PLEXUS — a General Computer Program for the Fast Dynamic Analysis — the Case of Pipe-Circuits

M. Lepareux, B. Schwab
C.E.A., CEN Saclay, IRDI/DEMT/SMTS/LAMS, F-91191 Gif-sur-Yvette Cedex, France
H. Bung
Compagnie Internationale de Services en Informatique (CISI), 35 bd. Brune, F-75680 Paris Cedex 14, France

Abstract

The dynamics of pipe circuits containing flowing liquids, and in particular the case of an accident, present a problem for the safety of nuclear power plants.
The rupture of the pipe involves a depressurization in the circuit and this is followed by a whipping action of the brocken pipe.
More generally the situations which involve the rapid speeding up of fluids in circuits, with or without fluid-structure coupling, may be treated by PLEXUS.
The results for the rigid-pipe case, incorporating the most caracteristic situations, will be shown first. The second part will present some preliminary calculation results for the complete system with deformable pipe circuits and taking into account the fluid-structure coupling.

I. INTRODUCTION

The safety of nuclear power stations depends on a good understanding of the movement of pipes in the presence of an internal flow. A sudden piping burst in particular leads to depressurization of the circuit followed by whipping of the pipe in question.
Developments made in the program PLEXUS enable us to analyse this problem. More generally all situations involving the acoustical and speed acquisition aspects of a fluid inside a pipe, with or without fluid-structure interaction, can be treated in a similar way.
After a brief description of the PLEXUS program and its possibilities part 1 of this paper will deal with phenomena related to the fluide alone, the pipes being assumed rigid. The new potentialities of the program will be illustrated by a number of tests. Part 2 will be devoted to the fluid-structure interaction proper, and some examples will be given to show the different effects of this coupling.

II. PLEXUS

II.1 - General aspects

PLEXUS, a general program of the CASTEM system developed by the CEA-DEMT [1][3], can analyse structures modelised in one, two or three dimensions. The program calls on the finite element method and the time resolution is explicit. For a few special cases however (incompressible fluids for instance) an implicit method may be used.

The formulation is Lagrangian or A.L.E. (Arbitrary Lagrangian Eulerian). In the former case the use of an automatic remeshing algorithm is possible. The formulation accounts for major displacements and major deformations. Many non linear laws of behaviour have been specially developed :

- Elastoplasticity (effect of deformation rate)
- Concrete model
- Compressible, incompressible, viscous fluids,
- Water (homogeneous two-phases liquid-vapour mixture)
- Hydrogen explosion
- Sodium-water reaction
- etc...

An example of application of PLEXUS to reinforced concrete slabs is dealt with in another paper at the present SMIRT [2].

II.2 - Lagrangian formulation

The algorithm used is quite conventional. Deformations are estimated from the speeds and accelerations of the previous step, then new stresses are calculated using the constitutive equation. Large strains are taken into account.

For greater computing speed the mass matrix is diagonal. As with all explicit algorithms a stable time step is involved based on the time needed for a wave to pass through the smallest element.

In spite of this disadvantage, the explicit formulation is very useful for describing non-linear behaviour.

II.3 - A.L.E. formulation

The schema used is derived from the Lagrangian with additionnal terms related to various fluxes through the boundaries of the elements. Our method is very similar to that proposed by J. DONEA (ISPRA) [4][5].

The meshing speed $\vec{W}$ is arbitrary and usually different from the material speed $\vec{V}$.

Two end cases being :

$\vec{W} = 0$ Eulerian formulation

$\vec{W} = \vec{V}$ Lagrangian formulation

We shall have two kinds of nodal quantity, related to the mesh on the one hand and to the particle present at the node at this instant on the other.

To obey the laws of conservation we shall have to modify the mass matrix and account for energy fluxes in the behaviour laws, while transport forces will add to the other forces.

II.4 - Interactions and boundary conditions

These often take the form :

$$Cu = D \geqq 0$$

where C is a coefficients matrix, u a displacement vector and D a second member.

The presence of interactions brings an extra force into the movement equation obtained by the Lagragian multipliers method :

$$F_c = C^T \lambda$$

a simple equation then gives the multipliers :

$$\lambda = - H^{-1} C M^{-1} [F_e - F_i]$$

The method calls on inversion of the matrix $H = C \ \ H^{-1} \ C^T$, the size of which depends on the number of degrees of freedom coupled. The inversion will only be performed at each step if the coupling matrix depends on the coordinates.
This schema covers fluid-structure interactions as well as unilateral contacts.

III. CIRCUITS

The aim is to describe the movement of piping in the presence of a flow. The Lagrangian method is well suited in the case of the pipes whereas an Eulerian description of the inner fluid is simpler. The A.L.E. method is therefore indispensable.
The program was developped in two stages :

1 - Treatment of fluide alone, the pipes being assumed rigid.

2 - Resolution of the whole system with allowance for the fluid-structure interaction.

III.1 - Specific problems

A good description of even a simple circuit entails a modelisation suited to the specific problems of the piping :

1 - Connections between branches

2 - Varied boundary conditions

3 - Local singularities

4 - New behaviour laws.

III.2 - Fluid only

In the case of rigid piping the basic element ("TUBE") will be a two-node segment with one degree of freedom per node. An equation of state will determine the mean pressure of the element. For the junctions we can choose between a cavity ("CAVI"), where a certain volume is under a mean pressure described by a behaviour law similar to that of the "TUBE" elements, or merely a simple branch-off ("BIFU") where the volume is practically nul. The flow conservation will then be written for the different branches.
Finally the end conditions will be entered by means of one-node elements ("CL1D") where the laws of behaviour will dictate pressure losses, external pressures or critical flow conditions.
Since the length of the circuits to be represented is sometimes very great the sequence of physical events taking place inside them must be properly described. For a depressurization, in particular the vapourisation effects are essential, which is why a two-phase behaviour law based on a homogeneous model has been introduced for water.

III.3 - Incompressible fluid

For similar reasons, the phenomena sometimes lasting for long periods where the acoustical aspect is no longer essential, the behaviour of the fluid will be described without the impediment of an explicit acoustics-related time step.
The method is based on an implicit pressure resolution.
In the Lagrangian formulation the displacements were computed from an estimation of the velocity :

$$\Delta u = t \ v^* \qquad \text{with } v^* = v^n + \frac{t}{2} \gamma^n \qquad (t : \text{time step})$$

We now write :

$$v^* = v^n + \frac{t}{4} (\gamma^n + \gamma^{n+1})$$

The displacement becomes :

$$\Delta u = t\tilde{v} + \frac{t^2}{4} \gamma^{n+1}$$

The quantity $\tilde{v}$ depends only on the step n.

The deformations are :

$$\Delta\varepsilon = B [t\tilde{v} + \frac{t^2}{4} \gamma^{n+1}]$$

In the special case of an isothermal compressible fluid the behaviour law is written :

$$\Delta p = - \rho c^2 \Delta\varepsilon$$

$$p^{n+1} = p^n - \rho c^2 B [t\tilde{v} + \frac{t^2}{4} \gamma^{n+1}]$$

Taking the movement equation :

$$\gamma^{n+1} = M^{-1} [F^{n+1}_{ext} + B^T p^{n+1} \Omega]$$

We obtain, after elimination :

$$[B M^{-1} B^T \Omega + \alpha I] p^{n+1} = \alpha p^n - \frac{4}{t} B\tilde{v} - BM^{-1} F^{n+1}_{ext}$$

In the special case of an incompressible fluid :

$$\alpha = \frac{4}{\rho c^2 t^2} \xrightarrow[c \to \infty]{} 0$$

After inversion of a quadratic matrix this equation thus gives the new pressures, which are entered into the movement equation.

This method is related to the interactions described in II.4.

IV. VALIDATIONS FOR THE FLUID ALONE

The possibilities of PLEXUS are shown by simple validation examples.

IV.1 - Shock tube

This classical problem is described in Fig. 1.

The tube is divided into two regions by a membrane which bursts at instant zero. Each region contains a perfect gas (P_2/P_1 = 10). The shock and expansion wave velocities as well as the pressure stage and the amplification due to reflection are computed correctly (Fig. 1). This is a hard test for the A.L.E. algorithm, given the velocities involved.

IV.2 - Decompression

A tank at constant pressure empties through a pipe 50 m long and 0.3 m in diameter. The end leads out to the atmosphere through a diaphragm ($\frac{D}{d}$ = 3) introducing a pressure loss. The transient regime involving first the to-and-fro of acoustic waves then the rise to speed of the fluid, which ends by reaching a permanent regime, is studied and the different phases are shown in Fig. 2.

This calculation was repeated for a fluid assumed incompressible, according to the algorithm described above. The agreement is remarkable, though only 60 time steps were needed instead of 3000 for the compressible fluid calculation.

IV.3 - Decompression and vapourisation

We tried to match by calculation the results of a depressurization experiment carried out at the D.E.M.T. This work is dealt with in another paper presented at the SMIRT [8]. The good agreement observed justifies the method adopted in PLEXUS.(Fig. 3)

V. FLUID STRUCTURE INTERACTION

The behaviour of both solid and fluid must be described in the same element. The basic element will be a two node segment similar to a beam in mechanics and to a fluid tube, i.e. seven degrees of freedom per node.

The interaction will affect both the masses, to account for the mass added by the fluid in a transverse direction, and the transport forces related to the movement of the structure. Here, the A.L.E. method proves invaluable. It remains to express the forces applied on the structure by the fluid in movement, done simply by writting the equilibrium of the solid + fluid system.

Let us take the case of a straight tube. During time **dt** the node **i** has moved by :

$$\vec{dl_i} = \vec{W_i}\ dt$$

For the degrees of freedom associated with the fluid we can write :

$$m_f\ \gamma_f = F_f$$

$$F_f = \int_{\partial\Omega} P\ d\ S \int_{\partial\Omega} \rho\ V\ (\vec{W}\ .\ \vec{n} - V)\ dS$$

Calculation of the transport forces only involves the axial component of $\vec{W}$. For the degrees of freedom associated with the structure the applied forces will be broken down into axial and normal forces :

$$(m_s + m_f)\ \gamma_s^{\vec{n}} = F_s^{\vec{n}} \qquad\qquad m_s\ \gamma_s^a = F_s^a - F_f$$

The interaction is then written very simply.

The boundary conditions will give specific interactions. Analysis of the bottoms for example implies writting the equality of fluid and structure axial accelerations. The Lagrangian multiplier method described in II.4 applies without trouble.

V.1 - Rocket

To illustrate this interaction we computed the behaviour of a simple tube closed at one end and open through a diaphgram at the other. The pressure is set for the element nearest the bottom, which amounts to introducing a source.

The fluid reaches velocity and the permanent regime set up applies a constant force to the structure. The rocket thus gathers speed regularly and the permanent regime is seen to hold well with respect to the moving structure (Fig. 4).

V.2 - Whip of pipes

The aim is to check that a guide tube of the core instrumentation system in 900 MWe power plants, subjected to 150 bars pressure at 290°C, will not if broken cause whip damage to neighbouring guide tubes.

Experiments have been carried out at the D.E.M.T. [6] to validate the PLEXUS programme for this problem, where the fluid-structure interaction is an important aspect. The first results are encouraging and the study is being continued.

VI. CONCLUSION

PLEXUS has been extensively developped to describe piping. Flows can already be analysed without fluid-structure interaction and the development of the coupled version has given very encouraging results. Application to an experimental validation is under way.

REFERENCES

[1] C. CHAVANT
"PLEXUS - A General Computer Code for Explicit Lagragian Computation"
Paper B2/8 - 5th SMIRT (1979)

[2] A. HOFFMANN, Ph. JAMET, M. LEPAREUX, A. MILLARD, B. BARBE, Ph. MAUREL
"Ultimate Flexural Behaviour of Reinforced Concrete Shells under Static and Dynamic Loading"
Paper J1/8 - 8th SMIRT (1985)

[3] A. HOFFMANN, M. LEPAREUX, B. SCHWAB, H. BUNG
"PLEXUS - A General Computer Program for Fast Dynamic Analysis"
Conference on Structural Analysis and Design on NUclear Power Plant - PORTO ALEGRE (BRESIL - 1984)

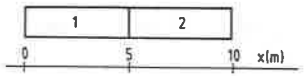
[4] J. DONEA, P. FASOLI-STELLA, S. GIULIANI, J.P. HALLEUX, A. V. JONES
"An Arbitrary Lagrangian Eulerian Finite Element Procedure for Transient Dynamic Fluid-Structure Interaction Problems"
Paper B1/2 - 4th SMIRT (1977)

[5] J. DONEA, S. GIULANI, J.P. HALLEUX
"An Arbitrary Lagrangian-Eulerian Finite Element Method for Transient Dynamic Fluid-Structure Interactions"
Second International Conference on Finite Elements in Nonlinear Mechanics, FENOMECH'81 - STUTTGART (1981)

[6] A. MILLARD, Ph. JAMET, J.L. LIEUTENANT, B. SCHWAB, D. GOETSCH
"Whip Analysis of Guide-Pipes of Instrumentation Below PWR Vessels"
Paper F1 4/7 - 8th SMIRT (1985)

PLEXUS

- Shock waves calculations -



<u>High pressure</u>: 1

P_1 = 10 MPa
ρ_1 = 0,1 kg/m^3
Perfects gas $\gamma_1=\gamma_2=1,4$

<u>Low pressure</u>: 2

P_2 = 1 MPa
ρ_2 = 0,01 kg/m^3

<u>Calculations</u>:

Artificial quadratic viscosity Q = 2 (von Neumann)

Pressure along the tube at 4 distinct times

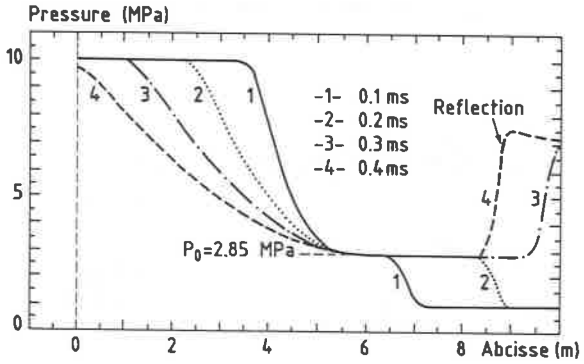
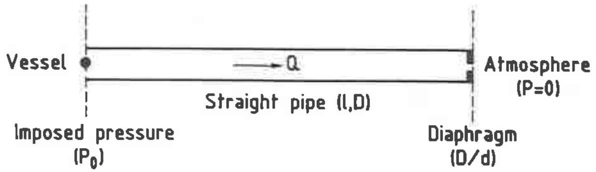


Fig. 1

PLEXUS

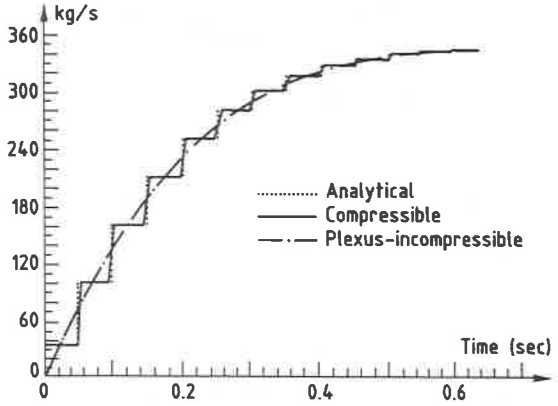
- Decompression of a vessel -



- l = 50 m
- D = 0.3 m
- d = 0.1 m

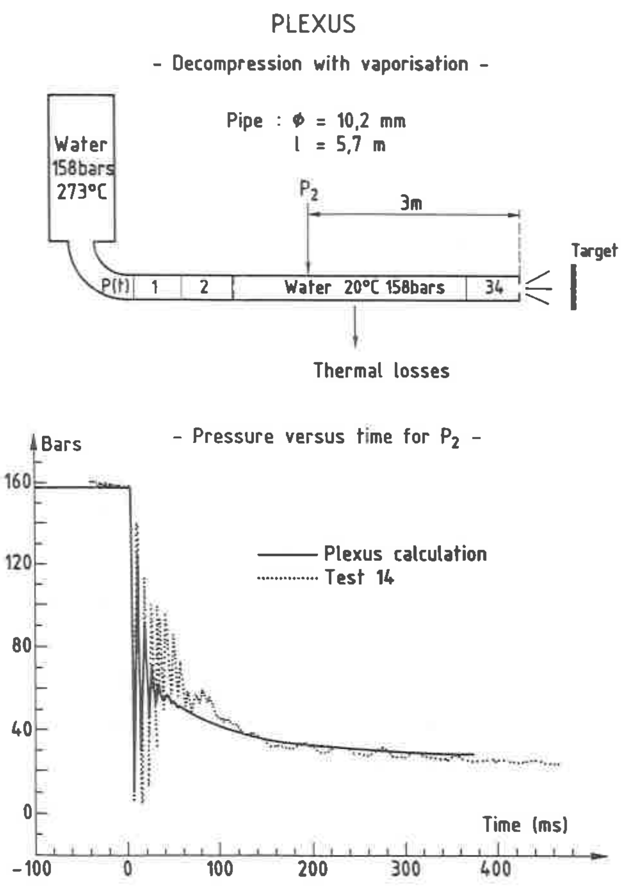
- P_0 = 10^6 Pa
- ρ_0 = 10^3 kg/m^3
- c = 2.10^3 m/s

<u>Results</u>: Mass flow



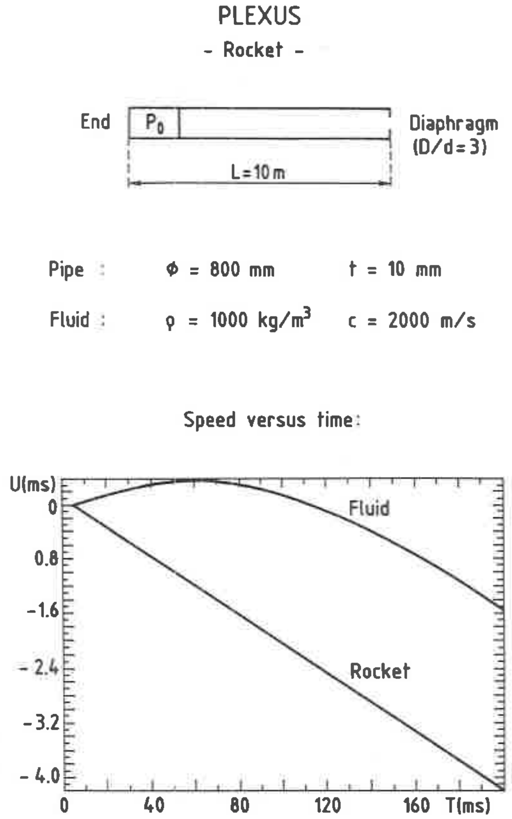
Mass flow versus time

Fig. 2



PLEXUS
- Decompression with vaporisation -
Water
158bars
273°C
Pipe : φ = 10,2 mm
l = 5,7 m
P2
3m
Target
P(t)
1
2
Water 20°C 158bars
34
Thermal losses
- Pressure versus time for P2 -
Bars
160
120
80
40
0
Plexus calculation
Test 14
Time (ms)
-100
0
100
200
300
400

Fig. 3



PLEXUS
- Rocket -
End
P0
Diaphragm
(D/d=3)
L=10 m
Pipe : φ = 800 mm t = 10 mm
Fluid : ρ = 1000 kg/m3 c = 2000 m/s
Speed versus time:
U(ms)
0
0.8
-1.6
- 2.4
- 3.2
- 4.0
Fluid
Rocket
0
40
80
120
160
T(ms)

Fig. 4