*Transactions of the 17th International Conference on Structural Mechanics in Reactor Technology (***SMiRT 17)**
Prague, Czech Republic, August 17 –22, 2003
Paper # P01-5

Modelling of Aquitaine II Pipe Whipping Test with EUROPLEXUS Fast Dynamics Code

Serguei Potapov

EDF R&D, Département Analyses Mécaniques et Acoustique, 92141 Clamart Cedex, France

ABSTRACT

This paper presents a numerical simulation with EUROPLEXUS fast dynamics software of a pipe whipping phenomenon occurring in thermal hydraulic conditions of a Loss of Coolant Accident in PWR primary circuit. Different physical phenomena take place simultaneously during the opening and whipping of the pipe such as plasticity, contact, large displacements, two-phase flow regime, fluid-structure interaction. Two numerical models, a simplified « pipe like » model and a mixed 1D/3D model, are considered and compared throughout modelling and computation. The numerical results are compared with experimental data belonging to Aquitaine II test campaign.

KEY WORDS: LOCA, fast dynamics, impact, whipping, blowdown, fluid structure interaction.

INTRODUCTION

EUROPLEXUS is a general fast dynamics software [1-3] dedicated to numerical analysis of accidental situations, involving transient dynamic phenomena in general systems composed of structures and fluids. Possessing very efficient fluid-structure interaction (FSI) algorithms, EUROPLEXUS has been used to assess the LOCA hydrodynamic effects in the main primary circuit of a PWR [4] and to simulate a steam explosion in a PWR reactor pit [5].

In order to validate the modelling with EUROPLEXUS of multi-physics phenomena we consider a pipe whipping problem occurring in thermal hydraulic conditions of a Loss of Coolant Accident in PWR primary circuit. Different physical phenomena take place simultaneously during the opening and whipping of the pipe such as plasticity, contact, large displacements, two-phase flow regime, fluid-structure interaction. We show how EUROPLEXUS takes up a challenge of modelling the whole non linear situation.

First, the physical problem to be studied is presented. Then, two FSI numerical models are described. They will be compared throughout modelling and computation. Next, the numerical simulation of the problem is presented. It is performed in two steps : a conditioning phase, when the pressure in the system is increased until a nominal value, and a phase of whipping. For each phase the results are discussed with respect to different physical phenomena taking place. Finally, the numerical results of 1D and 1D/3D calculations are compared with existing experimental data of Aquitaine II test campaign [6].

PROBLEM DESCRIPTION

The studied test facility is composed of a cranked pipe connected to a rigid reservoir of a great capacity (Fig. 1).

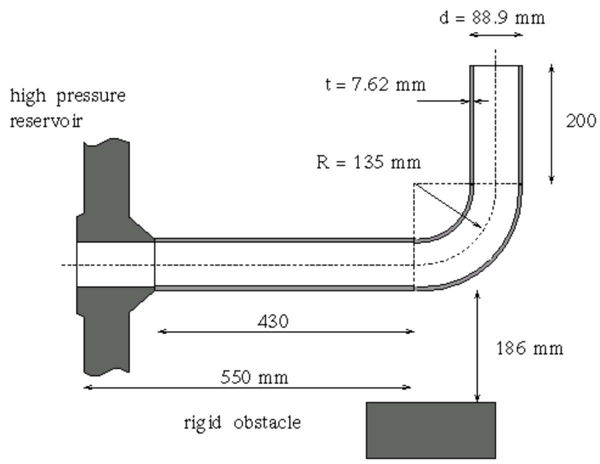


Fig. 1. Experimental configuration.

A rigid motionless device is disposed below the horizontal part of the pipe. The free extremity of the pipe is initially closed by a rigid membrane separating the internal volume of the pipe from the atmosphere. In the beginning of the experiment (phase of conditioning), the pipe and the reservoir are filled with hot water, pressurised until 16.6 MPa and heated up to 326°C. The whole system is maintained in static and thermal equilibrium.

The dynamic phase of the experiment starts with a sudden break of the membrane. Due to a great difference of internal and external (atmospheric) pressures the fluid is spurted out of the pipe, pushing the pipe downwards by a reactive force of a two-phase water jet. Because of flexibility of the pipe, plastic deformations occur in its horizontal part near the reservoir, and the free end of the pipe undergoes large vertical displacements resulting in the impact of the pipe elbow on the rigid obstacle. The pipe cross section is locally crushed absorbing in plasticity process the most of the piping kinetic energy.

TWO EUROPLEXUS MODELS

Finite element meshes

To simulate with EUROPLEXUS the Aquitaine II pipe whipping experiment we build two FSI models (Fig. 2). The first one is a simplified "pipe like" model using TUYA flexible finite elements having 7 degrees of freedom per node : 6 dof corresponding to Euler beam kinematics and 1 dof of fluid under a plane wave assumption. Those TUYA elements are intrinsically coupled and no complementary FSI conditions are needed. All flexible parts of the piping are discretised with TUYA elements (Fig. 2, a), only the flanged part of the pipe (OA line) is modelled with TUBE elements having 1 (fluid) dof per node. In the following, this « pipe like » model will be also referred to as 1D model because of 1D fluid modelling.

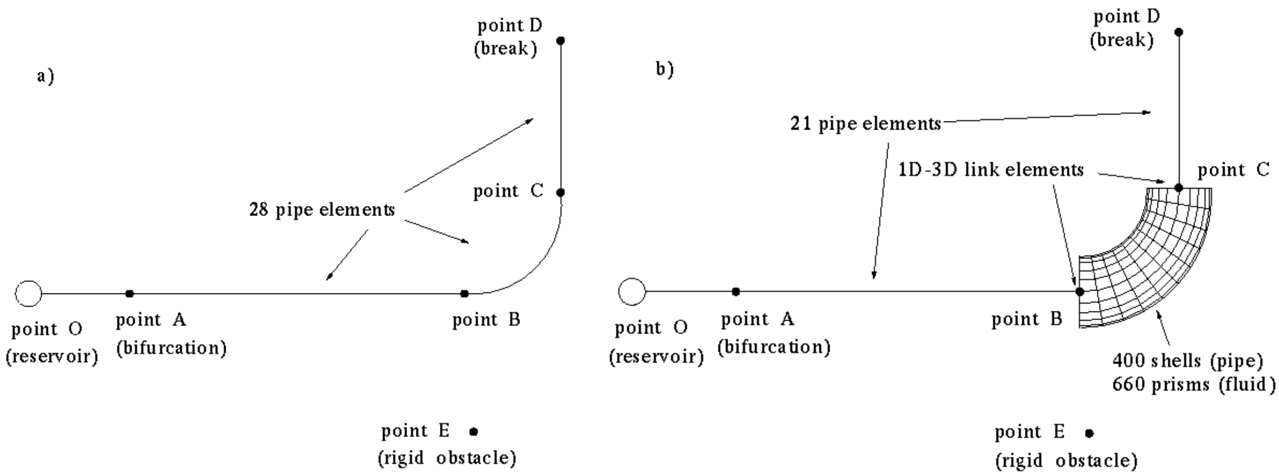


Fig. 2. « Pipe like » (a) and mixed 1D/3D (b) EUROPLEXUS models.

The second model possesses a mixed kinematics (Fig. 2, b). The straight parts of the piping (AB and CD) are still represented with TUYA elements, whereas the elbow zone (BC) is modelled more finely : the fluid is considered to be in 3D conditions, thus it is discretised with prism elements (Fig. 3, a), and the pipe elbow walls are discretised with TRI3 shell elements (Fig. 3, b) allowing the pipe cross section to deform.

Fig. 3. Fluid (a) and pipe (b) 3D finite element meshes of the elbow zone.

Structural degrees of freedom (3 translations and 3 rotations per node) of the « pipe like » and the shell meshes are interconnected using a special link procedure which imposes the beam's rigid cross section kinematics to the nodes of shell elements belonging to the link surface.

In both EUROPLEXUS models the short tube OA, connecting the flexible pipe to the reservoir, is considered rigid and the fluid contained in this tube is modelled by TUBE fluid elements. To interconnect TUBE and TUYA elements a special bifurcation BIFU element is used. It links a single dof of the TUBE element with the 7^{th} (fluid) dof of the TUYA element. From the mechanical point of view the connection of TUBE and TUYA elements is supposed rigid and fixed, modelling the pipe clamping.

Both models use the same 1D definition of the reservoir and the break.

Fluid-structure coupling in 1D/3D model

In order to interconnect different parts of the mixed 1D/3D fluid mesh, a new link element called TUYM has been developed and introduced to the code. It allows the coupling of a single fluid dof of TUYA element with 3 dof of the 3D fluid model. The fluid flow continuity is prescribed on the 1D/3D interface by means of Lagrange multipliers method. The link surface and outward normals at linked faces of 3D elements are updated at each time step of calculation in order to follow the global piping motion.

Along the fluid-structure interface between the 3D fluid volume of elbow zone and the shell mesh of the pipe elbow a special FSA (Fluid-Structure A.L.E.) model is used [7].

Because the pipe undergoes large displacements and deformations during the whipping phase, the fluid domain configuration changes considerably. To manage the fluid mesh motion we use a special CONTOUR directive. For each cross section of the pipe elbow this directive constrains internal fluid mesh nodes to follow the surface which contour is defined by shell nodes of the pipe mesh.

Material characteristics

In our simulation the material of the pipe is assumed to be elastoplastic with isotropic hardening and Von Mises plastic criterion (the kinematic hardening being not available in the code). The pipe mechanical characteristics at 326 °C are: Young's modulus E = 180 GPa, Poisson's ratio $\nu = 0.33$, density $\rho = 7800$ kg/m^3, yield stress $\sigma_y = 155$ MPa, rupture limit stress $\sigma_r = 475$ MPa, deformation at rupture $\varepsilon_r = 25$ %.

The fluid is modelled using a homogenous equilibrium model which postulates the thermal and mechanical equilibrium (same pressure, same temperature, same velocity) between liquid and steam phases. Different thermodynamic parameters of the liquid-steam mixture are determined by interpolation from the temperature and the local pressure values using steam tables [8]. At the break a more complex fluid model is used. It follows Moody's hypotheses [9] postulating pressure equilibrium between liquid and steam and possibility of phase sliding (different velocities of liquid water and steam). A one dimensional annular flow is also supposed at the break.

In all calculations we suppose that there is no hit exchange with the outside (adiabatic hypothesis), and that pressure losses along the pipe walls can be neglected.

Observation transducers to compare results

In order to compare numerical results of EUROPLEXUS calculations with experimental data, some element and node located observation transducers are introduced with respect to the finite element meshes of two models (Fig. 4).

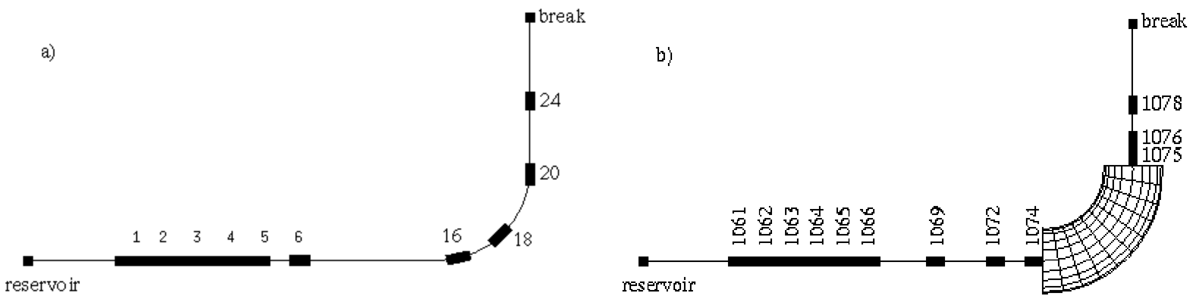


Fig. 4. Post-treatment elements : (a) « pipe like » model, (b) mixed 1D/3D model.

To extract the time history of node located variables such as fluid and pipe velocities, pipe displacements and contact force we use the points O, A, B, C, and D (Fig. 2) coinciding with finite element nodes.

EUROPLEXUS COMPUTATIONS

Simulation of the conditioning phase

In order to initialise accurately the phase of whipping, a static equilibrium of the system under the internal pressure action is first calculated. Because the code does not allow an artificial progressive increase of internal pressure in the fluid-structure system in order to obtain a quasi-static solution, we use a dynamic relaxation procedure. In this purpose, the membrane at the free end of the pipe is modelled by a closed end condition and the whole internal pressure value of 16.6 MPa is instantaneously applied inside the system. To dump dynamic effects due to a sudden pressure application, we use a numerical dumping for low frequencies. This computation is performed until 10 ms, the time where dynamic effects in the pipe are practically disappeared and dynamic solution is stabilised close to static solution. Evolution curves of displacements and stresses at different points of the « pipe like » model are presented in Figures 5 and 6.

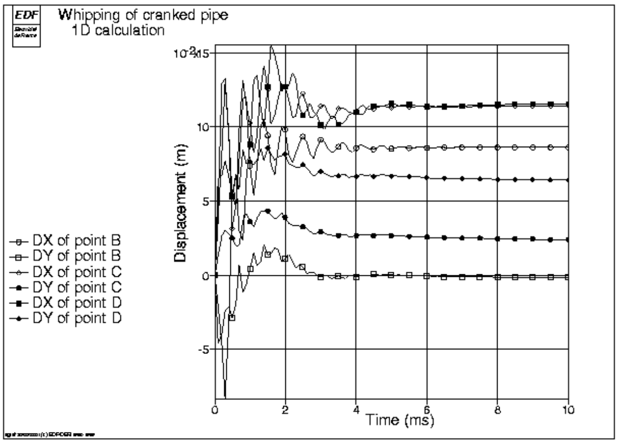


Fig. 5. Conditioning phase: displacements.

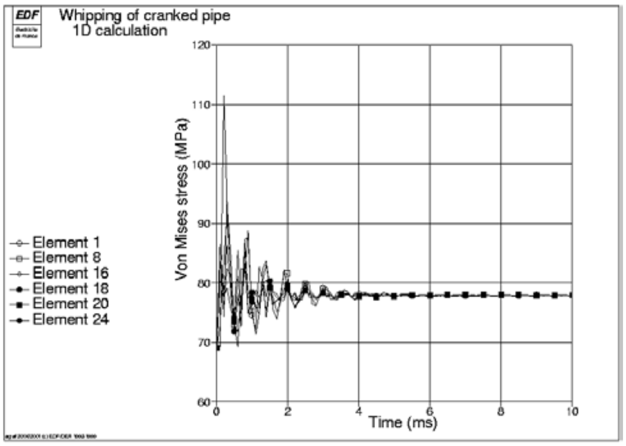


Fig. 6. Conditioning phase: Von Mises stresses.

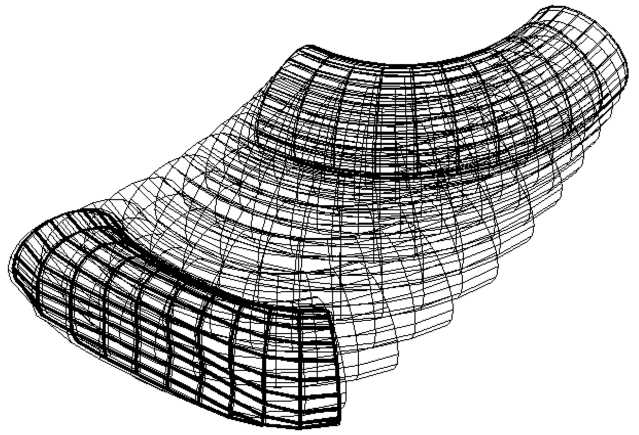
One can see that during the stabilisation process Von Mises stresses do not reach the yield stress equal to 155 MPa. Thus there is no hardening of the material during the conditioning phase, and the procedure we use to obtain the static solution remains acceptable because spurious oscillations do not generate any artificial hardening. Under the action of internal pressure the pipe elbow opens, but pipe displacements are relatively small.

Simulation of the phase of whipping

The computation of the whipping phase is started from the static solution obtained in the conditioning phase. In both « pipe like » and mixed models the closed end condition at the free extremity of the pipe (point D) is replaced by an open break condition based on a critical flow rate law according to Moody's model [9]. The computation is carried out until 80 ms of physical time.

Dynamic response of the pipe

Because of membrane rupture, the internal pressure falls down in the break zone. A strong reactive force due to water jet occurring pushes the pipe downwards. The pipe undergoes large vertical displacements resulting in impact of the pipe elbow on the rigid obstacle. The pipe cross section is locally subjected to large deformations (Fig. 7) absorbing by plasticity process the most of piping kinetic energy.



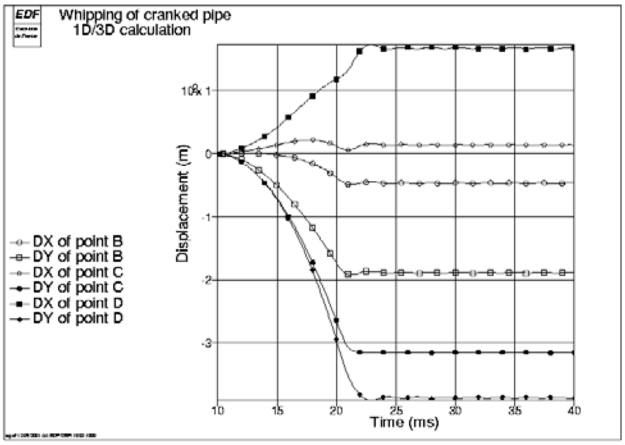
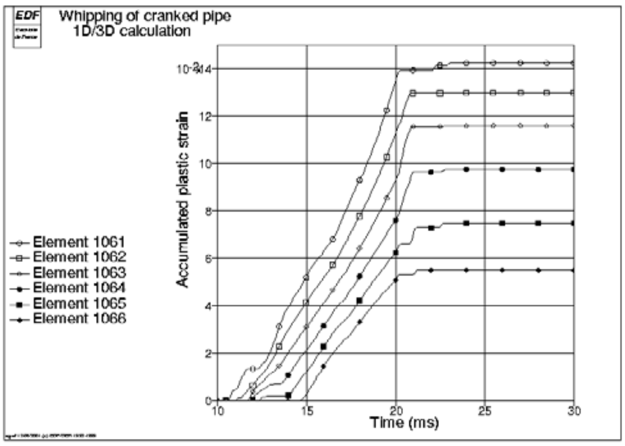


Fig. 7. Elbow deformation shape at various times. Fig. 8. Displacements of piping points.

Figure 8 presents the evolution of displacements predicted by the mixed 1D/3D model for the points B, C and D. One can observe a sudden change of curves' trend occurring 10 ms after the break opening (20 ms after the beginning of computation). This change is due to the impact of the pipe elbow on the rigid obstacle. After the first severe impact, several small elastic rebounds of the pipe on the obstacle are visible.

The plastic flow occurs first in the connection zone 0.6 ms after the break opening. This zone extends and progresses in the direction of the elbow (Fig. 9,10). The maximum value of the plastic strain in the horizontal part of the pipe is of about 14.5 %. It is observed in element 1061 10 ms after the beginning of the membrane rupture.



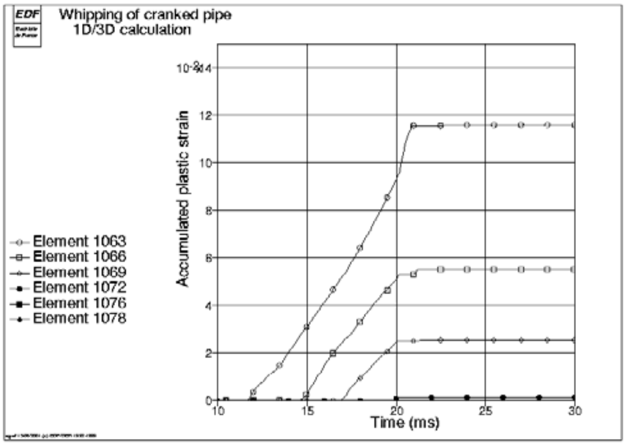


Fig. 9. Accumulated plastic strain in the connection zone. Fig. 10. Accumulated plastic strain in straight legs.

Time evolution of fluid behaviour

Some interesting phenomena are observed in the fluid. Before the membrane rupture, the fluid in the system is maintained in liquid state due to the high internal pressure. After the break opening, a two-phase flow regime occurs at the opened end of the piping. The pressure at the break drops down almost immediately, whereas near the reservoir one can observe a rather progressive decrease until the saturation pressure (Fig. 11, 12).

Due to the impact of the pipe on the obstacle the vertical motion of the fluid volume is suddenly stopped producing a rapid pressure increase in the fluid, that can be observed on pressure history curves. Furthermore, the 1D/3D model allows us to detect a local pressure perturbation due to a local deformation of the pipe cross section during the shock (Fig. 12). Because of its reduced kinematics, the 1D model does not allow the detection of this effect.

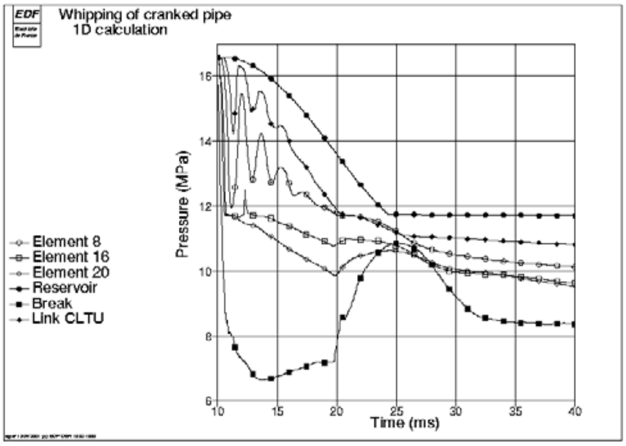


Fig. 11. Pressure time history in 1D calculation.

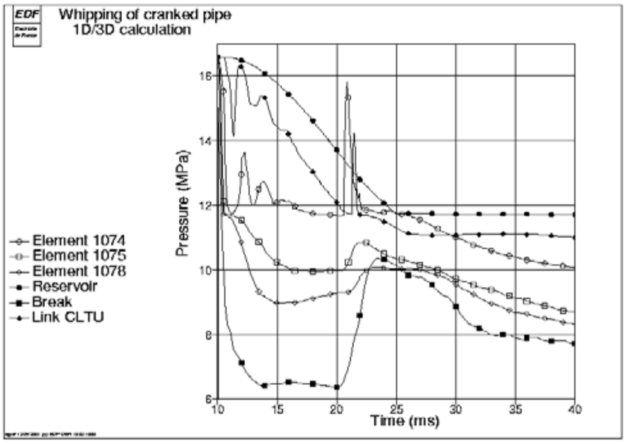
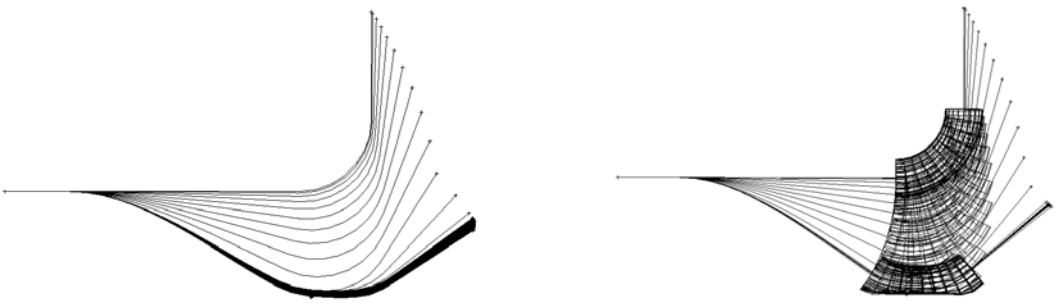


Fig. 12. Pressure time history in 1D/3D calculation.

Comparison of 1D and 1D/3D solutions

In this paragraph we present the comparison of numerical results given by the two EUROPLEXUS models. First we show the deformed shapes of the pipe for various successful times of calculation (Fig. 13). Quantitatively both models present the same kind of movement and the same final deformed position. This conclusion is corroborated by comparison of displacements of different points of the pipe (Fig. 14). One can observe that in 1D/3D calculation the displacements are slightly more important. Indeed, the 3D model of the elbow takes into account local crushing of elbow's wall, thus vertical displacements are greater. When comparing the evolution of Von Mises stresses for three elements of the pipe located near the clamped end, one can note a very good agreement of curves for the two models until the moment of impact of the elbow on the obstacle at 20 ms. After impact, stresses in 1D/3D calculation are practically one and a half times greater than those predicted by 1D model.



a) b)

Fig. 13. Deformation shapes during the whipping, (a) « pipe like » model, (b) mixed 1D/3D model.

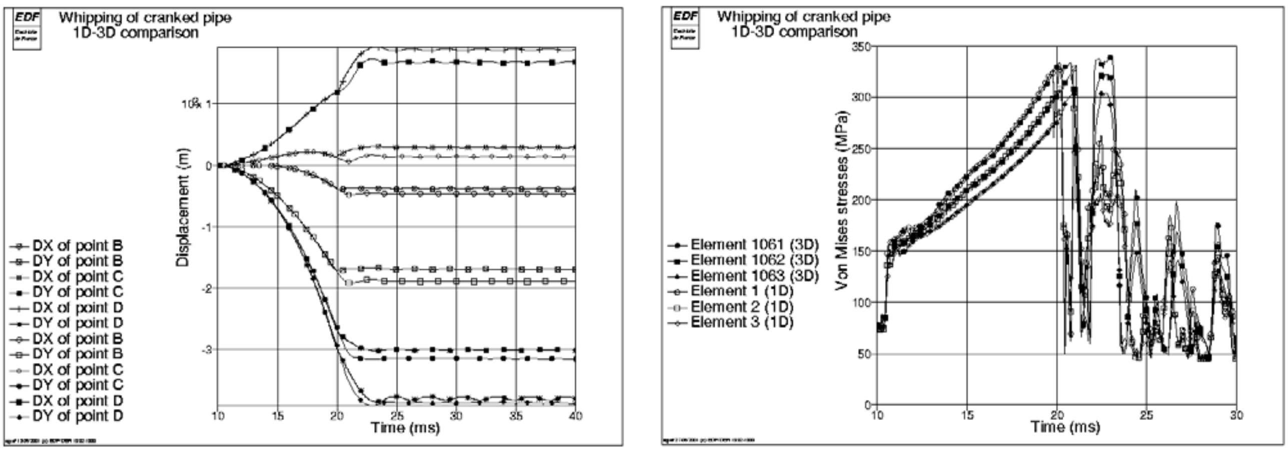


Fig. 14. Displacements and Von Mises stresses in 1D and 1D/3D calculations.

Until 20 ms the predictions of both models for fluid velocity and pressure are similar (Fig. 15). After impact, the velocities upstream the elbow are lower and downstream the elbow greater in 1D/3D calculation than in 1D calculation.

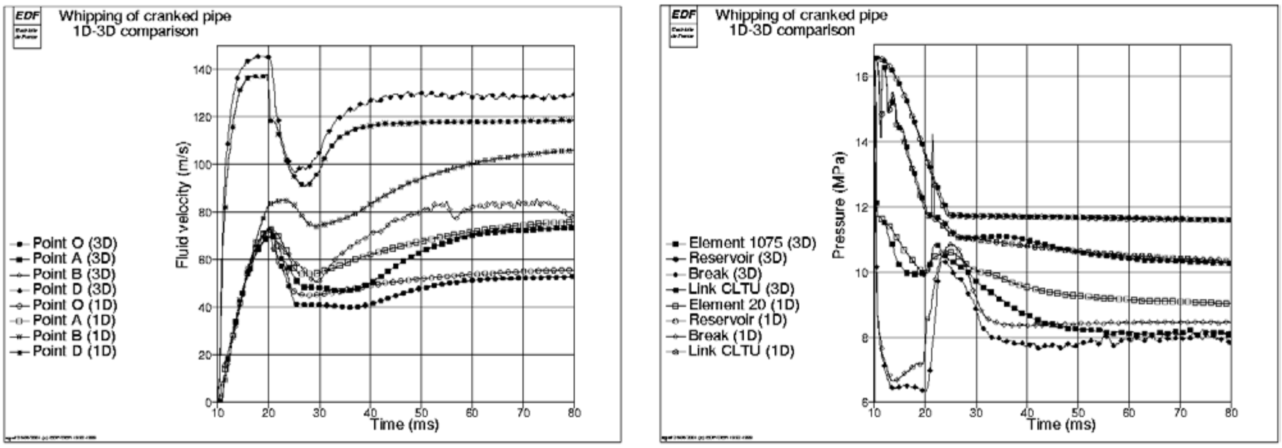


Fig. 15. Velocities and pressure in 1D and 1D/3D calculations.

This effect is due to a sudden narrowing of the pipe cross section, which slows down the fluid flow before the elbow and speeds it up just after the elbow. The pressure time history plots show a good global agreement for 1D and 1D/3D calculations. However, only the 1D/3D model allows us to detect a local pressure perturbation due to a local deformation of the pipe cross section during the shock. Because of its reduced kinematics, the 1D model does not allow the detection of this effect.

Two models considered here predict different values of the contact force between the pipe and the rigid obstacle (Fig. 16, 17). The 1D calculation gives the maximum value of 1200 kN which is nearly three times greater than the experimental value equal to 460 kN. The contact force time history in 1D calculation is also very different being compared with experimental curve (Fig. 16,18). This result is due to reduced kinematics (rigid cross section hypothesis) of the 1D model which cannot take into account the deformation of pipe elbow's wall. The use of the mixed 1D/3D model allows us to avoid this difficulty and catch the maximum value of contact force as well as its time history.

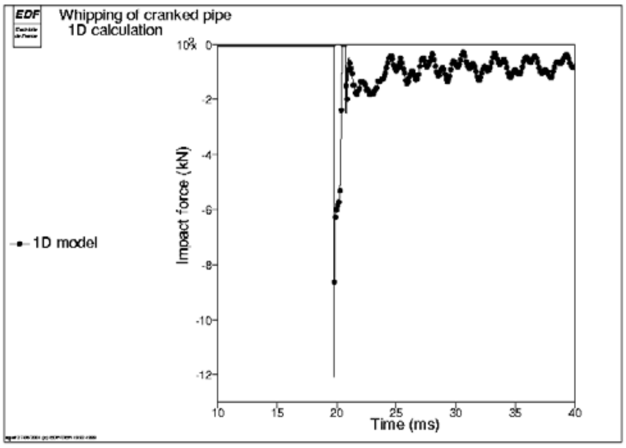


Fig. 16. Impact force in 1D calculation.

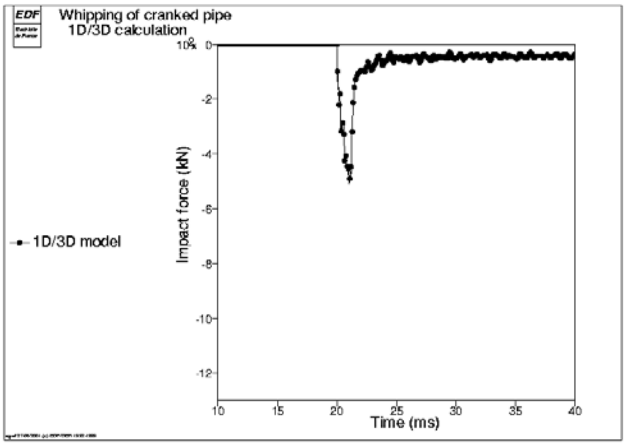


Fig. 17. Impact force 1D/3D calculation.

Comparison of numerical results with experimental data

Now we compare numerical results of 1D/3D model with experimental data. Taking account of existing measurement data, we compare the time of impact, the contact force value and evolution of pressure in the reservoir.

The time of impact is the time measured from the beginning of dynamic phase until the shock of the pipe on the rigid obstacle. Calculated value of 10 ms is slightly lower than the measured one which is of about 10.5 ms.

The mixed model allows us to estimate relatively well the maximum value of the contact force, about 485 kN, comparatively to the experimental value equal to 460 kN. The contact force time history predicted by the simulation is also similar to that measured in the experiment (Fig. 18). Namely, the calculation succeeds in reproducing the progressive force increase containing several small snapbacks followed by a rather fast shutdown. It should be reminded that the 1D model without contact stiffness update does not allow the correct detection of the contact phenomenon.

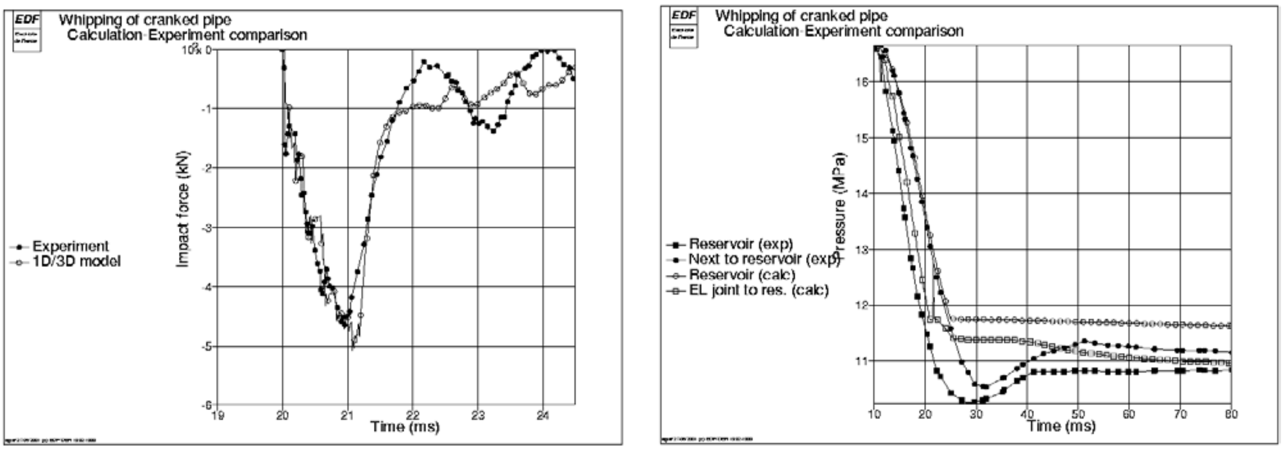


Fig. 18. Comparison of 1D/3D calculation with experimental data for the impact force and the pressure.

Large elastic rebounds following the impact are however not captured by the simulation. It is probably due to the use of isotropic hardening model modifying the elastic domain of the material. Indeed, the cyclic response of the steels should be better predicted with kinematic hardening model not available in the code.

We compare calculated and measured pressure values in the reservoir and in a point of piping nearest to the reservoir (Fig. 18). One can observe a good agreement between numerical and experimental curves until about 10 ms, the time when the pressure in the calculation drops down to saturation pressure value of 11.8 MPa in the reservoir and 11.4 MPa in the pipe. Then, the pressure level in calculation decreases slightly during several milliseconds, whereas experimental curves show a more complex physical phenomenon. In the experiment the pressure shuts down below the saturation pressure, before coming back to that level. This phenomenon of under-shut is due to a loss of thermal-hydraulic equilibrium in transient regime. Our homogeneous equilibrium model does not allow to reproduce this under-shut. Nevertheless, after several milliseconds of calculation a quasi-established regime is reached, and numerical and experimental curves show a better concordance.

CONCLUSION

To validate the modelling of multi-physics phenomena with EUROPLEXUS code we considered a pipe whipping problem occurring in thermal hydraulic conditions of a Loss of Coolant Accident in PWR primary circuit. Two numerical FSI models, a simplified « pipe like » model and a mixed 1D/3D model, were used to simulate both the conditioning phase and a phase of whipping. The results of calculations were compared with existing experimental data.

Analysis of numerical results shows that both models give a good prediction of global behaviour of the coupled fluid-structure system, namely for pipe displacements and stresses in the pipe walls, as well as for pressure and velocity in the fluid. By comparison with experimental data, we show that only the mixed EUROPLEXUS model, where the pipe elbow is discretised with shells, allows us to estimate correctly the time history and maximum value of the contact force between the pipe and the obstacle. The 1D model with reduced kinematics (rigid cross section hypothesis) does not allow the correct detection of contact phenomenon.

This study shows that the use of mixed numerical models containing simplified and totally 3D parts duly interconnected allows a very efficient and CPU inexpensive numerical analysis which is able to take into account different global and local physical phenomena.

REFERENCES

1. Chavant, C., Hoffmann, A., Lepareux, M., Verpeaux, P., Dubois, J., 1979. PLEXUS : A general computer code for explicit lagrangian computation, Proc. SMIRT 5, Vol. B 2/8, Berlin, Germany.
2. Hoffmann, A., Lepareux, M., Schwab, B., Bung, H., 1984. PLEXUS : A general computer program for fast dynamic analysis, Proc. Conf. On Structural Analysis and Design on Nuclear Power Plants, Porto Alegre, Brazil.

3. Bung, H., Casadei, F., Halleux, J.P., Lepareux, M., 1989. PLEXIS-3C : A computer code for fast dynamic problems for structures and fluids, Proc. SMIRT 10, Vol.B, Los Angeles, USA, pp. 85-90.
4. Potapov, S., Robbe, M.F., Téphany, F., 2000. Hydrodynamic consequences of a LOCA in a 4-loop PWR, Proc. ECCOMAS, Barcelona, Spain.
5. Casadei, F. Simulation of a Steam Explosion in a Cavity by PLEXIS-3C, Special Publication N.I.95.12, February 1995.
6. Martin, A, Sermet, E, Garcia, J.L, 1984. AQUITAINE II complémentaire : Rapport final d'essais. Note technique DRE/STRE/LMA-84/373.
7. Casadei F., Halleux J.P., An Algorithm for Permanent Fluid-Structure Interaction in Explicit Transient Dynamics. Computer Methods in Applied Mechanics and Engineering, 128, 3-4, 231-289, December (1995).
8. Haar, L., Gallagher, J.S., Kell, G.S., 1984. NBS/NRC steam tables, National Standard Reference Data System.
9. Moody, F.J., 1965. Maximum flow rate of a singlr component two-phase mixture, Journal of heat transfer.