

Developments in the Analysis of 3D Piping and Shells by Means of PAULA Code

L. Lazzeri, M. Scala, M. Agrone
*AMN S.p.A.,
 Via G. D'Annunzio 113, I-16121 Genova, Italy*

SUMMARY

The 3D analysis of piping in the non linear field has been developed as a way to cope with the extreme loading conditions (pipe whip, earthquake, aircraft impact) postulated in the safety analysis of a nuclear power plant. PAULA code gives a way to analyze such conditions. The code represents a development of ADINA code whose characteristics of incremental non linear analysis have been retained, a special finite elements library (original) has been added as follows :

- PAP 3D : a non linear pipe element using a generalized St.Venant's formulation together with Prandtl-Reuss equations
 - SOAP : a non linear pipe element using an integral section approach together with normality
 - SAPPIPE: a linear pipe (tangent and bend) element using conventional beam's theory (with correction for bends) Shell elements (i.e. using shell's theory or modifications)
 - PETS: a shell element (original) using Vlasov's equations for strain-displacement, relationships, Iliouchine's yield locus for shells and normality rule together with isoparametric formulation, which is described in greater detail.
 - SAPEA : a shell superelement obtained by condensation (static or consistent) on any assembly of PETS elements and TEPS elements
 - TJPA : a special case of SAPEA where the geometry is the T joint
 - PAMEL : an elbow element formulated by means of Vlasov's theory, Fourier expansion of the displacements along the pipe circumference, Iliouchine's yield locus and normality rule and static condensation special elements (i.e. elements to cope with special situations)
 - TEPS : an element to cope with the transition between a beam element and shell elements (Vlasov's and Iliouchine's theory with isoparametric formulation)
 - IPRA 3: an element to study the interaction between pipe and restraint on the basis of experimental data
 - SMAUG : a non linear superelement obtained by condensation of any finite element assembly.
- For each of the elements the basic mathematical basis is described, the physical model is described and some examples are given. Particular attention is paid to the case of the yielding of shell elements assembly, examples are given and some conclusions are drawn, leading possibly to simplified analysis.

1. Introduction

Non linear analyses of three dimensional piping and shells are becoming more and more common, in the safety analysis of nuclear power plants. The pipe whip accident /1/, the Hypothetic core Destructive Accident (HCDA) /2/ for LMFBR represent two significative examples, where non linear analyses of the pressure boundary have been used with considerable success. Seismic analysis /3/ and other extreme loading of conditions /4/ are other cases, where non linear analyses have been used even if not extensively due to cost reasons.

The same authors have presented a code, named PAULA to deal with the 3D non linear analysis of piping /5,6/ ; it is the aim of this paper to briefly describe the basic library of PAULA and to describe the new shell elements in some more detail.

2. Finite Elements Library

2.1 Pipe Elements

The basic pipe elements are as follows

- PAP 3D /7,8/ : a non linear pipe element, using a generalized Saint Venant's formulation together with Prandtl-Reuss equations
- SOAP : a non linear pipe element using an integral section approach after Larson /9/ together with normality rule
- SAPPIPE : a linear pipe element (together with elbow flexibility correction factors)
- PAMEL /10/ : a non linear elbow element using shell's theory (see 2.2) and a generalized Iliouchine's /11/ yield locus together with normality rule.

The derivatives in the shell strains formulae are expressed by means of Fourier expansions and Lagrange polynomials. Static condensation is used to reduce the stiffness matrix (12x12)

- IPRA 3 : an element to study the interaction between a pipe and the restraints; experimental /12/ results are used in order to express the local crushing of the pipe zone of interaction with the restraint. Moment and moment-shear instability are also considered after Gerard /13/ and the results of ref./12/.

2.2 Shell Element PETS

All the shell elements use Vlasov's /14/ thin shell theory, which in matrix form is /15/

$$(1) \quad |\epsilon_i| = \sum_p \sum_q [S_{ik}^{pq}] \cdot \left| \frac{\partial \delta_k^{p+q}}{\partial \alpha^p \partial \beta^q} \right| \quad \begin{array}{l} p = 0,1,2 \\ q = 0,1,2 \\ p+q = 0,1,2 \end{array}$$

where

$|\epsilon_i|$ = generalized strain vector

$|\delta|$ = midthickness displacement

α, β = natural coordinates of the shells

$[S_{ik}^{pq}]$ = matrix, a function of shell geometry

The special matrices for the case of the torus (used in the elbow element PAMEL),

cylinder, cone and sphere are implemented in PETS element /16/; however the formulation is quite general and, provided that the relative routines are given, it may be used for any shell (Eulerian or not Eulerian). Isoparametric formulation is used to express the derivatives of the displacement at some points (Gauss points) as a function of the displacements of some other points (nodes): PETS elements are sixteen nodes generalized shell elements using quartic interpolation functions to formulate displacements and geometry; hence one has :

$$(2) \quad \left| \frac{\partial \delta_k^{p+q}}{\partial \alpha^p \partial \beta^q} \right| = [Z_{km}^{p,q}] \cdot |\delta_m|$$

If (1) and (2) are combined the strain vector is computed as a function of the displacements at the nodes :

$$(3) \quad |\epsilon_i| = [A_{i,m}] \cdot |\delta_m|$$

However it is convenient to perform a change of coordinates, so that rotations too are expressed, let θ be the vector of the rotations (around the natural shell coordinates); using isoparametric formulation one may compute :

$$(4) \quad |\theta_m| = [\xi_{m,n}] \cdot |\delta_n|$$

where the matrix $[\xi_{m,n}]$ depends from the actual shell geometry.

A generalized set of coordinates is defined $|\delta_j^*|$ as :

rotations of nodes 1,4

displacements of nodes 5,16

then by the use of correlation (3) one has :

$$(5) \quad |\delta_j^*| = [\eta_{jm}] \cdot |\delta_m|$$

Then if one may invert $[\eta_{jm}]$ (what depends from the actual shape function), one has :

$$(6) \quad |\epsilon_i| = [A_{i,m}] \cdot [\eta_{jm}]^{-1} \cdot |\delta_j^*|$$

In this way PETS element is a mixed displacement rotation element (somewhat similar to Semi-Loof element /17/); hence continuity is given partly for rotations (at the corner nodes) and partly for displacements (at the intermediate nodes).

Illiouchine's /11/ theory and normality rule are used to formulate :

$$(7) \quad |L_m| = [B_{mi}] \cdot |\epsilon_i|$$

where $|L_m|$ is the generalized load vector.

The stiffness matrix is formulated directly by means of the virtual work procedure :

$$(8) \quad [K] = \int_V ([A_{i,m}] [\eta_{jm}]^{-1})^T [B_{mi}] \cdot ([A_{i,m}] [\eta_{jm}]^{-1}) \cdot dV$$

The orientation matrices are computed directly by the program (what is depending from the actual shell geometry), referring the natural to the overall coordinate system.

2.3 Transition element TEPS

It has been mentioned that /15/ difficulties arise at the interface between shells and beams elements if the interface is not far away from the shell bending disturbances origin. Letting the details of the formulation to ref./15; 16/ it is enough to say that TEPS element solves the problem of the interface between shells and beams being a transition element with eight (two rotations and six displacements nodes) nodes at the shell end and the six section parameters (corresponding to the usual hypothesis of rigid cross section) at the beam end.

2.4 Shells Superelements : SAPEA, TJPA

It may be convenient to reduce the size of the problem by means of static condensation, as an example a T joint is a typical assembly of shell elements, however it may be convenient to reduce the active dofs to thirtysix (the six section parameters at each end). The reduction is performed by the program by means of Gauss elimination on the active columns scheme.

TJPA superelement is an assembly of shells to model a T joint ; the mesh is generated automatically by the program and Gauss reduction is performed to reduce the dofs to the three section parameters. SAPEA is a superelement , where any assembly of shell elements (PETS and TEPS elements) is reduced to any desired set of dofs.

2.5. General Superelement : SMAUG

A generalized superelement is available; it assumes that the stiffness matrix $[K]$ is a function of two basic loading parameters L_1, L_2 ; the element has two phases a registration phase and a recovery phase. In the registration phase the stiffness matrix is recorded for different values of the two parameters L_1, L_2 ; in the recovery phase the stiffness matrix of the superelement is found as a function of the actual values of the two loading parameters by means of an interpolation procedure.

3. Applications and discussions

Some applications of PAULA code for the cases of the pipe whip problem or the seismic case have been described extensively in references /15,16,18,5,6,4/; comparisons are also given with simplified codes for the special case of the pipe whip problem.

In this case some other examples are shown for the cases of pipes and shells. A rather complicated 3D line is shown in fig.2, a break is postulated at one terminal end as it is typical of the pipe whip accident. The results of the transitory in terms of displacement of the terminal end, forces on the restraints are shown in figs 3,4.

It is rather interesting to discuss the non linear behaviour of a shell assembly; in fig.5 a typical case of a nozzle on a spherical shell is shown . In fig.6 the results of an elastic analysis are compared with the ones obtained by means of BOSOR code /19/, as one can see the results compare rather well even if the size of the PETS element is quite large

(more than half the wave length of the shell). In fig.7 one finds the force-displacement correlation for this case showing non linear behaviour. However it is interesting to note that :

- the first plastic flow is due to secondary moment effects (i.e. not essential for the equilibrium but due to compatibility requirements)
- overall yielding is somewhat slower and the essential membrane components are not generally capable of producing plastic flow

Hence the overall behaviour of the shell assembly is such that it exhibits a quasi-linear behaviour well beyond yielding due to the moment components, as clearly shown in fig.7.

From that it follows that when dealing with extreme loading conditions of steel components (or other ductile materials), it may be not essential a completely correct and accurate reproduction of the bending behaviour, as it would be the case in other load cases (e.g. fatigue analysis). Rather, the membrane state only, which is much easier to be reproduced must be correctly and accurately modelled. Hence rather coarse meshes might be used as the overall behaviour is reproduced and some error on the yielding due to combined bending actions may be often tolerated.

Besides PETS element is "per se" a rather powerful shell element : one element is sufficient to satisfactorily deal with the whole bending transitory, so that the meshes are often quite coarse. As an example TJPA element is made up by only six elements but it generally gives quite reasonable predications /16,18/.

A last word may be spent about the use of Iliouchine's yield locus ; as it has been discussed in many cases /5,6,10/ it is approximate as it assumes an average strain through the thickness in computing the strain hardening parameters. However its simplicity, the uncertainties in the strain hardening rules, and the considerations just made make its use quite reasonable.

4. Conclusions

The special finite elements library of PAULA code has been described and discussed. The special non linear pipe items (straight pipe, elbow, pipe-restraint interaction) are briefly described; some more detail is given to the shell element :

- Vlasov's equations (valid for any shell) are referred to in order to compute the strain-displacements correlations
- isoparametric formulation is used to express the derivatives (up to the second order) in terms of nodal displacements
- Iliouchine's yield locus and associated normality rules are used to express stress-strain relationships
- a change of coordinates is performed to have a mixed formulation in terms of rotations

and displacements

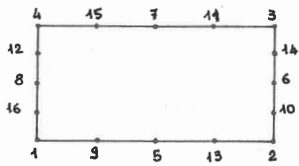
Some superelements are available to reduce the number of active dofs, static condensation by means of Gauss elimination on the active columns scheme is used.

Some examples are given . Particular attention is given to the case of the non linear response of shell assemblies; it is shown how first yield is generally due to local bending actions, which are however not essential to correctly model the overall behaviour of the assembly. Hence if the extreme loading capability of a ductile structure is under examination, a rather coarse mesh may be tolerated to correctly reproduce the membrane actions with some inaccuracy in the bending predictions.

References

- /1/ Lazzeri L. "Pipe whip analysis" chapter of advanced Dynamic Analyses, Applied Science Editors 1978
- /2/ Donea J. "Finite elements analysis of transient dynamic fluid structure interaction" ibidem
- /3/ Powell G.H. "Inelasti pipe element for the analysis of pipe whip SMiRT 4, San Francisco 1977, paper F3/7
- /4/ Giuliano V., Lazzeri L., Scala M. "3D analysis of piping in the non linear field under extreme loading conditions" paper K10/11, 6 SMiRT Conf., Paris 1981
- /5/ Lazzeri L., Scala M., Agrone M. "On the 3D non linear analysis of piping " Nuclear Engineering and Design Vol 61, No 2, Nov. 1980
- /6/ Lazzeri L., Scala M., Agrone M. "PAULA a 3D code for the non linear analysis of pipes" ASME paper 80/C2/149, PVP Conference, San Francisco , 1980
- /7/ Lazzeri L. "A model for a strain hardening pipe element" ASME paper 77-PVP-47
- /8/ Lazzeri L. "Application of plastic analysis to the pipe whip problem" Int. J.Pressure Vessel and Piping (7), 1979
- /9/ Larson L.D., Stokey W.F., Franzen W.F. "An approximate model for anelastic plastic pipe element under combined loading" Trans. ASME 97(1), Feb.1975
- /10/ Lazzeri L. "An elastoplastic elbow element : theory and applications" (8), 1980 197-213
- /11/ Iliouchine A.A. "Plasticité" ed. Eyrolles, Paris , 1957
- /12/ Ikonen K., Kukkola K., Kangas M.K. "Local crush rigidity of pipes : experiments and applications to pipe whip restraint design" F6/4, 5 SMiRT, Berlin 1979
- /13/ Gerard G. "Handbook of structural stability " NACA-TN 3783
- /14/ Vlasov V.Z. "General theory of shells and its applications in engineering" transl. from Russian, NASA, N64, 19833
- /15/ Lazzeri L. "Developments in the non linear analysis of pipes and shells" Internal AMN document
- /16/ Lazzeri L. "On the non linear analysis of shells - Part 1 : theory" submitted for presentation to ASME
- /17/ Irons B.M. "The semiloof shell element" Finite elements for thin shells and curved members " Gallager and Ashwell ed. Wiley, London

- /18/ Lazzeri L. Scala M. "On the non linear analysis of shells - Part 2 : Applications" submitted for presentation to ASME
- /19/ Bushnell " Buckling of elastic-plastic shells of revolution with discrete elastic plastic ring stiffness" - Part.3 of rep LMSC-D-407168, Dec. 1974.



PERS element geometry

Fig.1

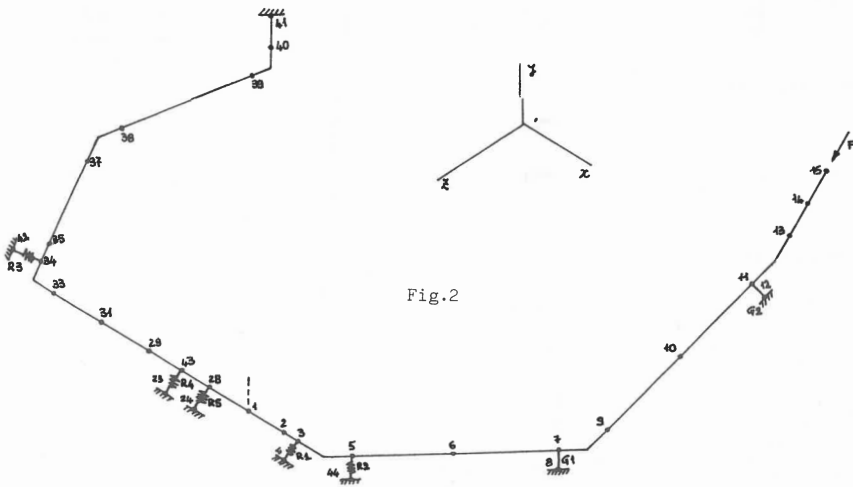


Fig.2

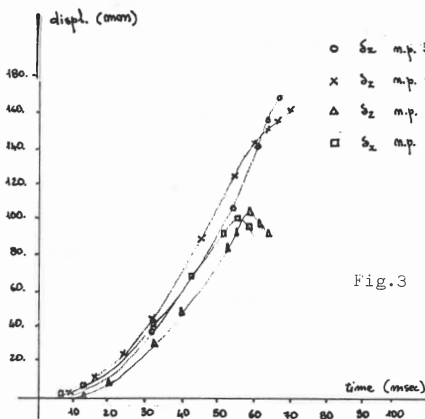


Fig.3

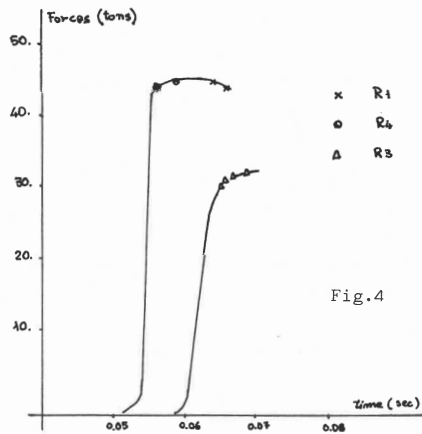
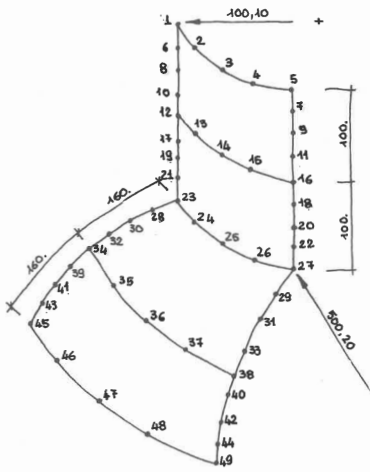


Fig.4



Nozzle geometry

Fig.5

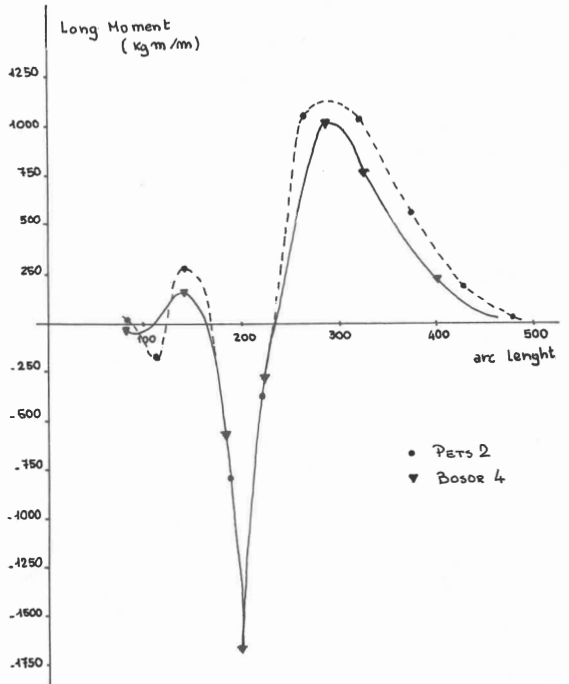
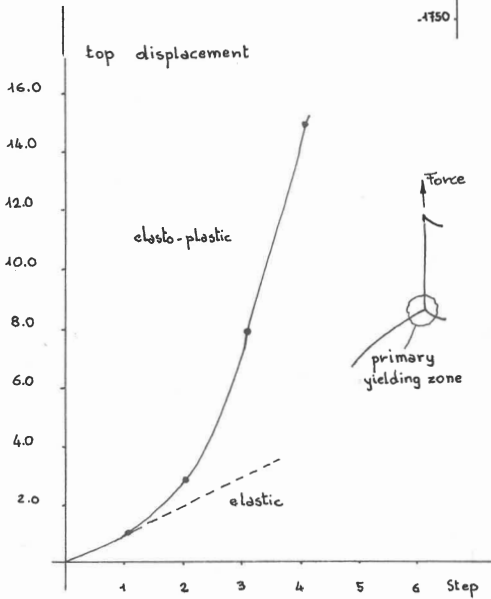


Fig.6a



Non linear behaviour of a nozzle

Fig.7

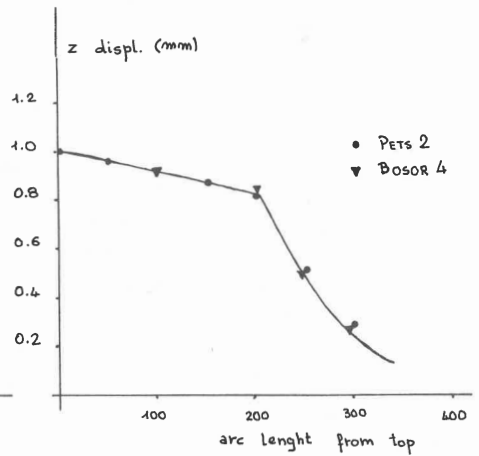


Fig.6b