NUMERICAL SIMULATION OF THE BEHAVIOUR OF RUBBER SEALING JOINTS IN LARGE CONTAINMENT PENETRATIONS

FERNANDO CASTRO LÓPEZ - ALFREDO ORDEN MARTÍNEZ

INTRODUCTION

In the hypothetical case of a severe accident in a nuclear power plant, the environmental impact depends on the integrity of the containment system. This is the last barrier which prevents the release of radioactive material to the environment.

If the integrity of the containment pressure retaining boundary is assured, the level of leaks to the outside will fundamentally depend on the seals of the rubber joints of the large penetrations, like the equipment hatch and the personnel access locks.

The new severe accident sequence postulated can generate pressure and temperature transients which are greater than those for which the containment system was originally designed. For this reason, the behaviour of the rubber sealing joints of the above mentioned penetrations could be affected, resulting in an increase in the level of leaks through these joints due to greater degradation of the material and greater local deformations in the closure areas.

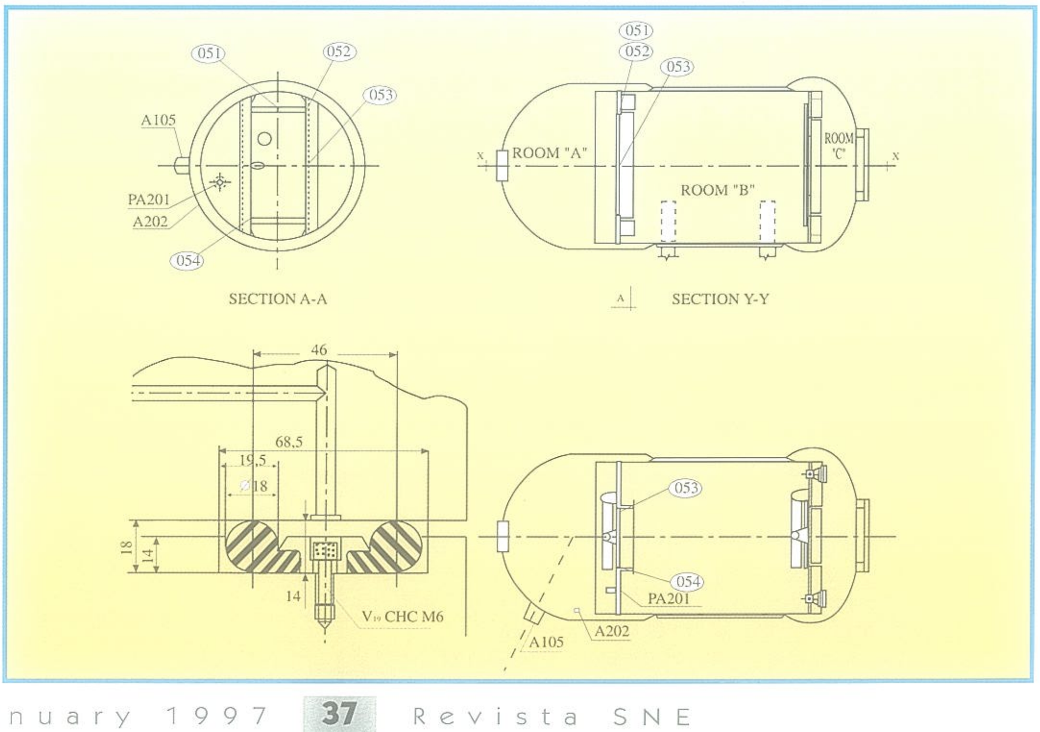
At the present time EMPRESARIOS AGRUPADOS is participating with ENEL (leader), EDF, CIRTEN and CISE in the ATHERMIP project. This project is 50% financed by the European Community Commission (DG-XII-F). The main objective of the project is to develop, and validate by tests, a theoretical approach and the related computer model to simulate the thermo-mechanical behaviour of the largest containment penetrations in which the leaktightness function depends on the performance of the elastomeric materials. The load conditions (pressure and temperature) will be greater than in the original design.

The material used in the design of these joints was silicone (methyl-vinyl-silicone). The stress-deformation and deformation-temperature responses of this material have been characterised using a series of tests. In the relevant pressure and temperature range the behaviour of the material was elastic non-linear, ie no residual deformations but with rigidity varying as a function of the level of deformation and temperature.

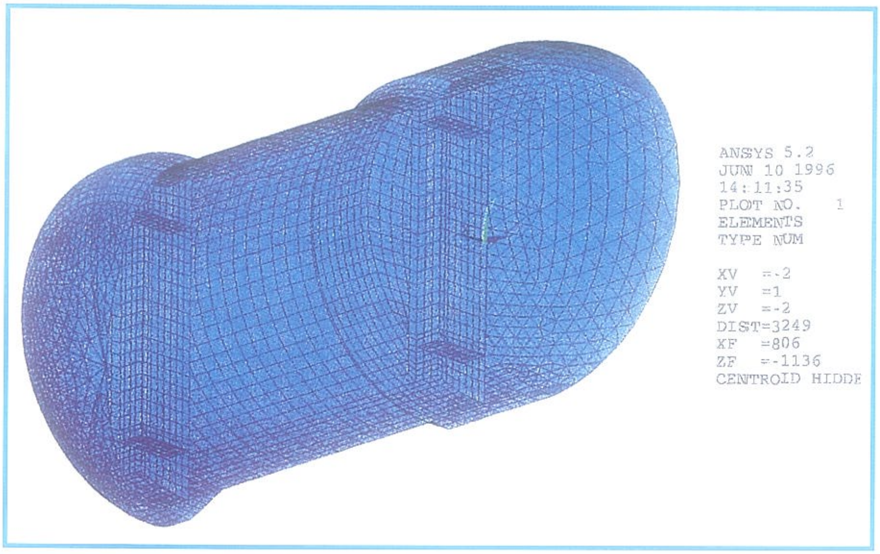
This fact, along with the ageing of the material caused by the environmental variables (temperature, radiation) means that advanced numerical techniques are required to simulate the behaviour of the sealing joints.

The calculation program used in the analysis of the thermo-mechanical behaviour of the joints was version 5.2 of the ANSYS commercial program. The Mooney-Rivlin option of this program was used to represent the material of the

F 1 - *Test Prototype and Sealing Joints*



F II - *Global 3D Finite Element Model*



sealing joints; the non-linearities (contacts/separations, large deformations, etc) in the calculation model were handled in the same way by the standard options in the calculation program. Special attention is required when selecting the elements indicating contacts or leaks, as these elements will largely depend on the qualification of the joints.
The analytical model validations are based on a full-scale test campaign to be performed on the personnel airlock of the never completed Alto Lazio NPP (BWR, GE, Mark III) in Italy.

DESCRIPTION OF THE TEST PROTOTYPE

Figure I shows the test prototype. Two caps have been welded at both ends of the personnel locks in order to be able to pressurise both doors independently. Similarly, the accessories required to control the pressure and the temperature, and the installed instrumentation (manometer, thermometer, thermocouples, displacement transducers and strain gauges) have been placed in both chambers.
The test prototype is anchored to the floor using two supports.
The prototype was designed in such a way that the structural behaviour of the door and of the bulkhead will not be affected, and will faithfully reproduce the behaviour of the personnel lock installed in the containment.
The closure area and the housing of the sealing joints is not modified with respect to the original design (Fig I).

DESCRIPTION OF THE SEALING JOINTS

The sealing joints considered in the analyses have a transversal section quite similar to an "O-ring" and are housed inside a cavity in the exterior perimeter of the closure doors of the personnel lock. The position and geometry of these joints are shown in Figure I.
The material used in the manufacture of these joints was silicone and the thermo-mechanical characterisation was based on experimentation. By means of uniaxial, equibiaxial and plane traction tests (ASTM D-395 and D-412), the stress-deformation response curves were obtained at different temperatures and as a function of the ageing of the joints.
The thermal expansion coefficients of the material and their evolution with temperature and ageing were also obtained.
According to the data supplied to the analyst, the elasticity module at the origin is around 2.3 MPa at ambient temperature and without considering exposure to radiation.
The range of relevant deformations reaches 700%, for which the tangential elasticity module could be reduced by up to 90% with respect to the value corresponding to a null deformation.

STATEMENT OF THE PROBLEM

In general, two conditions must be taken into account when analysing the leaktightness between the sealing surfaces of large containment penetrations. Firstly, the relative movement of the metal contact surfaces caused by the applied load or by the initial conditions. Secondly, the thermo-mechanical behaviour of the sealing joints material, both in normal operation and in the event of severe accident.
The relative movement of the contact surfaces can arise as a result of phenomena such as:

- Initial gap caused by the closing mechanism (insufficient compression of the sealing joint)
- Internal pressure causing bending in the closure door and bulkhead
- Relative warping between the metal surfaces

In practice, the relative movement between the two surfaces is calculated as a function of pressure and temperature by means of numerical simulation (finite element codes).
The operation of the sealing joints largely depends on phenomena such as:

- Ambient temperature and radiation (ageing)
- Effects of oxidation

In practice, the operation of the joints in combination with the above phenomena is determined experimentally and, in parallel, analytically using the data generated during the tests.

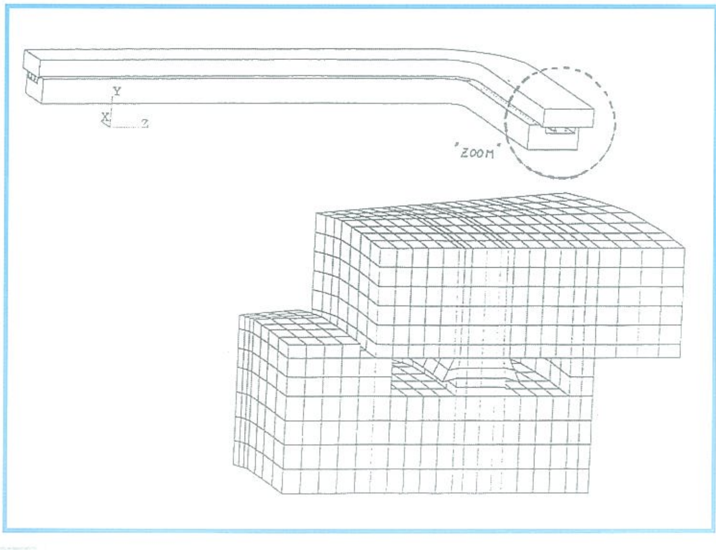
NUMERICAL TREATMENT

The calculation model (finite element model) required to simulate in an approximate manner the sealing joints inside their cavity must take the following two aspects into account:

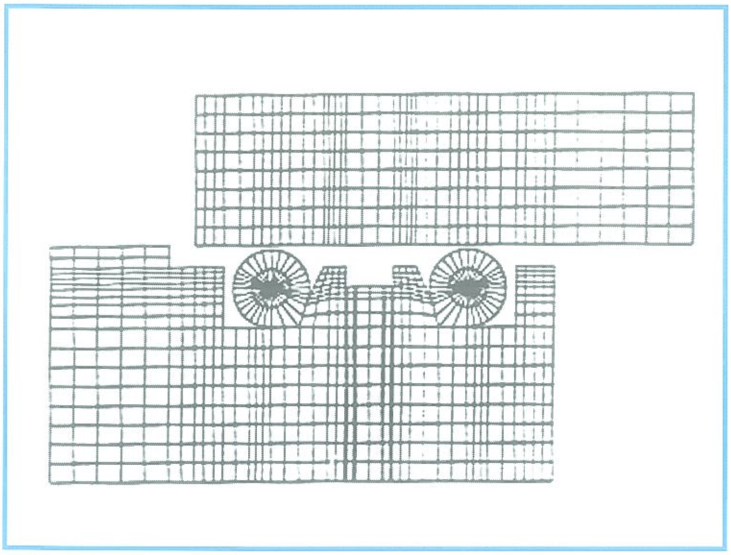
- The non-linearity of the joint material
- The interaction between joints, joint cavity and sealing surfaces

Version 5.2 of the ANSYS program, used in the numerical simulation of the problem, gives the user the possibility of handling these aspects by means of the standard program options.
The behaviour of the joint material is described using a hyper-elastic model. This model handles the material in terms of a deformation energy potential, which defines the deformation stored in the material as a function of the deformation. The potential used in the simulation was the Mooney-Rivlin, which is expressed as follows:

F III - *Local 3D Finite Element Model*



F IV - *2-D Finite Element Model*



$$W = \sum_{k+l=1}^{N} a_{kl} (\bar{I}_1 - 3)^k (\bar{I}_2 - 3)^l + 1/2k (\bar{I}_3 - 1)^2$$

where W is the deformation energy, N is a parameter of the material, a_{kl} are constants obtained from the results of the tests, I_1, I_2, I_3 are the invariants of the deformation tensor and K is a parameter which depends on the ratio of the elastic volume without considering the effects of thermal expansion.

The contact/separation areas involved in the numerical simulation were handled by elements indicating contact or leak, which are included in the calculation program, especially contact surfaces and gap elements.

This numerical treatment, along with the consideration of large deformations, is being used to analyse the behaviour of the joints and of the possible failure modes for this type of joint.

APPLICATION

Following are the calculation models which are being used to analyse the conditions described above in sealing joints of the personnel lock. These calculation models which are used to predict the behaviour of the elastomeric joints are being validated by means of a series of tests which are carried out with a full scale prototype of the personnel lock described above.

Figure II shows the 3D model developed by CIRTEN in order to verify the global behaviour of the test prototype. The results of this model will be the movement of the bulkhead and of the door for each test case.

Figure III shows the local 3D model of the closure area developed by EMPRESARIOS AGRUPADOS. The nodes corresponding to the movements obtained in the complete model of the test prototype are introduced in this model. The main objective of the model is to analyse the deformed geometry and the relative movements of the door and bulkhead in the closure area. This calculation model must allow the least favourable sections to be selected, ie those section where the relative rotation between the metal components is greatest.

Figure IV shows the second 2D calculation model developed by EMPRESARIOS AGRUPADOS. The basic objective of this model is to analyse the behaviour of the sealing joints for each one of the sections selected with the previous model.

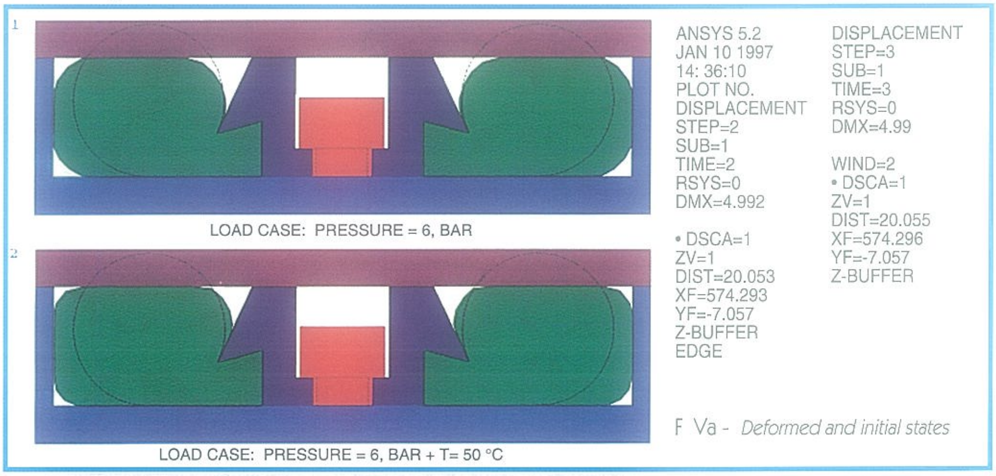
Figure V shows the behaviour of the model for the cases analysed previously with a pressure of 6 bar and a temperature of 50 °C.

TEST CAMPAIGN

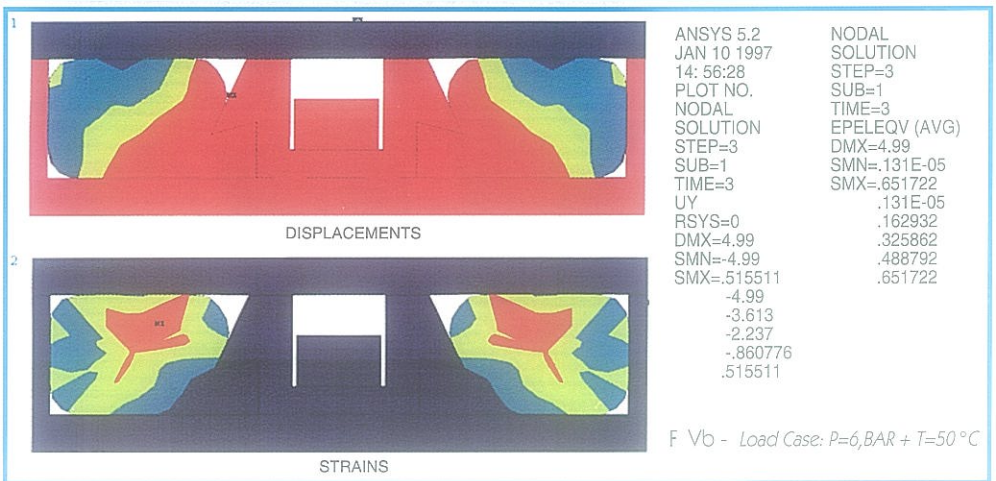
ENEL and CISE will carry out the test campaign with the full-scale test prototype in the first quarter of 1997. In parallel, the numerical analyses will be carried out with the analytical models described in the previous section, reproducing the ambient conditions of each test and of the state of the material of the sealing joints.

During the comparison of the analysis and the test firstly a check will be made to ensure that the strain level is the same for both cases. Then the movements calculated with the CIRTEN model will be superimposed onto the 3D model developed by EMPRESARIOS AGRUPADOS. The most critical section will be selected and the behaviour of the sealing joints will be studied using the model developed by EMPRESARIOS AGRUPADOS. The

F V - *Sealing Joint Behaviour*



F Va - *Deformed and initial states*



F Vb - *Load Case: P=6,BAR + T=50 °C*

potential leak areas due to lack of contact between rubber joint and metal surface will be determined.

CONCLUSIONS

The discussion in this paper, as in many other cases, shows how a numerical simulation based on experimental data applied to specific design problems with incompressible elastic materials allows analyses to be carried out and the thermo-mechanical behaviour of a mechanical component to be determined. The efficiency of the numerical process will be checked during the test campaign to be carried out in 1997.

REFERENCES

1. *ANSYS User Manual Volume IV Theory;* Version 5.2 SAS IP 31 August 1995
2. Thomas J R Hughs. *The Finite Element Method. Linear, Static and Dynamic Finite Element Analysis*
3. *Containment Performance Working Group Report.* US Nuclear Regulatory Commission
4. *Seals & Sealing Handbook.* 3rd edition. Melvin W Brown

Fernando CASTRO LOPEZ graduated as an Industrial Engineer from the ETSII, Zaragoza, in 1978.
He began his professional career in the ETSII in 1986/87 working on projects related to numerical simulation. In 1988 he joined Applied Mathematics and Computer Methods Department of the ESTIM at UPM, as a collaborator; he worked on several R&D programmes related to the numerical simulation of different physical phenomena.
He has been working in the Component Analysis Section at EMPRESARIOS AGRUPADOS since 1990 as Project Engineer in the nuclear and space sectors.

Alfredo ORDEN MARTINEZ graduated as an Industrial Engineer from the ETSII, Madrid, specialised in mechanics in 1978.
In 1980 he joined the Component Analysis Section at EMPRESARIOS AGRUPADOS and was the engineer responsible for the Containment Building at Valdecaballeros NPP.
He has been in charge of the Component Analysis Section of the Mechanical Department since 1987 and at the present time he is involved in the Tacis and Phare programmes related to the improvement of containment systems and buildings in pressurised water power plants designed in the former USSR.
He has been collaborating with the IAEA since 1991 as a containment building design and testing expert.