Transactions of the **14th International Conference on Structural Mechanics in Reactor Technology** (SMiRT 14), Lyon, France, August 17-22, 1997

**G06/1**

# Industrial 3D computation of cracked elbows

**Martelet B., Ignaccolo S., Eymard I.**
*EDF, France*

ABSTRACT

The scope of this paper is to present a new software called ASCOUF and developped by the R&D department of EDF in order to enable the engineer to make a 3D mesh, including the crack zone, in a very short time.

It also explains the main operating process of this tool and underlines the way it helps the engineer to get rid of many usual tasks when carrying out a computation. Afterwards, this document briefly presents the actions performed to validate the tool.

## 1 - INTRODUCTION

Cast duplex stainless steel elbows of French Pressurized Water Reactors are considered as sensitive components as far as the disponibility of the production resource is concerned. Indeed, those structures of the primary loop may contain flaws which can grow during the operating time of the plant, if we assume they behave like a crack. Moreover, as the cast duplex stainless steel tends to lose ductility when submitted to thermal ageing [1], the mechanical properties of the material (particularly the toughness) decrease. Then, the technical (and financial) aim of the studies and experiments carried out by EDF and FRAMATOME is to assess and predict the behaviour of cracked elbows until the end of life of the plant.

In the last years the needs in terms of fast analysis in the field of flaw assessment of steel components of nuclear industry have become more and more important. And thanks to the enlarging capabilities of the computer codes, it is now possible to complete very big 3D elasto-plastic finite element calculations of cracked bodies to evaluate the energy release rate J. However, previous studies [2] [3] which dealt with elbows have clearly shown that meshing takes a very long time (from 4 to 8 weeks, including validation). That is why a meshing tool, ASCOUF, has been created by the R&D department of EDF [4] to generate automatically an accurate mesh of the structure containing a semi-elliptical crack..

## 2 - OPERATING PROCESS

As a geometrical entity, an elbow can be accurately defined by 4 parameters (see figure 1) :
$\alpha$ : elbow angle
$R_c$ : bending radius
$R_m$ : mean radius
e : thickness

The semi-elliptical crack located on the structure is determined by other variables :
a : depth
2c : length
$\alpha_c$ : angular position along the elbow
$\phi_f$ : azimutal position ($\phi_f = 0°$ at the extrados ; $\phi_f = 180°$ at the intrados)
$\xi_f$ : crack orientation ($\xi_f = 0°$ for a longitudinal crack, $\xi_f = 90°$ for a circumferential one)
position : inner or outer surface crack

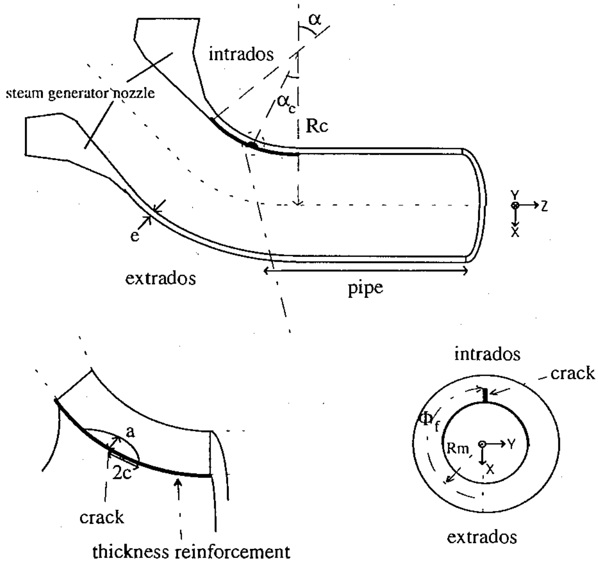

*figure 1 : Main geometrical parameters of a cracked elbow*

Moreover, the software allows the user to model reinforcement at the inner surface of the elbow. Thanks to ASCOUF, one can also represent the outlet or inlet nozzles of the components of the primary loop connected to the elbows : figure 2 shows how the elbows are linked in the loop (pipes, steam generator, reactor coolant pump, pressure vessel).

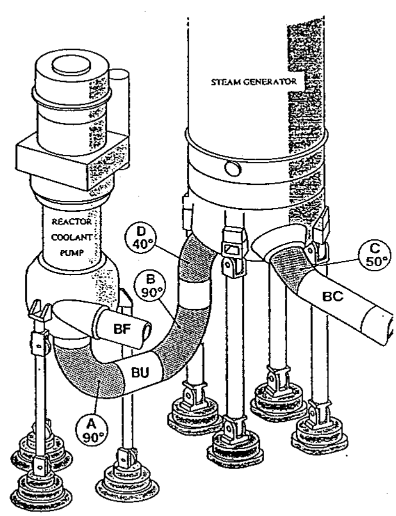

*figure 2 : Primary loop of a PWR*

To mesh the entire structure we work first on a cracked plate. At the first step, a crack block is inserted into an octogonal one in order to obtain 3 crack orientations by rotating the octogone ($\xi_f$ = 0°, 45° or 90°). The other orientations between 0° and 90° can be generated by a local distorsion of the mesh. Around the octogone the mesh is adjusted and includes 2 boundaries : the limit between the component nozzle and the elbow and the junction between the straight pipe and the elbow. At this stage, it only remains to roll the plate around the middle fiber and then to curve the tube previously obtained according the bending radius of the elbow $R_C$. Let us precise that this bending concerns exclusively the part of the tube transformed into the cracked elbow : during the bending, the nozzle and the pipe are only rotated through a rigid body displacement.
Those steps are represented on figure 3.

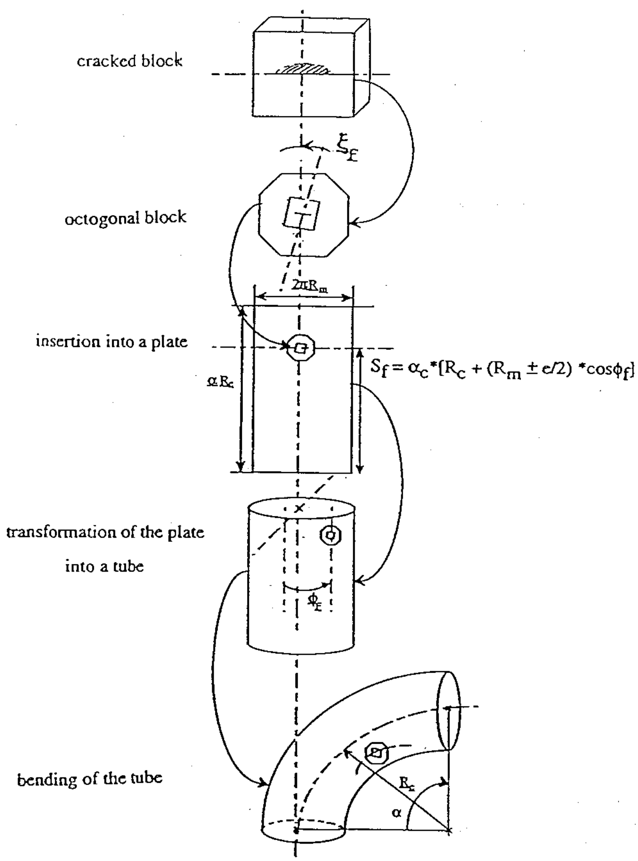


*figure 3 : Operating process of ASCOUF meshing*

Of course, such a sequence implies that the geometrical transformation remains under control. In fact, the mathematical conversion applied to the nodal coordinates of the plate is analytically determined, to ensure that the crack size and localization are consistent with those expected. Moreover, that procedure provides the user with the possibility to stop the transformation at any step in order to get either a cracked plate or a cracked tube.
Let us add finally that it is also possible to get a non cracked mesh with ASCOUF if desired.

## 3 - CONSEQUENCES FOR THE USER

Thanks to the development of the computer (CRAY C90) environement AnTheMiX, ASCOUF turns to be of a very easy use. Indeed, meshing a complete structure with or without a crack is summed up in one operation : fill in the data file ASCOUF with the parameters given above (see figure 4). This file also requires the material characteristics, the stress-strain law and the loadings. Now, it is possible to specify combined loadings such as pressure, in plane and out of plane bending moment, twisting moment and thermal shock.
In fact, node and element groups usefull for the boundary conditions and the post processing are automatically generated by ASCOUF. So, with just one data file the software creates the control file which can be used by the finite element software *Code_Aster®*. Many post-processing facilities are also set up in this file (J by the "Theta" method [5], crack opening displacement, ...) in order to reduce the manual action during the study including meshing, computation and assessment steps.
ASCOUF generates other files which can help the user to visualize the mesh or convert it into an I-DEAS or a CASTEM format. As far as tracebility is concerned, the software provides us with a specific file (the Information file) which gives the different steps of the meshing operation (figure 4).

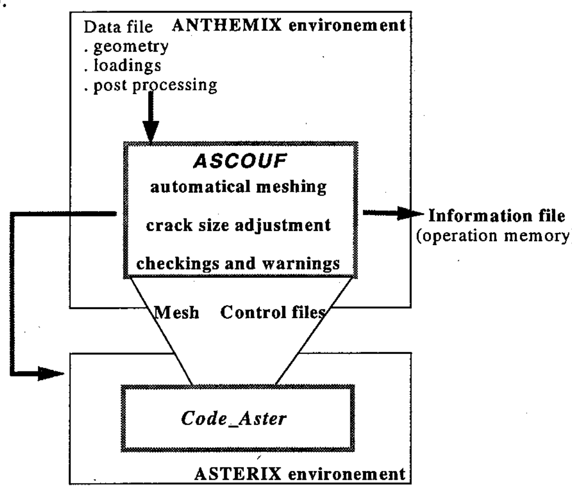


*figure 4 : ASCOUF computer environement*

## 4 - VALIDATION

Several studies were made to validate the meshes coming from the software when they are submitted to complex loadings like pressure, bending moments and thermal shock, even in a wide range of plasticity extension.
As a matter of fact, except near the cracked block, there are 3 elements in the thickness of the elbow and one in the straight pipe (those data were set in the specifications in order to limit the number of nodes).
The aim of the validation is then to ensure that ASCOUF meshes are satisfactory in terms of number of nodes and refinement. So, we have carried out 2 studies to compare a few results between meshes coming from ASCOUF and from an other software (I-DEAS Master Series). Thus, we have shown that for both mechanical and thermal loadings, ASCOUF meshes are able to give good results as far as displacements, stresses and energy release rate are concerned (see figure 5 and figure 6).

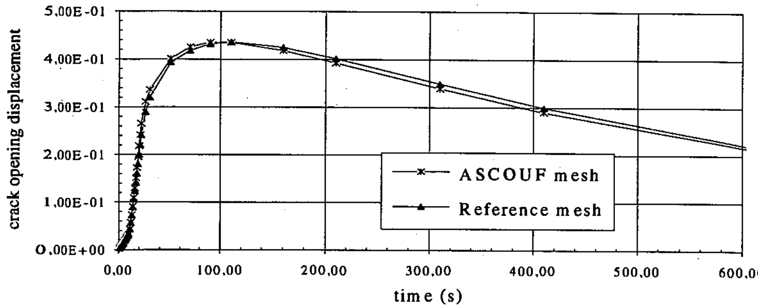


*figure 5 : Comparison of crack opening displacement during a thermal shock*

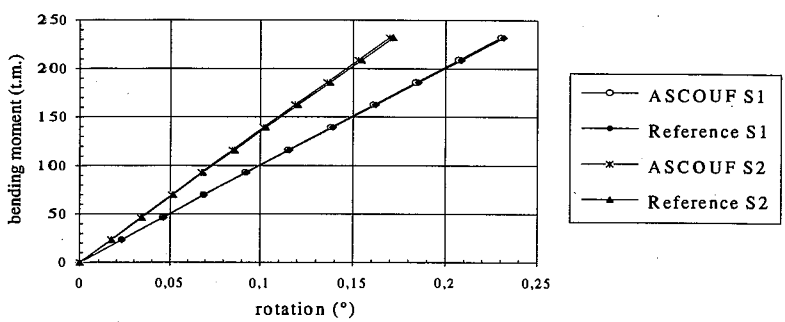


*figure 6 : Comparison of section rotations under a bending moment*

Even for thermal shock, in spite of an obvious lack of accuracy : indeed, only one element in the pipe part is not enough to reproduce important thermal gradient through the thickness of the structure. However, as the mesh around the crack is very refined one does not make any mistake using such a mesh. In fact we can show that during the transient phase of a thermal shock ASCOUF provides higher values of J (energy release rate) than the reference mesh. And when the maximum value of J is reached, ASCOUF overestimates the reference value only by 5% (figure 7).

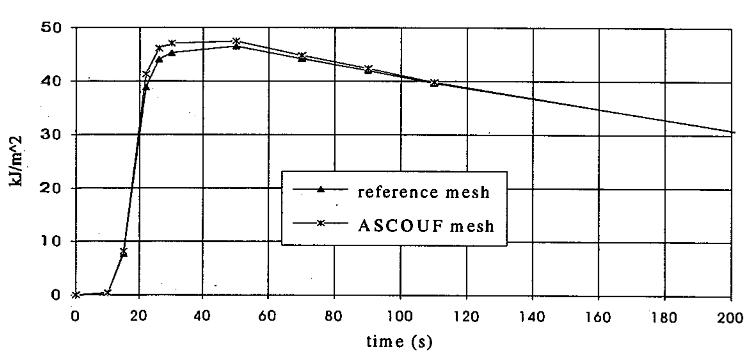


*figure 7 : Comparison of energy release rate during a thermal shock*

5 - PERFORMANCES

At the moment, the execution performance time on a CRAY computer depends on the mesh refinement required by the user. For instance, a 27 000 nodes structure (figure 8) including an internal reinforcement requires about 1400 CPU seconds.

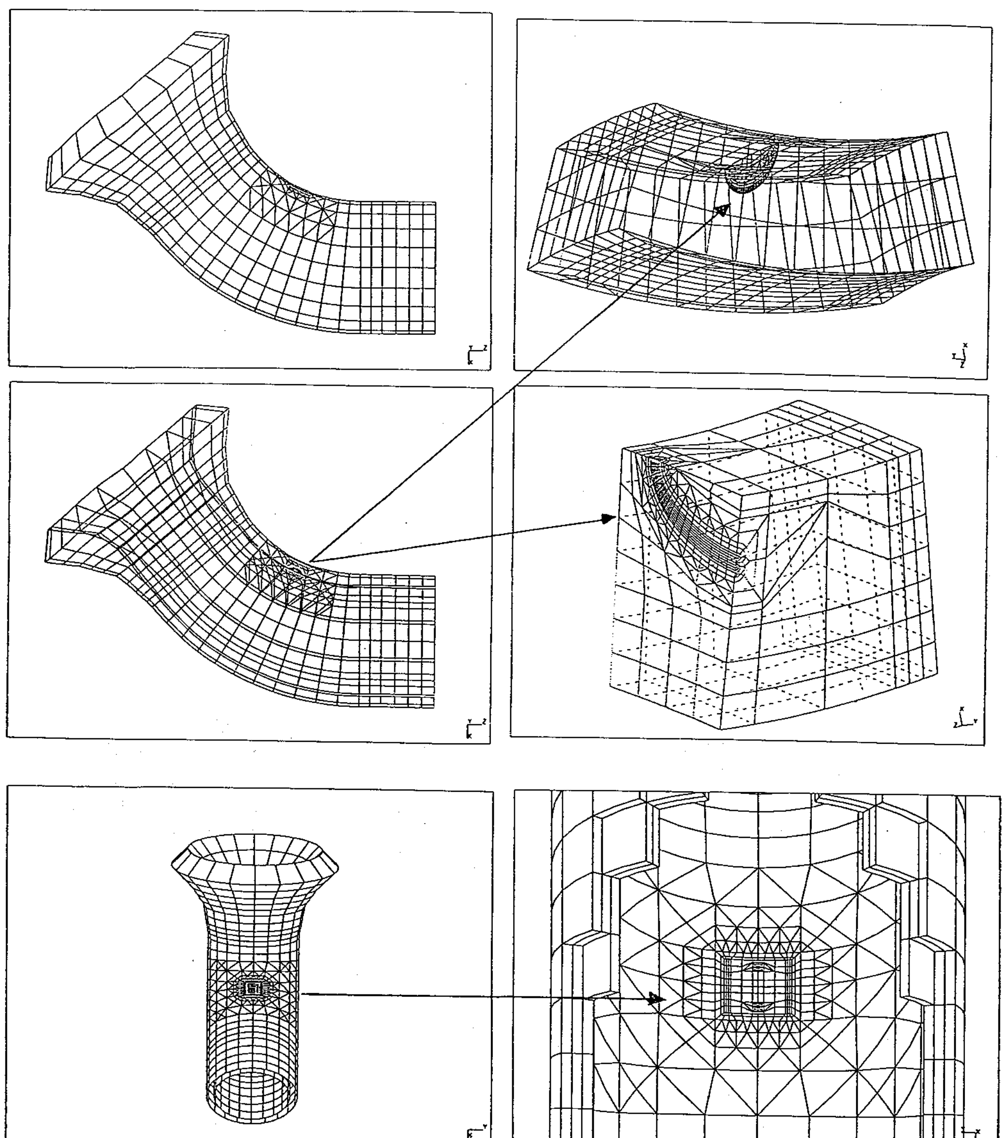

*figure 8 : Vue of a 27 000 nodes ASCOUF mesh*

A complete study carried out in 1994 on a cracked elbow needed about 5 months [3]. The time spent on the mesh was about 2 months, including its validation. In 1996, we have performed with ASCOUF an important industrial computational program based on 10 structural integrity analysis : the average engineering time for each of them has dropped down to 3 weeks.

## 6 - CONCLUSION

The development of the software ASCOUF is an answer to one challenge : reduce the time previously needed for the meshing of a 3D cracked plate, pipe or elbow. Thanks to this tool, we can now carry out a lot of non-linear finite element analysis in order to understand better the behaviour of the cracked structures.
Thus, ASCOUF will surely help us to improve the definition of the main parameters of the simplified engineering methods based on the parameter K or J (R6 procedure [6], $J_S$ [7] and $K_J$ [8] [9] approaches). As a matter of fact, the main set of finite element computations conducted in France by CEA, EDF and FRAMATOME [10] is relying on 2D computations of pipes. So this tool will allow us to treat the case of pipes or elbows containing semi-elliptical cracks.
Moreover, the skills gained on the course of the development of this software have permitted to build up a new version which contains a package on local loss of thickness. In the future, the application fields of ASCOUF could also include throughwall cracks for the assessment of Leak Before Break [11].

## REFERENCES

1. Massoud, J.P., Jayet-Gendrot, S., Le Delliou, P., Semete, P., Churier-Bossennec, H., Ignaccolo, S 1997. *Thermal ageing of cast duplex stainless steel primary components. Overview of the research program conducted by EDF.*
Proceeding of ICONE 5, May 26-30 1997, Nice, France 1997.

2. Ignaccolo, S., Churier-Bossennec, H., Faidy, C., Martelet, B 1994. *3D computations of cracked elbows.*
Saclay International Seminar on Structures Integrity, Saclay, France 1994.

3. Ignaccolo, S., Churier-Bossennec, H., Faidy, C., Martelet, B. 1995. *3D computations of cracked elbows.*
SMIRT 13, transaction Vol. II (August 95), Porto Alegre, Brasil.

4. *Une automatisation de l'utilisation du Code_Aster pour l'analyse mécanique des coudes moulés fissurés.*
EDF Etudes et Recherches. Faits marquants 1995, pp. 84-86.

5. Destuynder, P., Djaoua, M. 1981. *Sur une interprétation mathématique de l'intégrale de Rice en théorie de la rupture fragile.*
Mathematical methods in applied sciences, 3, pp. 70-87, 1981.

6. Milne, I., Ainsworth, R.A., Dowling, A.R., Stewart, A.T. 1986. *Assessment of the integrity of structures containing defects.*
R/H/R6 - Rev 3, Central Electric Generating Board.

7. Moulin, D., Drubay, B., Clémant, G., Nedelec, M. 1995. *Evaluation of ductile tearing in a cracked component with a simple method ($J_S$).*
SMIRT 13, transaction Vol. II (August 95), Porto Alegre, Brasil.

8. Gilles, P. 1996. *Jestimation scheme for surface cracked pipings under complex loadings : Part I : Theoretical basis.*
ECF 11, Mechanisms and mechanics of damage and failure, Poitiers, France 1996.

9. Gilles, P. 1996. *Jestimation scheme for surface cracked pipings under complex loadings : Part II : Complex shaped elbow solutions..*
ECF 11, Mechanisms and mechanics of damage and failure, Poitiers, France 1996.

10. Ignaccolo, S., Drubay, B., Michel, B., Gilles, P. 1996. *Behaviour of cracked cylinders under combined thermal and mechanical loadings.*
ECF 11, Mechanisms and mechanics of damage and failure, Poitiers, France 1996.

11. Gerard, R., Taupin, P. 1995. *Leak before break analysis of the primary loop pipings of Belgian nuclear power plants.*
SMIRT 13, transaction Vol. II (August 95), Porto Alegre, Brasil.