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

Reliability of the repairing of double wall containment vessels in the context of leak tightness assessment

Bruno Sudret[1)], Marc Berveiller[1)], Grégory Heinfling[2)]

[1)] Electricité de France – R&D Division – Site des Renardières – 77818 Moret-sur-Loing Cedex, FRANCE
[2)] Electricité de France – SEPTEN – 12-14 Av. Dutriévoz – 69628 Villeurbanne Cedex, FRANCE

ABSTRACT

A probabilistic framework is developed to assess the design of a composite liner which is used in order to restore the air tightness of concrete containment vessels of French 1300 MWe nuclear power units. The liner is designed by computing the "tensile stress area" around the material hatch area under pressurization of the vessel. These tensile stresses indeed are favourable to the opening of cracks which degrade the air tightness of the vessel. Supposing now that the design parameters (material properties of concrete and pre-stressing cables) are random, the probability that tensile stresses appear *outside* the region covered by the liner is computed. The paper presents the use of a finite element reliability approach to solve the problem. This is performed by coupling the probabilistic code PROBAN with EDF's finite element code Code_Aster©.

KEY WORDS: containment vessel / leak tightness assessment/ finite element reliability / FORM-SORM

INTRODUCTION

The containment of French 1300/1450 MWe pressurized water reactors is ensured by two concrete vessels. The outer one, designed to withstand external aggressions, is made of reinforced concrete. The inner containment is designed so as to avoid leakage of radioactive elements in case of LOCA (loss of coolant accident). The leak tightness does not rely on a steel liner as in other containment vessels, but on the prestressed concrete itself. The containment is biaxially prestressed so that it remains in compression under the pressure and temperature loading associated with LOCA. The space left between the two containment is kept in depression. Thus in case of unsatisfactory leak tightness, the air/vapour mixture that may have flown into the space between the two walls is pumped back into the inner containment.

To assess the leak tightness of the inner vessel, full-scale experimental tests are performed periodically, namely at the end of the civil works, after 4 years and then every 10 years. These tests are carried out at the ambient temperature at a pressure close to the design pressure, in a conservative way compared to LOCA. The average leak from the inner containment to the interspace is measured and should fulfil a prescribed criterion.

In the recent years, this leak tightness criterion has not been satisfactorily fulfilled for few French containment vessels. This was mainly due to macroscopic cracks that had developed in specific areas of the wall (material hatch area, gusset) and that were clearly identified. A procedure has been developed by EDF and accepted by the Safety Authority to solve the problem :

- a finite element computation of the prestressed concrete vessel under the test pressure is carried out and those areas of the intrados of the wall that undergo tension at the pressure peak are identified.
- a composite liner is applied onto the intrados over a surface that encompasses the computed "tensile stress area", in order to restore the air tightness.
- another full-scale test is carried out to confirm the efficiency of the liner.

The aim of this paper is to develop a probabilistic methodology of assessment of this repairing solution which will allow to take into account the uncertainties in the design parameters. Precisely, once the liner geometry has been determined, the probability of having tensile stress and possible crack openings *outside* the covered area will be computed. The analysis allows also to determine which parameters are the most important in the analysis (sensitivity analysis). A parametric study finally shows the reliability associated with different geometries of the composite liner.

DETERMINISTIC STRESS ANALYSIS OF A CONTAINMENT VESSEL

A finite element model of ¼ of a French 1300MWe concrete containment vessel has been developed using EDF's own finite element code called Code_Aster© (This code can be downloaded for free at `http://www.code-aster.org`). The model corresponds to azimuth $. \; . \; [0\,;180\theta\,]$ and altitude $z \, . \; [30,3\,\mathrm{m}\,;54,15\,\mathrm{m}]$. The main cylindrical part has an internal radius of 22.5 m and a thickness of 0.9 m. The material hatch is bored into this cylinder around an horizontal axis located at the altitude $z = 30.3$ m. Details are given in [1] and in Figure 1. The dome of the vessel is not represented, it is replaced by equivalent boundary conditions (see below). The mesh of the structure comprises the following elements (Figure 2):

- quadratic isoparametric 20-node elements for concrete (6 elements within the thickness of the wall),
- horizontal and vertical pre-stressing cables: the exact geometry of each cable is used around the material hatch area, whereas the straight cables are gathered in "equivalent" cables in the main cylindrical part.
- reinforcement bars that are meshed using bar elements.

The model comprises 19769 nodes and 14412 elements as a whole.

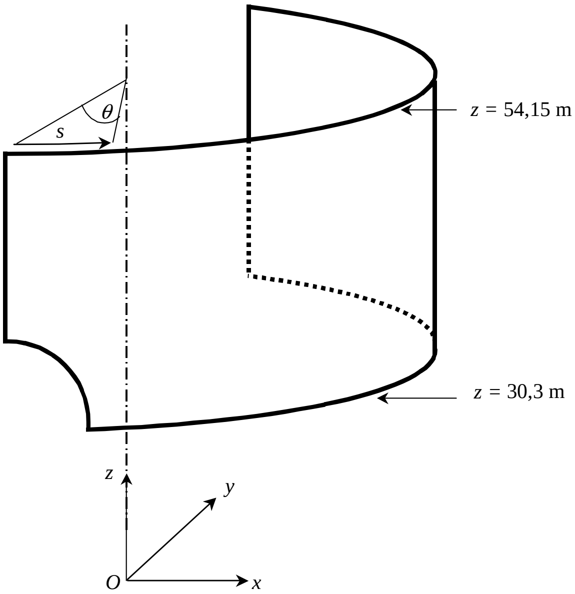
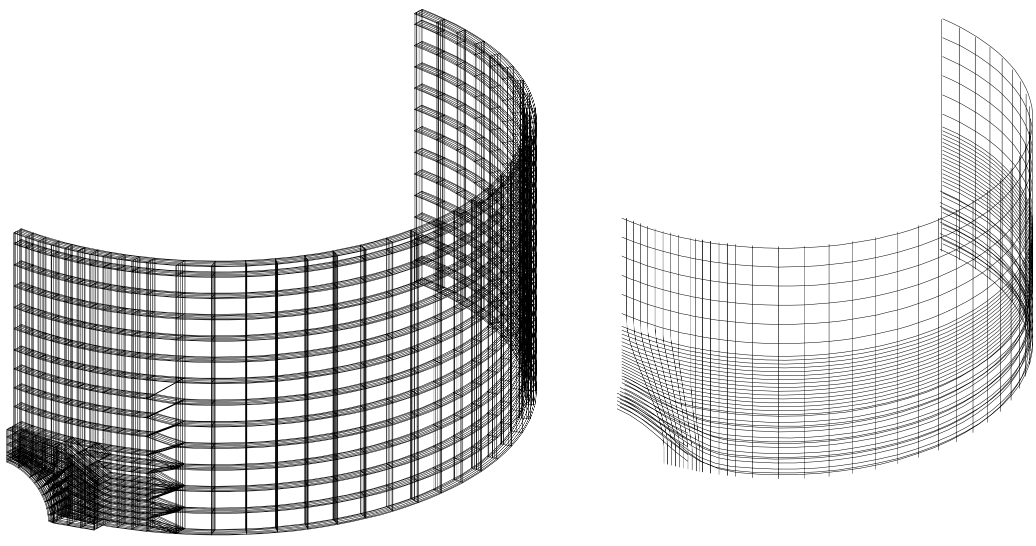


Fig.1: Sketch of the model of containment vessel



(a) Concrete wall (b) Pre-stressing cables

Fig.2: Mesh of the structure

The constitutive law of each material (steel, concrete) is supposed to be *linear elastic*. The related parameters are given in Table 1.

Tab. 1: Material parameters (concrete and pre-stressing cables)

Material	Parameter	Value
Concrete	Young's modulus E_b	31,000 MPa
	Poisson's ratio ν_b	0.22
	Mass density ρ_b	2,500 kg/m^3
Steel	Young's modulus E_a	192,000 MPa
	Mass density ρ_a	7,850 kg/m^3

The following loading is considered:

- the dead load of the structure, computed from the mass density of each material as given in Table 1. The additional load of the dome is taken into account by a uniform pressure applied onto the top layer of the concrete mesh.
- the tensioning of the cables: different nominal tensions are used according to the cables under consideration. The tension profiles of the cables are computed according to the French design code BPEL [2] by accounting for pre-stress losses due to friction.
- a uniform pressure $P_0 = 0.38$ MPa applied onto the intrados of the mesh. The effect of this pressure onto the dome is taken into account by a *negative* pressure applied onto the top layer of the concrete mesh. The effect onto the supposedly closed hatch is represented by a set of nodal forces applied onto the nodes of the boundary of the hatch.

The following boundary conditions are imposed:

- UX = 0 at the vertical generatrices $\theta = 0°$ and $\theta = 180°$ due to vertical symmetry,
- UZ = 0 at the lower surface of the mesh z = 30.3 m due to horizontal symmetry,
- UX = UY = 0 at the upper surface of the mesh z = 54.15 m in order to take into account the stiffening effect of the dome (UZ being kept free).

PRINCIPLES OF STRUCTURAL RELIABILITY ANALYSIS

Problem statement

Structural reliability analysis aims at computing the probability of failure of a mechanical system with respect to a prescribed failure criterion by accounting for uncertainties arising in the model description (geometry, material properties) or the environment (loading). Let us denote by $\underline{X}(\omega) = \{X_1(\omega), X_2(\omega), ..., X_n(\omega)\}$ the set of random variables describing the randomness in the geometry, material properties and loading. The failure criterion under consideration is mathematically represented as a *limit state function* $g(\underline{X})$ defined in the space of parameters as follows :

- $g(\underline{X}) > 0$ defines the *safe state* of the structure
- $g(\underline{X}) \leq 0$ defines the *failure state*. In a reliability context, it does not necessarily mean the breakdown of the structure, but the fact that certain requirements of serviceability or safety limit states have been reached or exceeded.
- $g(\underline{X}) = 0$ defines the *limit state surface.*

Denoting by $f_{\underline{X}}(\underline{x})$ the joint probability density function of random vector $\underline{X}$, the probability of failure of the structure is :

$$P_f = \int_{g(\underline{x}) \leq 0} f_{\underline{X}}(\underline{x})\, d\underline{x} \tag{1}$$

In all but academic cases, this integral cannot be computed analytically. Indeed, the failure domain is often defined by means of *response quantities* (e.g. displacements, strains, stresses, etc), which are computed by means of computer codes (*e.g.* finite element code) in industrial applications, meaning that the failure domain is *implicitly* defined as a function of $\underline{x}$. Thus numerical methods have to be employed.

FORM method

The *First Order Reliability Method* has been introduced to get an approximation of the probability of failure at a low cost (in terms of number of evaluations of the limit state function). The first step consists in recasting the problem in the standard normal space by using a *probabilistic transformation* $\underline{X} \to \underline{U} = T\left(\underline{X}\right)$. The Rosenblatt or Nataf transformations may be used for this purpose ([3], chap. 7). Thus Eq.(1) rewrites:

$$P_f = \int_{g(\underline{x})\le 0} f_{\underline{X}}(\underline{x})\, d\underline{x} = \int_{g(T^{-1}(\underline{u}))\le 0} \varphi_n(\underline{u})\, d\underline{u} \qquad (2)$$

where $\varphi_n\left(\underline{u}\right)$ stands for the standard multinormal probability density function (PDF):

$$\varphi_n(\underline{u}) = \frac{1}{\left(\sqrt{2\pi}\right)^n} \exp\left(-\frac{1}{2}\left(u_1^2 + \cdots + u_n^2\right)\right) \qquad (3)$$

This PDF is maximal at the origin and decreases exponentially with $\left\|\underline{u}\right\|^2$. Thus the points that contribute the most to the integral in Eq.(2) are those of the failure domain that are closest to the origin of the space.

The second step in FORM thus consists in determining the so-called *design point*, *i.e.* the point of the failure domain closest to the origin in the standard normal space. This point P^* is obtained by solving an optimisation problem :

$$P^* = \underline{u}^* = Arg \min\left\{\left\|\underline{u}\right\| \;/\; g\left(T^{-1}(\underline{u})\right) \le 0\right\} \qquad (4)$$

Several algorithms are available to solve the above optimisation problem, *e.g.* the Abdo-Rackwitz or the SQP (sequential quadratic programming) algorithm [4]. The search for the design point is iterative and thus requires several evaluations of the *g*-function as well as its gradient with respect to the basic parameters. These gradients can be computed by a finite difference scheme if not implemented directly in the code. The corresponding reliability index originally proposed by Hasofer & Lind [5] is defined as :

$$\beta = \text{sign}\left[g(T^{-1}(0))\right] \cdot \left\|\underline{u}^*\right\| \qquad (5)$$

It corresponds to the algebraic *distance* of the design point to the origin, counted as positive if the origin is in the safe domain, or negative in the other case.

The third step of FORM consists in replacing the failure domain by the half space *HS*(*P**) defined by means of the hyperplane which is tangent to the limit state surface at the design point (see Figure 3).

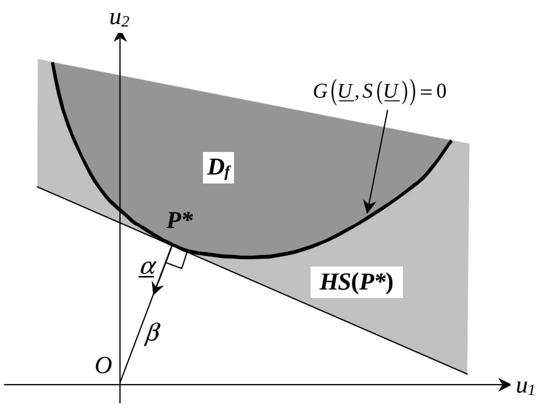


Fig. 3: Principles of FORM method

This leads to :

$$P_f = \int_{g(T^{-1}(\underline{u}))\le 0} \varphi_n(\underline{u})\, d\underline{u} \;\approx\; \int_{HS(P^*)} \varphi_n(\underline{u})\, d\underline{u} \qquad (6)$$

The latter integral can be evaluated in a closed form and gives the first order approximation of the probability of failure :

$$P_f \approx P_{f,FORM} = \Phi\left(-\beta\right) \qquad (7)$$

where Φ denotes the standard normal cumulative distribution function(CDF).

Application of FORM in the context of finite element analysis

In this study the probabilistic code PROBAN [6] is used to solve the optimisation algorithm associated with FORM. When the quantities involved in the limit state function are obtained numerically, a coupling between this code and the finite element code has to be implemented [7,8]. This is performed by the implementation of a FORTRAN routine `g.f` that computes the value of g from the current values of the design parameters by calling the finite element code Code_Aster© and post-processing the results. This coupling is sketched in Figure 4, where the user-defined library `sublib.f` also appears.

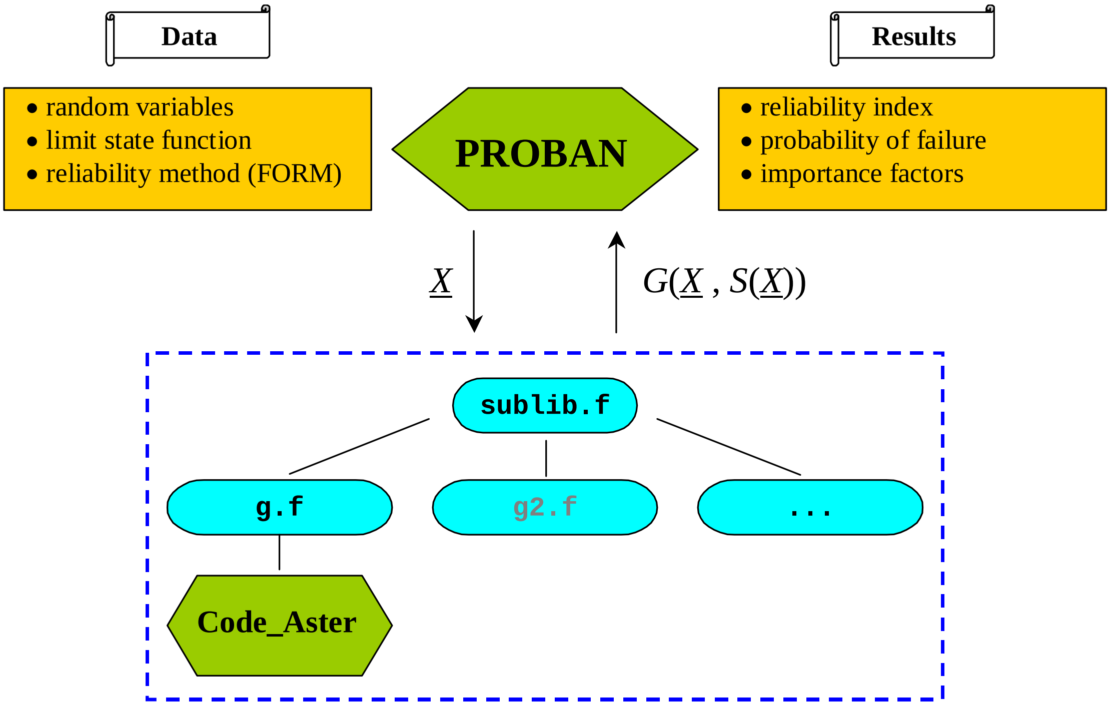


Fig. 4: Principles of the coupling between PROBAN and Code_Aster©

RELIABILITY OF THE COMPOSITE LINER

Problem statement

The role of the composite liner is to ensure air tightness of the concrete wall in regions where cracks may appear during the pressurization of the vessel at $P_0 = 0.38$ MPa. The “tensile stress area” of the intrados is determined from a finite element calculation using the mean values of the parameters (see Table 1). It is represented in figure 5.

The geometry of the composite liner is designed from this calculation so that it encompasses the tensile stress area. For the sake of simplicity, a rectangular geometry of the liner is considered in this study. It is defined by the coordinates (s_0, z_0) according to Figure 6.

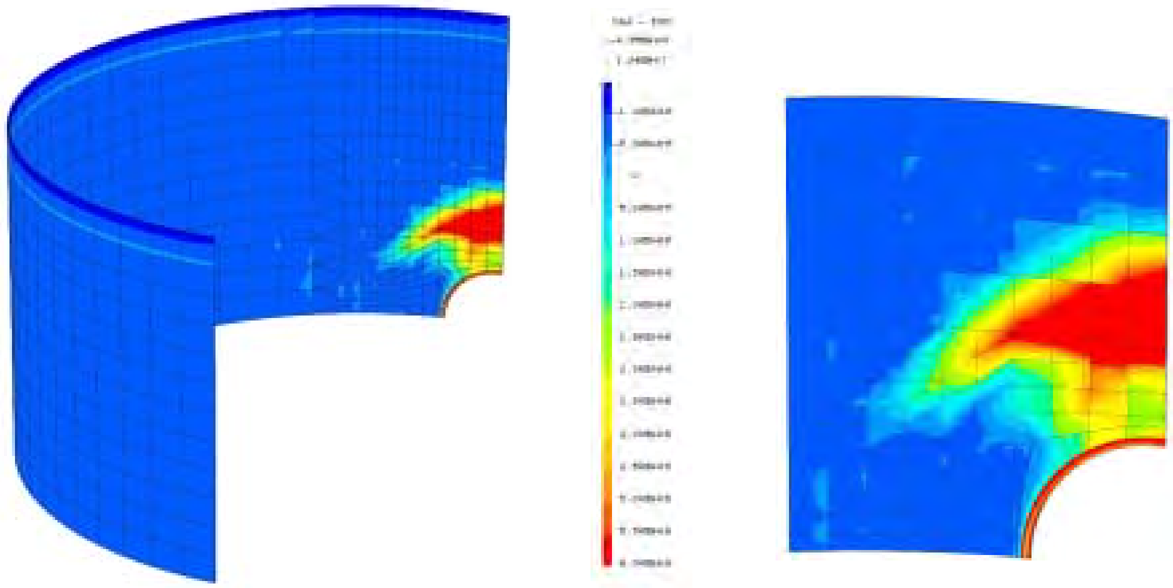
Limit state function and random variables

The failure of the repairing is defined by the fact that tensile stresses appear at the intrados *outside* the region covered by the composite liner. The associated limit state function thus reads:

$$g\left(\underline{X}\right) = -\max_{\overline{D_{rev}}} \sigma_3\left(\underline{X}\right) \tag{8}$$

where σ_3 is the maximal principal stress (computed at the nodes of the mesh) and $\overline{D_{rev}}$ is the region of the intrados that is *not* covered by the liner.

The random variables introduced in the analysis are gathered in Table 2. It is noted that a single random variable χ is used to model the randomness in the tension of all cables : this assumption introduced for the sake of simplicity corresponds to having a perfect correlation between the tension of various cables.



(a) Complete model (b) Zoom over the hatch area

Fig.5: Maximal principal stress σ_3(Pa) at the intrados of the wall

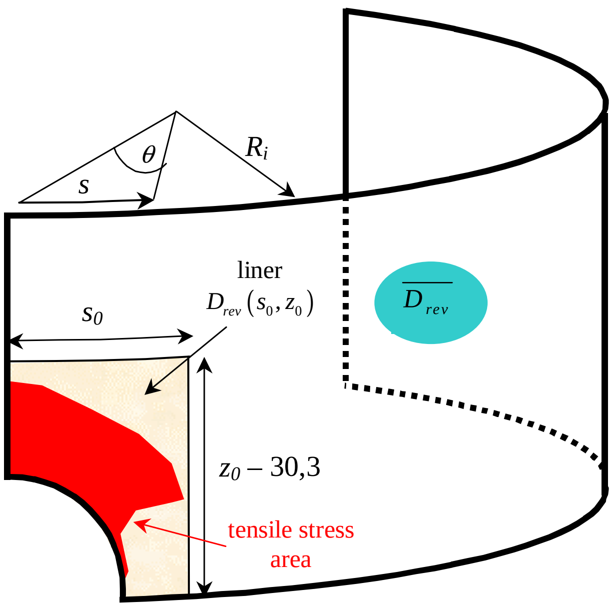


Fig.6: Definition of the liner geometry $D_{rev}(s_0, z_0)$ and its complementary $\overline{D_{rev}}$

Tab. 2: Definition of the random variables

Parameter	Type of law	Mean value	Coeff. of variation
Concrete Young's modulus E_b	lognormal	31,000 MPa	15%
Concrete Poisson's ratio ν_b	uniform	0.22	[0.10 – 0.34]
Concrete mass density ρ_b	lognormal	2,500 kg/m^3	10%
Steel Young's modulus E_a	lognormal	192,000 MPa	5%
Non dimensional tension of the cables χ	lognormal	1	10%

Results

The first analysis was carried out using the following geometry of the liner : $s_0 = 8.7$ m , $z_0 = 41.5$ m . It required 51 calls to the limit state function. The computed reliability index is $\beta = 0.4$ corresponding to a probability of failure $P_f = 0.34$, which means that this design is not conservative. The coordinates of the design point (back-transformed in the physical space) as well as the importance factors are given in Table 3.

Tab.3: Design point and importance factors obtained by FORM

Parameter	Importance factor (%)	Design point
Concrete Young's modulus E_b	0.3	30,558 MPa
Concrete Poisson's ratio ν_b	65.0	0.189
Concrete mass density ρ_b	0.3	2,482 kg/m^3
Steel Young's modulus E_a	0.0	191,829 MPa
Non dimensional tension of the cables χ	34.4	0.972

From this analysis it is observed that only the concrete Poisson's ratio ν_b and the tension of the cables χ have non negligible importance factors. This means that the three other variables can be set as deterministic parameters. This conclusion is validated by carrying out another analysis using solely ν_b and χ as random variables : the same results are obtained. The design point, *i.e.* the most probable failure point in the standard normal space correspond to values of ν_b and χ below their respective means.

It is also observed that the geometrical point of $\overline{D_{rev}}$ where the tensile stress is maximum is located at the same node all along the iterative procedure yielding the design point : it is indeed located on the first vertical line of nodes L_1 immediately to the right of D_{rev} .

A parametric study with respect to the size of D_{rev} has been carried out. First this domain is extended by 10 cm to the left in order to cover the vertical line of nodes L_1 defined above, which corresponds to $s_0 = 8.8$ m , $z_0 = 41.5$ m . The reliability index computed in this second analysis is $\beta = 11.3$ *i.e* $P_f = 10^{-30}$. This means that this extension of the liner is enough to get a very high level of reliability compared to the initial design. The geometrical point of $\overline{D_{rev}}$ where the tensile stress is maximum is now located on the first horizontal line of nodes L_2 located immediately above D_{rev} . Here again this critical point remains unchanged all along the iterative procedure.

In order to check if the design of the liner could be improved, a third analysis is carried out by cropping D_{rev} in the vertical direction, *i.e.* using $s_0 = 8.8$ m , $z_0 = 40.7$ m . The probability of failure obtained in this case is close to 1, meaning that this design would not appropriate at all.

The fact that such huge variations of the probability of failure exist when the geometry of the liner is slightly modified means that the tensile stress area is rather *independent* on the design parameters (which is an interesting conclusion *per se*) : changing the liner geometry by one single row (or column) of elements make the reliability increase from almost zero to almost one. A detailed analysis of the sensitivity of P_f with respect to the size of D_{rev} would require considering non rectangular liner regions, *i.e.* a liner design closer to the real shape of the tensile stress area.

CONCLUSION

A probabilistic framework has been developed to assess the design of a composite liner which is used in order to restore the air tightness of concrete containment vessels of French 1300 MWe power units. The liner is design by computing the "tensile stress area" around the material hatch area under pressurization of the vessel. Then the probability that tensile stresses (and possibly crack openings) appear *outside* this covered region is computed.

The paper presents the use of a finite element reliability approach to solve the problem. This is performed by coupling the probabilistic code PROBAN with EDF's finite element code Code_Aster©. The paper first shows the feasibility of such a coupling for an industrial structure, the numerical model of which contains about 20,000 degrees of freedom.

The analysis allows to select those parameters whose randomness directly influence the probability of failure, namely the concrete Poisson's ratio and the tension of the pre-stressing cables. Finally a parametric study is carried out with respect to the size of the region covered by the liner : only three analysis are necessary to find the best-fit rectangular zone that provides a sufficient (and, in the present example, very high) level of reliability.

This study will be completed in the near future as follows:

- non rectangular geometries of the liner will be considered in order to better fit the shape of the "tensile stress area".
- the aging behaviour of the concrete (creep and shrinkage) will be taken into account in order to evaluate the evolution in time of the probability of failure with respect to a given initial geometry of the liner.
- other areas possibly undergoing tensile stresses under pressure (*e.g.* the gusset) will be studied using the same framework.

NOMENCLATURE

E_a	Young's modulus of the cables
E_b, ν_b, ρ_b	Young's modulus, Poisson's ratio, mass density of the concrete
P_0	Pressure corresponding to LOCA
$\underline{X}$, $f_{\underline{X}}(\underline{X})$	Vector of random design parameters, associated joint probability density function
$g(\underline{X})$	Limit state function
P_f, β	Probability of failure, reliability index
φ, Φ	probability density function, cumulative distribution function of the standard normal law
$D_{rev}, D_{rev}(s_0, z_0)$	Region of the intrados covered by the composite liner
$\overline{D_{rev}}$	Region of the intrados *not* covered by the composite liner

REFERENCES

1. Ghoussoub-El Gharib, J., Badel, P., Jean, C. and Cirée, B., *Méthodologie de modélisation mécanique du comportement des enceintes nucléaires*, 1ère Int. Conf. Albert Caquot « Modélisation et Simulation en Génie Civil – De la Pratique à la Théorie », Paris, 3-5 octobre 2001.
2. *Règles techniques de conception et de calcul des ouvrages et construction en béton précontraint suivant la méthode des états limites : BPEL 91 révisé 99*, Eyrolles, 1999.
3. Ditlevsen, O. and Madsen, H., *Structural reliability methods*, J. Wiley and Sons, Chichester, 1996.
4. Liu, P.L. and Der Kiureghian, A., *Optimization algorithms for structural reliability,* Struct. Safety, **9**, pp. 161-177, 1991.
5. Hasofer, A.M. and Lind, N.C., *Exact and invariant second moment code format,* J. Eng. Mech., ASCE, **100** (1), pp. 111-121, 1974.
6. Det Norske Veritas, *PROBAN user's manual V.4.3,* 2000.
7. Der Kiureghian, A. and Ke, J.B., *The stochastic finite element method in structural reliability,* Prob. Eng. Mech., **3** (2), pp. 83-91, 1988.
8. Flores Macias, O. and Lemaire, M., *Eléments finis stochastiques et fiabilité : application en mécanique de la rupture*, Rev. Fran. Génie Civil, **1** (2), pp. 247-284, 1997.