Transactions of the 15th International Conference on
Structural Mechanics in Reactor Technology (SMiRT-15)
Seoul, Korea, August 15-20, 1999

Condensation-Induced Water-Hammer Analysis on the Steam Generator Accident Relief System

Zhaohui Yao, Pengfei Hao and Xuefang Wang

Tsinghua University, China

ABSTRACT In this paper, the condensation-induced water-hammer analysis on the steam generator accident relief system is carried out. When accident occurs in the steam generator, the accident relief system is used. The high temperature and high pressure steam will flow into the cold water. In the juncture surface of steam and water, the steam will condense. At some condition, the condensation-induced water-hammer might occur and bring about enormous damage. This process is transient and of two-phase character. It is modeled by two-phase VOF method, in which the steam condensation is considered on the juncture surface. The velocity and pressure in the pipeline are obtained through numerical simulation. The results can supply the theoretical basis for engineering design.

INTRODUCTION

The water-hammer caused by cavity collapse has been a serious problem in engineering pipelines. There are some factors leading to this kind of water-hammer. If the main factor is vapour condensation, it is called condensation-induced water-hammer. The destruction effect of the condensation-induced water-hammer is very strong. As we know, the incidents occurred in American Indian Point Unit No. 2 on November 13, 1973 and San Onofre Nuclear Power Station on November 21, 1985, resulted from cavity collapse and brought about heavy losses. So how to determine the destructive force and avoid this kind of events has aroused much attention of scientists and engineers[1,2].

In this paper, the condensation-induced water-hammer analysis on the steam generator accident relief system is discussed. The paper presented the numerical simulation with modified VOF (volume of fluid) method. The simulation method is demonstrated successfully and the simulation results are in good agreement with experimental results.

1 PHYSICAL AND MATHEMATICAL MODEL

1.1 Physical Model

According to the structure of the generator accident relief system, a physical model for calculating condensation-induced water-hammer is proposed, as shown in Fig. 1.

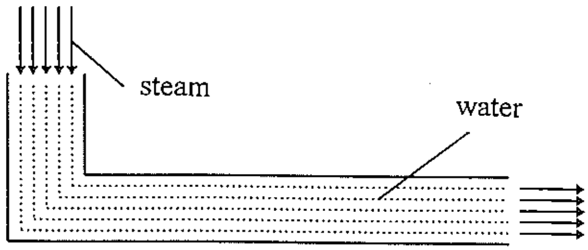


Fig. 1 Physical model of calculation

It is assumed that a two dimensional elbow tube is filled with water. At a certain time, the steam flows into water elbow tube. Condensation may take place at the surface between the water and steam. In the all course of the steam entering the elbow tube, it is supposed that the temperature and density of the cold water and steam remain the same.

1. 2 Mathematical Model

1. 2. 1 Governing equations

For a two-dimensional flow, the fluid equations to be solved are the Navier-Stokes equations.

$$\begin{cases} \frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+g_x+\nu[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}] \\ \frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+g_y+\nu[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}] \end{cases} \tag{1}$$

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{2}$$

Velocity components (u,v) are velocities in the Cartesian coordinate directions (x,y). Body accelerations are denoted by (g_x,g_y) , and ν is the coefficient of kinematic viscosity of water. ρ is the mixture density of steam and water.

1. 2. 2 Boundary condition

(1) For the entry of water, continuity flow condition and pressure condition should be used.
(2) For the wall of elbow tube, no-slip wall condition should be used.
(3) For the interface between water and steam, the free surface condition should be used. The condensation on the surface between steam and water should be considered.

2 THE SIMULATIING CALCULATION OF TWO-PHASE FLOW WITH VOF METHOD

In order to calculate the condensation on the surface between steam and water, the surface location must be determined. VOF method[3] is a powerful and simple way for the solution of transient fluid flow with free boundary. The basis of the VOF method is the fractionalization of fluid scheme for tracking free boundaries. In this technique, a function F(x,y,t) is defined. A unit value of F corresponds to a cell full of fluid, whereas a zero value indicates that the cell contains no fluid. Cells with F values between zero and one contain a free surface. In this paper, A unit value of F corresponds to a cell full of water, whereas a zero value indicates that the cell contains steam. Cells with F values between zero and one contain a surface between water and steam. The time dependence of F is governed by the following equation:

$$\frac{\partial F}{\partial t}+u\frac{\partial F}{\partial x}+v\frac{\partial F}{\partial y}=0 \tag{3}$$

This equation states that F moves with the fluid.

In numerical simulation, the staggered grid system is used, in which all dependent field variables (F, p) are calculated on a cell center, and flow variables are calculated on the surface of a cell.

The basic procedure for advancing a solution through one increment in time, δt, consists of three steps.

(1) Explicit approximations of Eq. 1 are employed to compute the first guess for new time-level velocities using the initial conditions or previous time-level values for all convective terms, pressure, and viscous acceleration.

$$\begin{cases} u_{i+1/2,j}^{n+1}=u_{i+1/2,j}^{n}+\delta t[-(p_{i+1,j}^{n+1}-p_{i,j}^{n+1})/\delta\rho x_{i+1/2}+g_x-FUX-FUY+VISX] \\ v_{i,j+1/2}^{n+1}=v_{i,j+1/2}^{n}+\delta t[-(p_{i,j+1}^{n+1}-p_{i,j}^{n+1})/\delta\rho y_{j+1/2}+g_y-FVX-FVY+VISY] \end{cases} \tag{4}$$

Where, FUX, FUY, FVX, FVY denote convective terms, $VISX, VISY$ stand for viscosity term, their expressions can be seen in Ref. 3.Superscript n denotes last time step, and $n+1$ for update time step. $\delta\rho x_{j+1/2}$ and $\delta\rho y_{j+1/2}$ are expressed as follows:

$$\delta\rho x_{i+1/2}=\frac{1}{2}\{[\rho_g+(\rho_f-\rho_g)F_{i,j}]\delta x_{i+1}+[\rho_g+(\rho_f-\rho_g)F_{i+1,j}]\delta x_i\} \tag{5}$$

$$\delta\rho y_{j+1/2}=\frac{1}{2}\{[\rho_g+(\rho_f-\rho_g)F_{i,j}]\delta y_{j+1}+[\rho_g+(\rho_f-\rho_g)F_{i,j+1}]\delta y_j\} \tag{6}$$

Where subscript g denotes steam and f for water.

(2) To satisfy Eq.3, pressures are iteratively adjusted in each cell and the velocity changes induced by each pressure change are added to the velocities computed in step(1). The finite difference form used for Eq. 2 is

$$\frac{(u^{n+1}_{i+1/2,j} - u^{n+1}_{i-1/2,j})}{\delta x_i} + \frac{(\mathrm{v}^{n+1}_{i,j+1/2} - \mathrm{v}^{n+1}_{i,j-1/2})}{\delta y_j} = 0 \tag{7}$$

This equation is an implicit relation for the new pressures. A Solution may be obtained by an iterative process.

(3) Finally, the F function defining fluid regions must be updated to give the new surface between water and steam.

In VOF method, changes of F in a cell at each time step reduce to fluxes of F across the cell faces. To preserve the sharp definition of surface boundaries, the calculation uses a type of Donor-Acceptor flux approximation[3] to get the fluxes of F across the cell faces. In our modified VOF method, if the condensation of steam exists in a certain cell face. The condensation of steam will be considered in the calculation of F flux across this cell face. The steam condensation quantity on the cell face in one step is: condensing rate j times the face cross sectional area and the time increment where

$$j = (\frac{M}{2\pi R})^{1/2}[\frac{p_g}{T_g^{1/2}} - \frac{p_f}{T_f^{1/2}}]\frac{2\sigma}{2-\sigma} \tag{8}$$

Eq.(8) is condensing rate formula in the form suggested by Ref.[4], where M, is relative molecular mass. R is universal gas constant, p_g and T_g are pressure and temperature (absolute) of steam, p_f and T_f are pressure, and temperature (absolute) of water. Condensing coefficient σ is 0.03-0.05 for water.

Once new F values are determined by the above method. The new vapor boundary location within a surface cell can be acquired. In the VOF technique, it is assumed that the boundary can be approximated by a straight tine cutting through the cell. By first getting the slope of this line and the side occupied by fluid, then it can be moved across the cell to a position that intersects the known amount of F volume in the cell. The detailed treatment can be found in Ref. [3]. This line is regarded as an approximation to the actual boundary of steam and water.

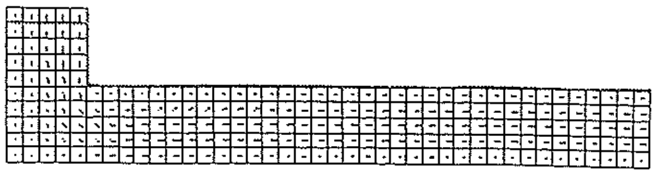
3 RESULTS AND ANALYSIS

According to the above method, the two-phase flow in the elbow tube is modeled. First, the section is filled with 104℃ water, then 440℃ hot steam enters the elbow tube. By numerical simulation, the process of the steam entering elbow tube is described in Fig. 2. The highest pressure curve in tube is obtained in Fig. 3. Because the pressure and speed of steam are high, the steam drives the water away quickly. The large steam bubble can not be formed, so no destructive condensation-induced water-hammer occurs. The steam only condenses at local surface to cause local pressure rising. The results of simulation agree with the physical reality. So the modified VOF method considering the condensation on the interface between water and steam can effectively simulate the process of two-phase flow with condensation.

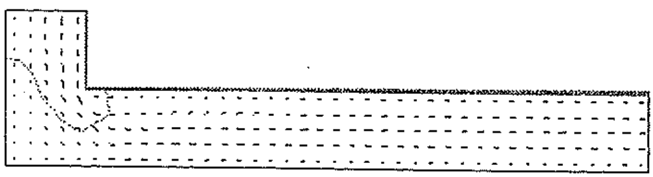
4 CONCLUSIONS

Based on the analysis of some available related accidents, a physical model and the corresponding mathematical model have been set up to describe the water-hammer caused by cavity collapse, which is induced by vapor condensation. The numerical simulation using the modified VOF method considers the condensation on the surface between steam and water, and is testified to be useful and reliable. However, the numerical simulation of condensation-induced water-hammer is only for two-dimensional cases. It needs to be developed to three-dimensional simulation to meet the requirements of engineering practice.

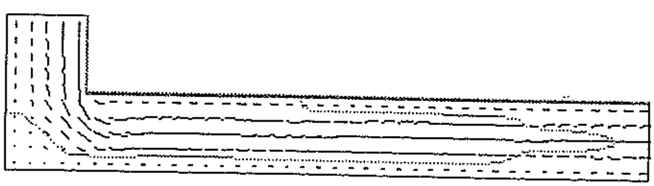
TIME=0.00E+00s SIZE: vt=-9.08E+00 m/s pt= 2.19E+06pa



TIME=2.00E-03s SIZE: vt=-9.28E+00 m/s pt= 2.19E+06pa



TIME=8.00E-03s SIZE: vt=-2.15E+01 m/s pt= 2.19E+06pa



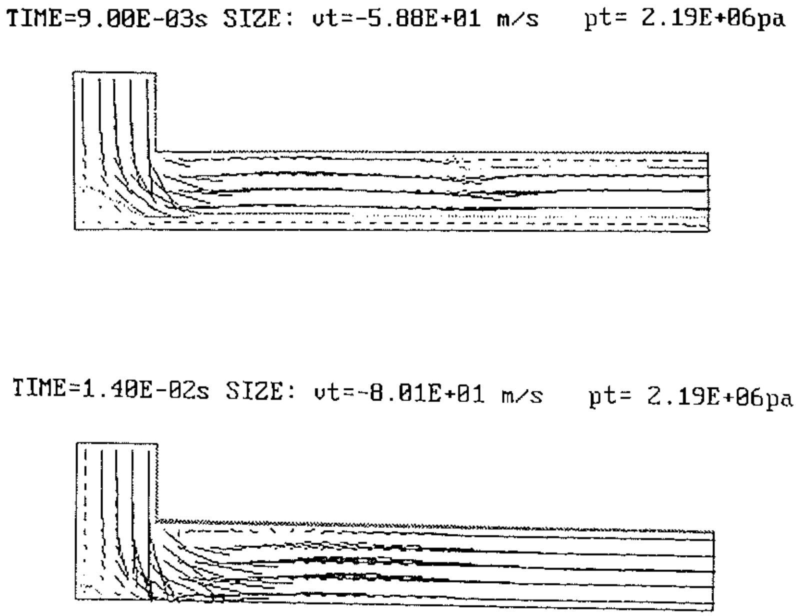


Fig. 2 The process of the steam entering elbow tube

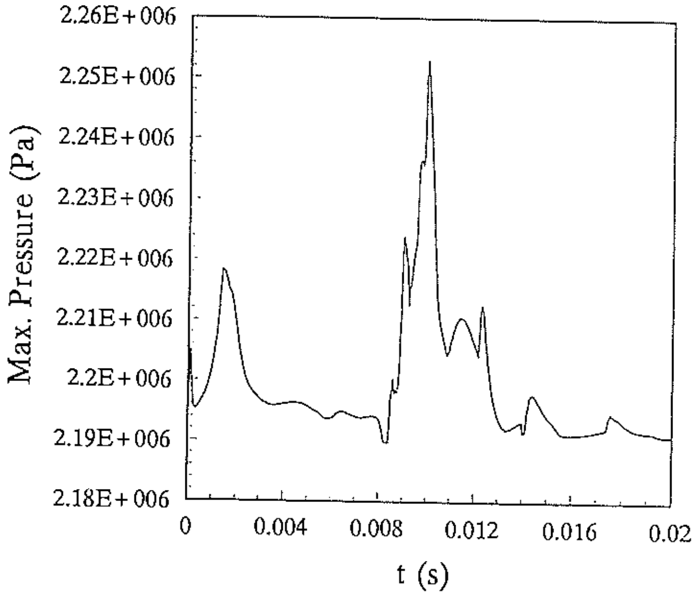


Fig. 3 The highest pressure curve in tube

REFERENCES

1. Chaudry M. F., *Applied Hydraulic Transients,* New York: Van Nostrand Reinhold Ltd, 1979.
2. Ye Hongkai. Wang Xuefang, Wu Lixin, " Forming and collapsing of steam bubble in feed water pipe of steam generator," *Chinese Atomic Energy Science and Technology,* Vol. 23, No. 5,1989, pp. 57-62.
3. Nichols B D., Hirt C W, Hothkiss R S., *Volume of Fluid(VOF) Method For the Dynamics of Free Boundaries,* Los Alamos Scientific Laboratory Report, LA-8335, 1980
4. Collier J G., *Convective Boiling and Condensation,* New York: McGraw-Hill International Book Co, 1981