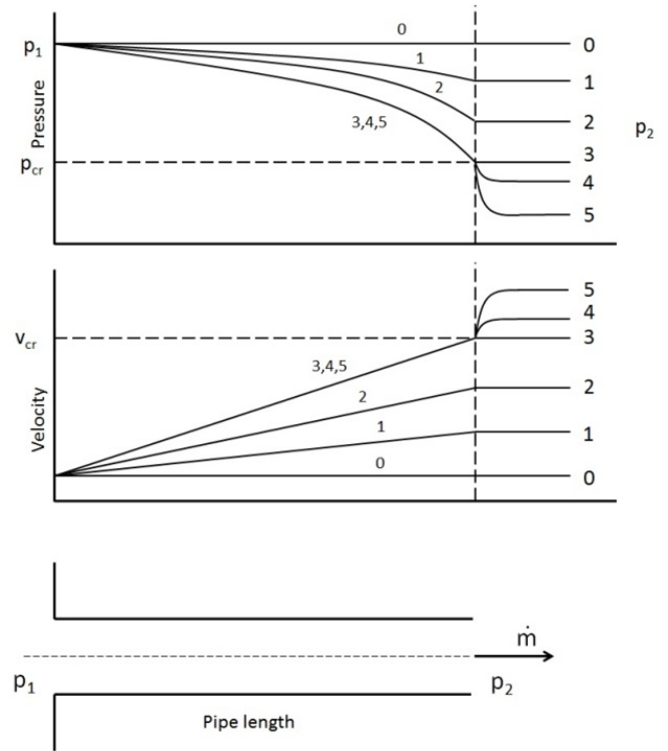


Figure 1: Pressure and velocity changes during critical flow

Two-phase critical flow is much more complicated in that the void present in the two-phase mixture influences the sonic velocity. Velocity may be estimated at the wave front, where no instantaneous phase change can take place. Critical flow rate, however, is determined by the sonic velocity behind the wave front - just where the phase change takes place. One needs to take into account sonic velocity and critical flows of two-phase mixtures independently.

The response of a liquid-vapor mixture to a pressure pulse is limited either by the mass transfer between phases in thermodynamic equilibrium or a frozen state with no mass transfer and both liquid and vapor being independently isentropic. Based on the premise that no mass transfer occurs, the estimated velocity is called “frozen velocity”. The computed velocity is considerably lower than frozen sonic velocity and the velocity detected at low and medium pressures.

3. Edwards pipe blowdown model

The standard test case for thermal-hydraulic calculation codes is the Edwards pipe blowdown problem.

Table 1: Initial conditions and pipe dimensions

Parameter	Value
Pipe length, m	4.10
Pipe inside diameter, m	0.0730
Flow area inside the pipe, m ²	0.00419
Flow area at the outflow, m ²	0.00364
Flow area at the outflow, MPa	7.00
Initial water pressure, K	502
Initial water temperature, m	4.10

This experiment was performed in the 1970s. Originally, a pipe with a high ratio of length to internal diameter was filled with water, under increased pressure conditions, and heated. The process was initiated by a rupture of the glass disc at the end of the pipe. The flow area was about 13% smaller than the flow area inside the pipe as a result of the inaccurate disk rupture. The data from the depressurization process was collected and used to develop the hydrodynamic model of the RELAP5 code. The size of the pipe and the initial conditions are presented below in Table 1.



Figure 2: Edward's pipe nodalization scheme: (1) Pipe, (2) Single Junction, (3) Time-dependent volume

The Edward's pipe model nodalization scheme in RELAP5 is presented in Fig. 2. Pipe (1) is divided into twenty equal control volumes. Initial conditions in each control volume are the same as in Table 1. The single junction (2) represents the reduced outflow area. The time-dependent volume (3) is used as a boundary condition with the fixed values of pressure and temperature, equal to the atmospheric conditions. The same approach was used while modeling the pipe in TRACE code. Both RELAP5 and TRACE models were prepared using SNAP software (Symbolic Nuclear Analysis Package). The maximum timestep used in calculations was 10^{-3} s while the minimum was set at 10^{-7} s. The calculated results are compared with experimental data, taken from [3] in the next section.

4. Pressure vessel blowdown model

A more realistic but still simplified simulation of the Loss of Coolant Accident would be the vessel blowdown phenomenon. A pressure vessel is inside the containment, where the pressure changes according to the water discharge flow. Initial pressure and temperature in the pressure vessel are similar to those in a typical PWR vessel in steady-state conditions. Water is under pressure of 15.5 MPa and is at a temperature below the saturation point. The containment is filled with air at a temperature of 300 K.

Table 2: Initial conditions, pressure vessel and containment dimensions

Parameter	Value
Pressure vessel inside diameter, m	5.00
Pressure vessel height, m	15.0
Pressure vessel volume, m ³	295
Initial pressure in the pressure vessel, MPa	15.5
Initial temperature in the pressure vessel, K	583
Outlet nozzle length, m	0.30
Outlet nozzle inside diameter, m	0.71
Containment inside diameter, m	50.5
Containment height, m	50.0
Containment volume, m ³	100,000
Initial pressure in the containment, MPa	0.10
Initial temperature in the containment, K	300
Pressure vessel inside diameter, m	5.00

All essential data regarding water conditions, pressure vessel and containment dimensions were adopted from the Pre-Construction Safety Report of the European Pressurized Reactor EPR [4]. Table 2 presents initial conditions in the pressure vessel and containment, and gives their dimensions.

The simulation was performed using the two calculation codes mentioned in the previous section and an additional analytical model set up by the authors. Moreover, pressure change in the containment was taken into account. To estimate pressure change in the containment with the authors' analytical model, the "piston" model described in [5] was used.

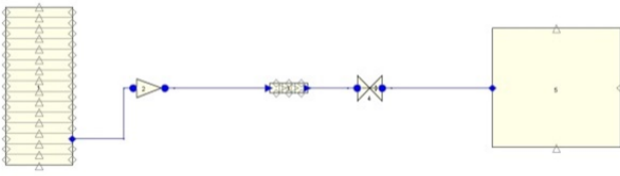


Figure 3: Pressure vessel, outlet nozzle and containment nodalization scheme: (1) Pressure vessel, (2) Single junction, (3) Pipe, (4) Single junction, (5) Containment

The nodalization scheme of the model used to simulate vessel blowdown in RELAP5 is presented in Fig. 3. The pipe (1) representing the vessel is divided into sixteen equal control volumes. Initial conditions in each control volume are the same as in Table 2. The single junction (2) represents a connection between the vessel and outlet nozzle (3). The single volume (5) is initially full of air under atmospheric conditions. The same approach was used in modeling the pipe in TRACE. Both RELAP5 and TRACE models were prepared using SNAP. The maximum timestep used in all calculations was 10–2 s while the minimum was 10–7 s. The calculated results are compared to the developed analytical model.

The thermal-hydraulic model incorporates conservation equations that need to be solved in order to determine the outflow from the reactor pressure vessel. In this analysis, under consideration are conditions and equations describing the vessel. The primary equation to be solved is the mass conservation equation of the reactor. It has been formulated for only one mass outflow (nomenclature from Moody [6]):

$$\frac{dM}{dt} = -w_{out} + w_{in} \quad (1)$$

In the first approximation inflow sources are neglected (e.g. inflows from emergency cooling systems).

$$\frac{dM}{dt} = -w_{out} \quad (2)$$

The next equation is the energy conservation equation describing the First Law of Thermodynamics, formulated in compliance with Moody's approach in [6] and Todreas [2].

$$\frac{dE}{dt} = q_{in} - q_{out} + (w_0 h_0)_{in} - (w_0 h_0)_{out} + \dot{W}_{kin} - \dot{W}_{kout} \quad (3)$$

Omitting work put in and done by the system as well as heat removed and added (e.g. decay heat), an equation is obtained which takes into account only energy lost by the moving liquid:

$$\frac{dU}{dt} \cong \frac{dE}{dt} = -(w_0 h_0)_{out} \quad (4)$$

This simplifying assumption is justifiable for a simple analysis of the first moments of the accident, when the outflow is isentropic and both decay heat and heat lost through the vessel walls can be neglected. The energy— E can be approximated by internal energy U by omitting the potential and kinetic energy.

$$E \cong U = uM \quad (5)$$

To finalize the model, an equation is needed allowing for the pressure change in time to be calculated. The pressure of the system is dependent on two thermodynamic parameters, which in Moody's [6] formulation are the specific internal energy and specific volume.

$$p = p(u, v) = p\left(\frac{U}{M}, \frac{V}{M}\right) \quad (6)$$

Differentiating the equation (3) in time, adding the conservation equations (1), (2) and the approximation (4), an equation for pressure change can be derived:

$$\begin{aligned} \frac{dp}{dt} = & \left(\frac{\partial p}{\partial u}\right)_v \frac{1}{M} [w_{in} (h_{0in} - u) \\ & - w_{out} (h_{0out} - u) + q_{in} - q_{out} + \dot{W}_{kin} - \dot{W}_{kout}] \\ & + \left(\frac{\partial p}{\partial v}\right)_u \frac{1}{M} \left[\frac{dV}{dt} - v(w_{in} - w_{out})\right] \end{aligned} \quad (7)$$

Taking into consideration the assumptions made for this model and acknowledging the fact that the vessel's volume does not change, the pressure change with time can be written:

$$\begin{aligned} \frac{dp}{dt} = & \left(\frac{\partial p}{\partial u}\right)_v \frac{1}{M} [-w_{out} (h_{0out} - u)] \\ & + \left(\frac{\partial p}{\partial v}\right)_u \frac{1}{M} [-v(-w_{out})] \end{aligned} \quad (8)$$

The practical integrating of equation (8) requires removing the pressure differentials of specific energy and specific volume. The practical approach by Moody [6] consists of enthalpies (from the enthalpy definition of $h = u + pv$):

$$\left(\frac{\partial p}{\partial u}\right)_v = \frac{1}{\left[\left(\frac{\partial h}{\partial p}\right)_v - v\right]} \quad (9)$$

and

$$\left(\frac{\partial p}{\partial v}\right)_u = -\frac{\left[\left(\frac{\partial h}{\partial v}\right)_p - p\right]}{\left[\left(\frac{\partial h}{\partial p}\right)_v - v\right]} \quad (10)$$

Removing the aforementioned derivatives needs using dependencies from general thermodynamics:

$$\left(\frac{\partial h}{\partial p}\right)_v = v + \frac{c_p v}{\beta_v c^2} \quad (11)$$

and

$$\left(\frac{\partial h}{\partial v}\right)_p = \frac{c_p}{\beta_v v} \quad (12)$$

For subcooled liquid:

$$\left(\frac{\partial p}{\partial u}\right)_v = \frac{\beta_v c^2}{v c_p} \quad (13)$$

and

$$\left(\frac{\partial p}{\partial v}\right)_u = -\frac{c^2}{v^2} + \frac{p \beta_v c^2}{c_p \vartheta} \quad (14)$$

where $\beta_v = \frac{1}{v} \left(\frac{\partial v}{\partial T}\right)_p$ is the volumetric expansion coefficient, $c_p = T \left(\frac{\partial s}{\partial T}\right)_p$ specific heat for constant pressure, $c^2 = \left(\frac{\partial p}{\partial \rho}\right)_s$ the speed of sound.

For equilibrium two-phase flow:

The main approach remains the same. This particular model was chosen because it can derive pressure as a function of time for each step for the two-phase mixture. Non-equilibrium models are rarely considered in analytical approaches (contrary to system codes), hence the choice of equilibrium two-phase flow.

Mass and energy conservation equations can be written as (15) and (16):

$$w_{out} - w_{in} + \frac{dM}{dt} = 0 \quad (15)$$

$$w_{out} h_{0,out} - w_{in} h_{0,in} + q_{out} - q_{in} + \frac{dU}{dt} = 0 \quad (16)$$

while the state equations are described by (17)

$$V = Mv; U = M\mu; \mu = \mu(p, \vartheta) \quad (17)$$

A different approach (from the subcooled flow) was derived for two-phase pressure change according to F.J. Moody's [6] formulation. Equation (18) denotes the pressure change in the vessel; it includes two functions $f(p)$ and $F(p, V/M)$ that are specific for different fluids.

$$\frac{dp}{dt} = \frac{w_{in}(h_{0,in} - f_p) - w_{out}(h_{0,out} - f_p) + q_{out}}{MF\left(p, \frac{V}{M}\right)} = 0 \quad (18)$$

For an equilibrium two-phase liquid vapor mixture with state equation given by:

$$\mu = \mu_f(p) + \frac{\mu_{fg}(p)}{\vartheta_{fg}(p)} [\vartheta - \vartheta_f(p)] \quad (19)$$

The functions are as follows:

$$f(p) = \mu(p) - \vartheta_f(p) \frac{\mu_{fg}(p)}{\vartheta_{fg}(p)} \quad (20)$$

and

$$F\left(p, \frac{V}{M}\right) = \mu'_f - \left(\vartheta_f \frac{\mu_{fg}}{\vartheta_{fg}}\right)' + \frac{V}{M} \left(\frac{\mu_{fg}}{\vartheta_{fg}}\right)' \quad (21)$$

where (') denotes $d()/dp$.

Equations (7) to (21) are sufficient to describe the pressure change in the vessel with time. They are only valid for a liquid-vapor mixture.

Adding the critical mass flux and pressure change models for both subcooled and two-phase flow models generates a full simplified representation of the blowdown phenomenon and can be used to calculate the depressurization time and represent its course as well as the discharge mass flow rate determining loss of coolant from the system.

Fig. 4 is a flow chart that describes the main idea behind the iteration scheme created for the blowdown analysis.

The algorithm programmed in MATLAB utilizes the above mentioned methodology.

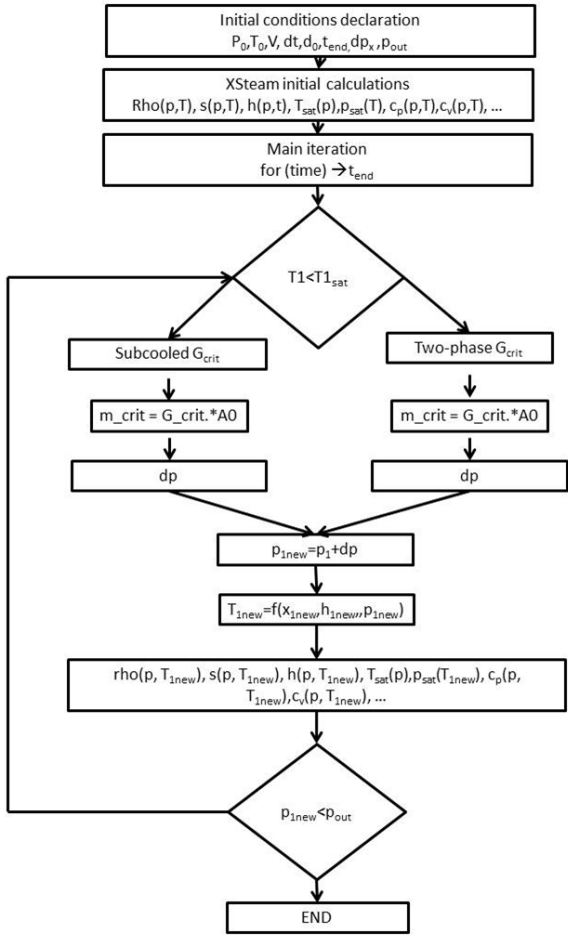


Figure 4: Iteration scheme flow chart

5. Results

There are two different events during the Edwards pipe blowdown: rapid depressurization due to single-phase discharge of water and the slower decrease as a result of the two-phase mixture discharge. The first one is almost instantaneous, approximately 2 ms. The second one continues until the pressure in the pipe is equal to ambient pressure.

Fig. 5 illustrates the pressure change measured during the experiment and the values calculated in RELAP5 and TRACE codes just after the initiation of the discharge. A large decrease can be seen at the start of the simulation, which is a result of the discharge of subcooled water of relatively high density, until saturation conditions are reached. The pressure decrease during the two-phase mixture discharge is compensated by steam generation, which causes an increase in the specific volume of the mix-

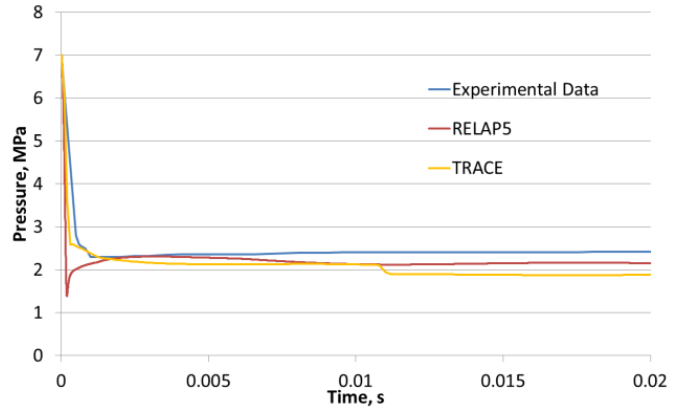


Figure 5: Short time pressure change inside Edwards pipe

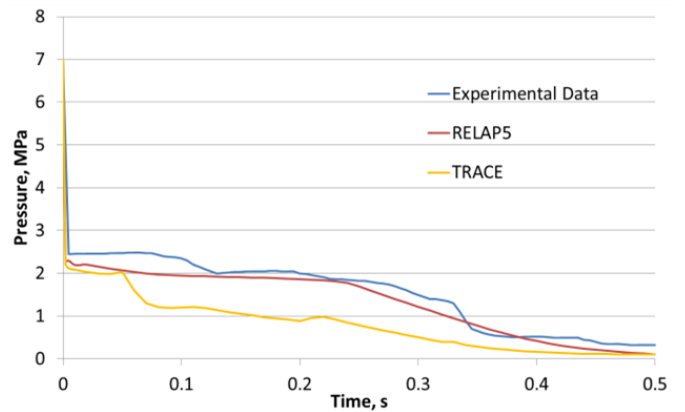


Figure 6: Long time pressure change inside Edwards pipe

ture and reduces the mass flow. In Fig. 6 the pressure change showing the full discharge phenomenon is illustrated.

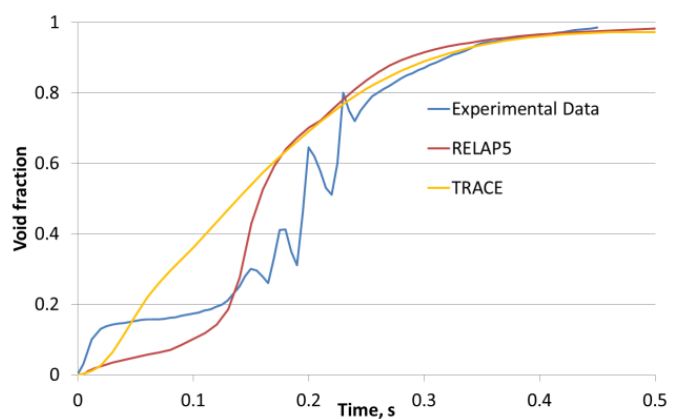


Figure 7: Void fraction change in the middle of the pipe

The void fraction calculated in the middle of the pipe is shown in Fig. 7. In the first milliseconds steam is generated relatively slowly. The largest increase in

the void fraction occurs in the middle of the simulation as a result of reaching saturation conditions and decreasing pressure. In the next stage, the generation rate decreases due to reaching equilibrium conditions with the environment.

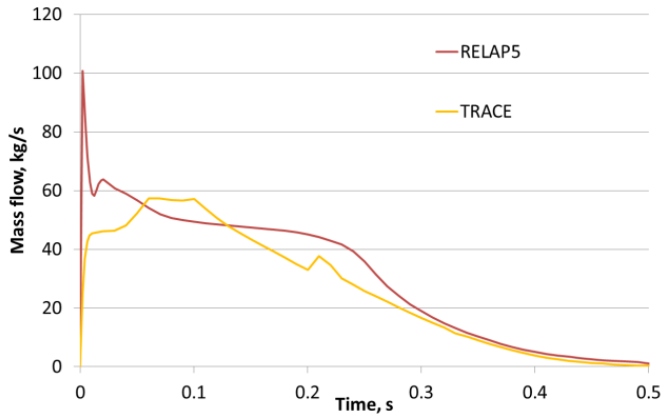


Figure 8: Mass flow rate change

The calculated mass flow is showed in Fig. 8. Its maximum value is reached at the start of the simulation due to the single-phase flow. Alongside the increase in void fraction, the density of the mixture decreases as does the mass flow from the pipe. Due to the lack of detail about the mass flow rate from the Edwards experiment it is not possible to compare the calculated results with the experimental data.

Apart from the test case for the thermal-hydraulic codes, which is the primary role of the Edwards pipe blowdown experiment, it is a testing case which, in a simple way, explains the physics of a Loss of Coolant Accident.

The simulation results are presented in graphs which show pressure change in the pressure vessel and the containment, void fraction in the vessel and mass flow rate of the escaping water. The simulation time was 40 seconds from the start of the discharge. Fig. 9 illustrates the pressure change in the pressure vessel. Rapid depressurization continues until the water falls under the saturation conditions. Then, over a much longer period of time, the depressurization rate decreases due to the discharge of a two-phase mixture of saturated steam and water. The flow of the mixture continues until the pressure in the vessel and containment become equal. Depending on the applied calculation code, pressure in the vessel slightly increases (RELAP5), decreases immediately

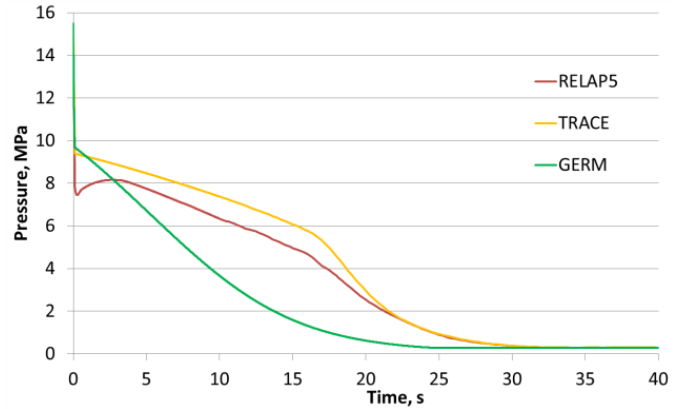


Figure 9: Pressure vessel pressure change

(TRACE) or remains constant for a short time, during the stage between the single-phase and two-phase discharge. This is caused by the boiling and evaporation processes compensating for the depressurization.

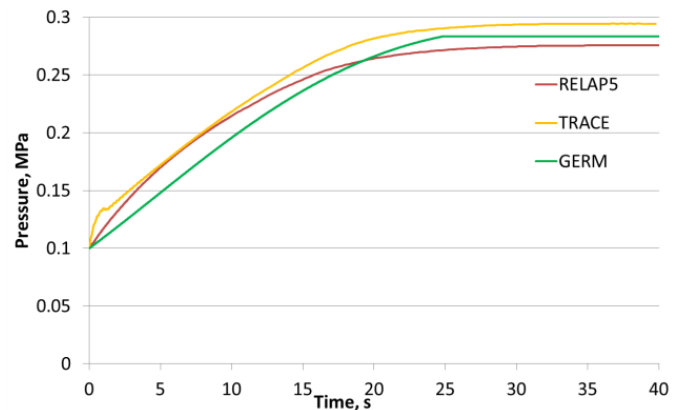


Figure 10: Containment pressure change

Pressure change in the containment building is shown in the Fig. 10. The rate of increase depends on the simulation code, but the final containment pressure values, calculated in every code and the analytical method, are in good agreement.

The critical mass flow rate, which is shown in Fig. 12, depends only on the pressure and water enthalpy inside the vessel and is not affected by the pressure change in the containment. Flow reaches its maximum value at the start of the discharge, when only single-phase flow occurs. When the water inside the vessel reaches saturated conditions and starts to evaporate, the void fraction increases as illustrated in Fig. 11. The mass flow rate decreases due to the

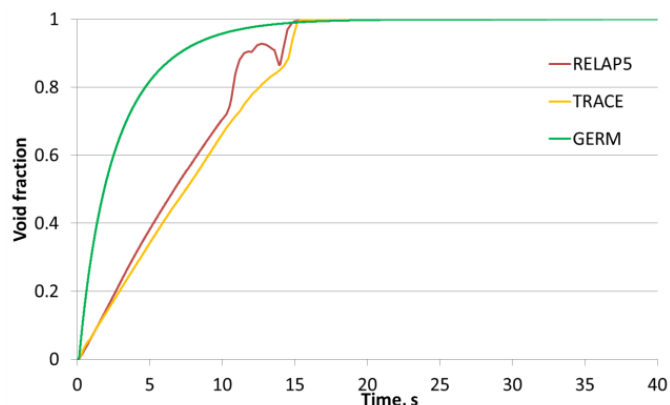


Figure 11: Void fraction change in the vessel

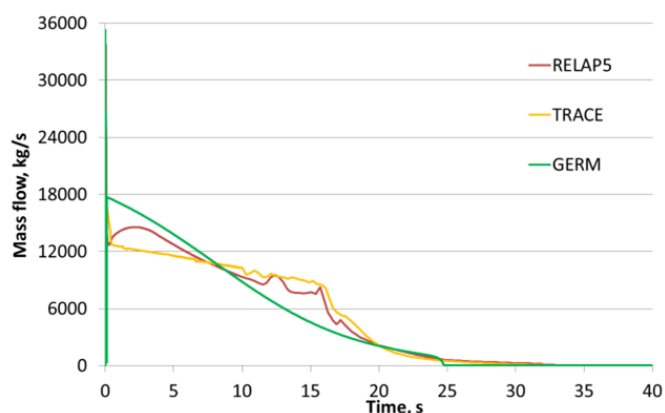


Figure 12: Mass flow rate change

decreasing density inside the vessel as well as the increasing void fraction. When the pressure is equal to the containment pressure, mass flow stops.

6. Conclusions

The results obtained seem to be in compliance with the literature, which confirms the correctness of the assumptions and approach.

There is still much work to be done on perfecting both analytical models and system codes in regard to critical flow modeling. The approach proposed in this paper is adequate for presenting only the timeframes of the phenomena, but is insufficient in terms of demonstrating where significant phases of the blowdown take place. Additional models should be taken into consideration, with a more extensive analysis including a sensitivity analysis of other homogenous and non-homogenous models. The differences between RELAP5 and TRACE are not signif-

icant but show that the TRACE model needs some fine tuning to produce better results. Additional work will be carried out to investigate and implement new models in system codes to better represent the critical two-phase flow.

Acknowledgments

The publication was created within the framework of a strategic project of the Polish National Center for Research and Development (NCBR): "Technologies for the development of safe nuclear energy", Research Task No. 9 entitled "Development and implementation of safety analysis methods in nuclear reactors during disturbances in heat removal and severe accident conditions".

References

- [1] F. D'Auria, P. Vigni, Two-phase critical flow models, Tech. Rep. 49, CSNI (1980).
- [2] N. E. Todreas, M. S. Kazimi, Nuclear Systems I Thermal Hydraulics Fundamentals, Taylor & Francis, 1993.
- [3] K. E. Carlson, V. H. Ransom, R. J. Wagner, The Application of RELAP5 to a Pipe Blowdown Experiment, Idaho National Engineering Laboratory, 1980.
- [4] Areva NP & EDF, UK EPR Pre-Construction Safety Report (2012).
URL <http://www.epr-reactor.co.uk/>
- [5] J. Lewandowski, M. Mościcki, Porównanie granicznych modeli rozprzestrzeniania się pary w budynku reaktora w trakcie awarii typu loca, Biuletyn Informacyjny Instytutu Techniki Ciepłej Politechniki Warszawskiej 81 (1995) 13–22.
- [6] R. T. J. Lahey, F. J. Moody, The Thermal-Hydraulics of a Boiling Water Nuclear Reactor, American Nuclear Society, 1993.