FINITE ELEMENT ANALYSIS OF CROWNING SEALING CAPS

Henri Champliaud,

L Van Ngan

École De Technologie Supérieure, Montréal, Canada

ABSTRACT

Metal closures, with an integrated gasket, are widely used in the food industry to ensure the sealing of the glass bottles by setting. The success of this bottle-capping is due to the relevant choice of the variables in the manufacturing process. Actually, the effects of the various variables of the process are not precisely understood.

In this paper, the leakage pressure of an assembly, with given parameters of a standard setting operation is predicted. The study is concentrated firstly on the simulation of setting the cap on the bottle, and secondly, on the global distribution of the efforts of contact on the gasket according to the internal pressure. Finally, the leakage pressure of the assembly is determined using practical tests that relate the leakage pressure with the global force exerted on the gasket.

The simulation of the setting operation by the finite element method using Ansys® (5.4 ver) follows four major steps: plunger descent, crowner descent, release of the plunger and release of the crowner. A good agreement exists between the predicted final geometry of the cap and cross sections of real assemblies.

The simulation of the application of pressure inside the assembly gives the rule of contact force decrease with the increased pressure. The result of this analysis is the predicted leakage pressure given by this result has a good agreement with real leakage pressure obtained by tests.

INTRODUCTION

The sealing of glass bottles in the breweries is ensured by a metal cap crushing a polymer gasket on the top of the neck. The role of the cap is to prevent from gas leakage until the opening by the consumer. So the maximum retained pressure of a given assembly measures its performance. How the final assembly is related to the parameters involved in the process is not precisely known. In fact the major part of a crowning process is based on experimentation, and any change will give hardly to predict final results.

The initial complex shape of a metal cap is obtained by a setting operation. The tools used for that step are very expensive, and it is not reasonable to search for improvement through trial and error. One motivation of this study is to replace the trial and error experimentation by a numeric simulation of the crowning process with Ansys®. The second motivation is to predict the leakage pressure of the assembly by coupling experimental tests on the gasket with the contact force decrease on the gasket of an under increasing pressure assembly.

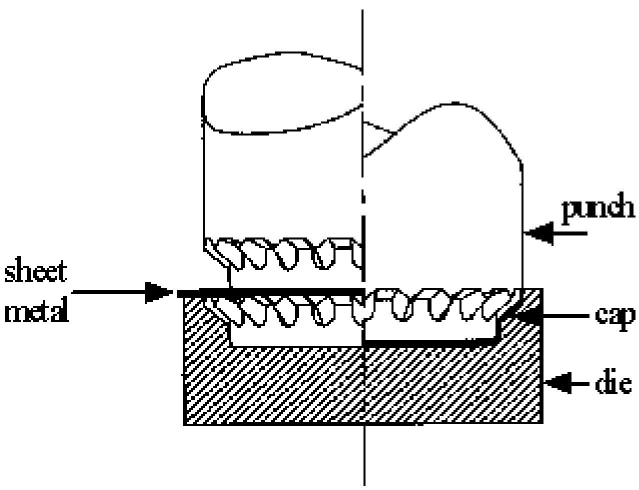
The 3D simulation of the crowning process involves elastic-plastic bricks elements for the sheet metal, hyper-elastic bricks elements for the gasket and contact plane elements for modeling the free of friction interaction between the tools and the different parts of the cap. Apart two experiments are realized. First, a molded cylinder of the gasket material is compressed to determined the Mooney-Rivlin constants and the Poisson ratio. These constants are needed to achieve adequately the numeric simulation. Second, leakage tests are conducted to determine the behavior of a gasket compressed on a bottle neck. This last experiment is needed to determined the characteristic of the gasket.

CAP AND BOTTLE ASSEMBLY

There is two majors steps in a cap life. The first step is the setting operation which gives the shape of the crowning cap to the sheet metal. Figure 1 shows the punch and die tools used to form the cap from the sheet metal. The sheet metal is a thin disk of about 40 mm diameter and 0.25 mm thick. After this operation the gasket is molded with a punch directly inside the cap. This process is the last one before the caps are shipped to the breweries.

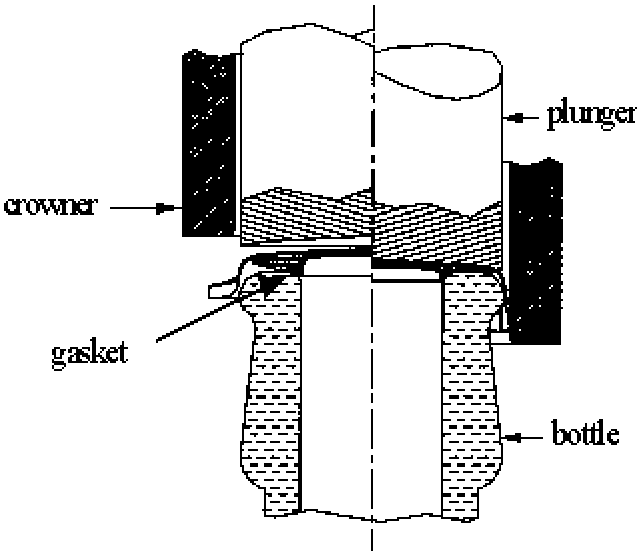
Then on the capping line in the factory the cap is placed on the top of a bottle. Actually there is two kind of caps: the "pry-off" cap, and the "twist off" cap. Usually, only the shape of the gasket is different from one kind to another. The "classic" cap is fitted by crowning on a bottle which has an axisymmetric section for its neck. The "twist off" cap is also fitted by crowning on the threads of the bottle. In that case the bottle has not an axisymmetric section. In the present work, we limit our analyses to the "pry-off" cap.

Figure 1. Setting the cap



The plunger is used to load the cap on its seat. By descending the crowner the final shape of the cap is directly done on the bottle opening. Finally the tools are extract and the assembly is achieve. Inside the bottle the gas escaping from the liquid begin to build a pressure. The assembly must be able to retain this pressure to keep the taste of the beverage. We can see in the figure 2 the crowning operation when the plunger is still loading the cap while the crowner is at its lower position (right side). The crowner is first retracted and only after that the plunger is withdrawn.

Figure 2. Crowning the cap



As we can see on the preceding figure, the crowning operation involves structural nonlinearities: 1) large displacement of the waves around the cap, 2) elasto-plasticity of the sheet metal, 3) hyperelasticity of the gasket and 4) contact between the gasket and the bottle, and between the cap and the tools.

METHODOLOGY USED TO PREDICT THE LEAKAGE PRESSURE OF AN ASSEMBLY

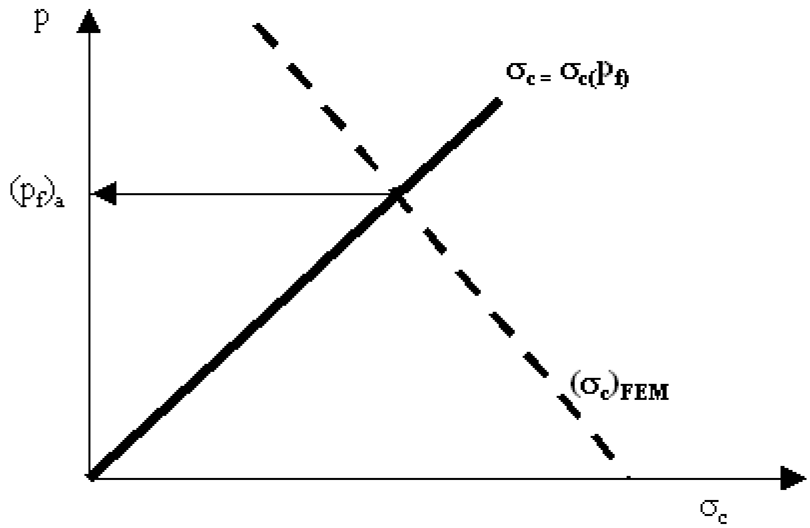
According to the ASME Code, the contact stress σ_c on a flat gasket is directly related to the leakage pressure p_f through a constant characteristic usually called gasket factor "m".

$$m = \frac{\sigma_c}{p_f}$$

(1)

In our case, the gasket contact surface is not flat. However we can determine by experiment a similar relationship for each particular gasket-bottle set. As an example, this relationship shown by the solid line in figure 3 represents the minimal contact stress needed to prevent leakage for a given pressure. This line is the sealing property of the gasket.

Figure 3. Leakage pressure of a given assembly



The dotted line in figure 3 represents the contact stress at gasket of the actual assembly with varying internal pressure, and can be obtained by FEM (Finite Element Method).

The principle of the present study is to determine experimentally the sealing property of the gasket-bottle set, like the solid line in figure 3; to study the sealing behavior of the assembly by FEM, like the dotted line; and to predict leakage pressure by intersecting these two lines.

In our case, the contact stress on the gasket is not constant along a radial path, and also along a tangential path. So it is very difficult to relate the contact stress to the leakage pressure. An alternative used for this study, is to relate the leakage pressure directly to the global force exerted on the gasket by the bottle. Good agreement has been obtained, as shown in section 6.

MATERIALS PROPERTIES

The sheet metal is a ductile material, with standard characteristics: Young modulus of 200 Gpa, yield at 300 MPa, and Poisson ratio of 0.3.

As we said earlier, the gasket is hyperelastic, so that its behavior is described by a strain energy density W. In addition the polymer material of the gasket is a rubber-like material, then nearly incompressible. In that case the Mooney-Rivlin model is appropriate to handle the behavior of that kind of material. Equation 2 show a simple 2 constants model in terms of the strain invariants J_1, J_2. Note that the third invariant J3 is equal to 1 for an incompressible material.

$$W = C_1(J_1 - 3) + C_2(J_2 - 3) \quad (2)$$

with

$$J_1 = \lambda_1^2 + \lambda_2^2 + \lambda_3^2$$

$$J_2 = \lambda_1^2\lambda_2^2 + \lambda_2^2\lambda_3^2 + \lambda_3^2\lambda_1^2 \quad (3)$$

$$J_3 = \lambda_1^2\lambda_2^2\lambda_3^2$$

where the λ_i is the principal stretch in the i direction, related to the engineering strain by:

$\lambda_i = 1 + \varepsilon_i$ (4)

Since in a compression test only along direction 1 we have

$S_{11} = \frac{1}{\lambda_1}\sigma_{11}$ (5)

where S_{11} is the second Piola-Kirchhoff stress along direction 1, and σ_{11} is the engineering stress also along direction 1. With the use of the right Cauchy-Green strain tensor we can show that:

$S_{11} = \frac{2}{\lambda_1}\frac{\partial W}{\partial \lambda_1}$ (6)

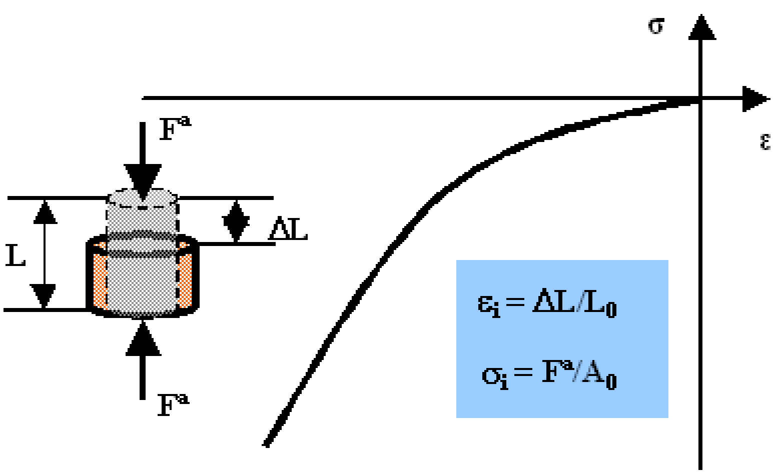
we finally can build a linear relation for the C_i:

$\frac{\sigma}{2\left(\lambda_1 - \frac{1}{\lambda_1^2}\right)} = C_1 + \frac{C_2}{\lambda_1}$ (7)

In order to determine the constants of the Mooney-Rivlin model, we mold a cylinder of the material of the gasket according to the same conditions of temperature and pressure required during this operation for the cap. Since the gasket work only in compression we test the specimen in compression. The experimental data of the test are recorded in a way to obtain the engineering strain versus the engineering stress (see figure 4 for formulas used). The initial dimensions are L_0 for the length of the specimen, and A_0 for the cross section.

By setting y_j equal first member of equation 7 calculated for each j recorded data and x_j equal $1/\lambda_1$ also calculated for each recorded data we can build a linear regression model that gives the two Mooney-Rivlin constants C_1 and C_2.

Figure 4. Compression test

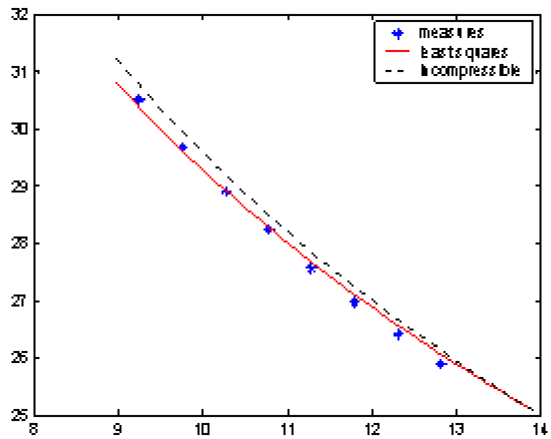


In addition, to complete the characterization of the material we need determine the Poisson ratio. By recording the diameter of the specimen during the compression test and using the logarithmic strain measure we can show that the actual diameter D is related to the actual length L by:

$$D = D_0 \left(\frac{L_0}{L} \right)^{\nu} \quad (8)$$

where D_0 is the initial diameter of the specimen and ν the Poisson ratio. To find ν we use a minimization of the least square which gives a non-linear expression function of the unknown ν. Finally we utilize a Newton method starting with a value for ν of 0.5. Figure 5 show the result of the method based on the least squares.

Figure 5. Poisson ratio calculated with the least squares method

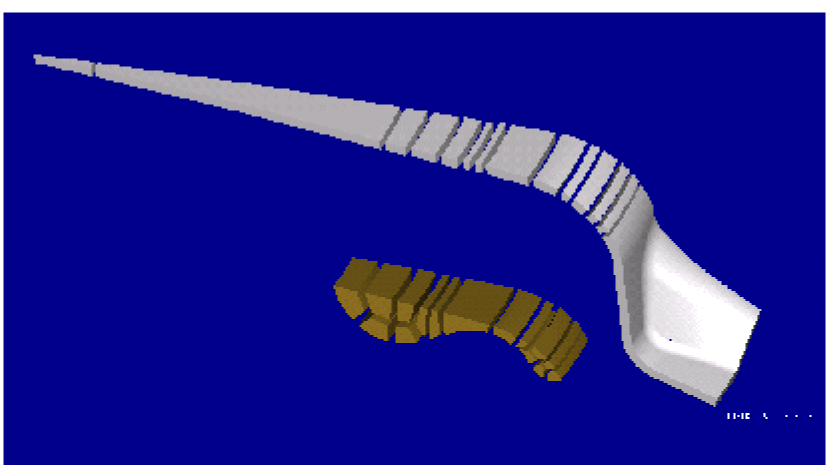


FINITE ELEMENT MODEL

As we said earlier the geometry of the waves around the cap is repeated 21 times, and every wave has a radial plane of symmetry. Since the bottle is axisymmetric and that we can reasonably suppose that the forces exerted by the crowner are equally distributed over the 21 waves, we can model only half of the repeated geometry. This give a great simplification for the model.

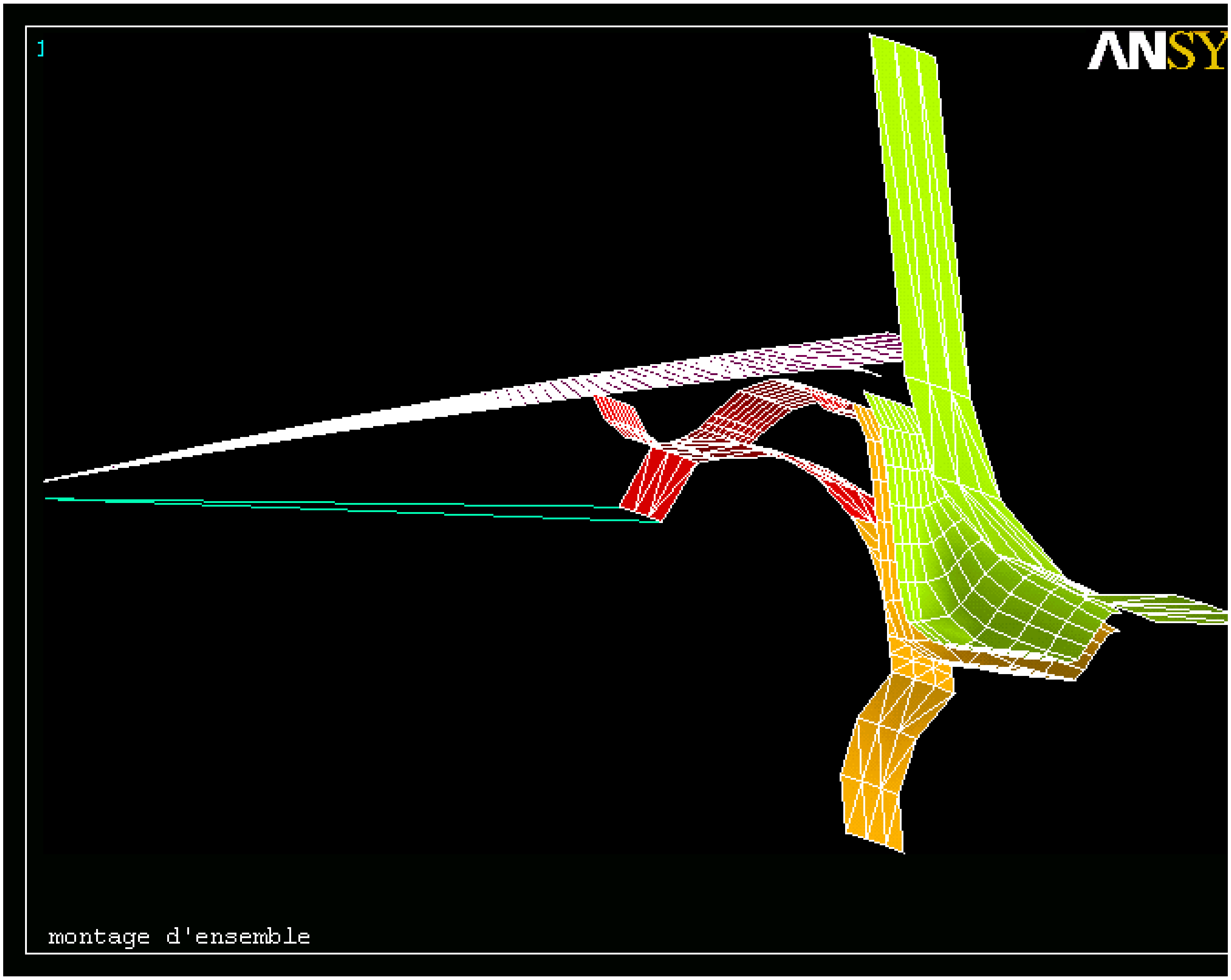
Even if we have only 1/42 of the cap to model, the shape of the wave is still complex. Since we are able to import IGES files in the Ansys® environment we draw a solid model of the cap using Proengineer®. To keep the compatibility between elements at the boundary of different material, the global volume is cut into several parallelepiped as shown in figure 6.

Figure 6. Volumes of gasket and sheet metal for a cap sector



The gasket is meshed into Hyper58 elements (8 node mixed u-p hyperelastic solid) and the sheet metal into Solid45 (8 node brick type) and Solid92 (10 node tetrahedral) for the sharp edge. Figure 7 shows a regular mesh for the sector cap.

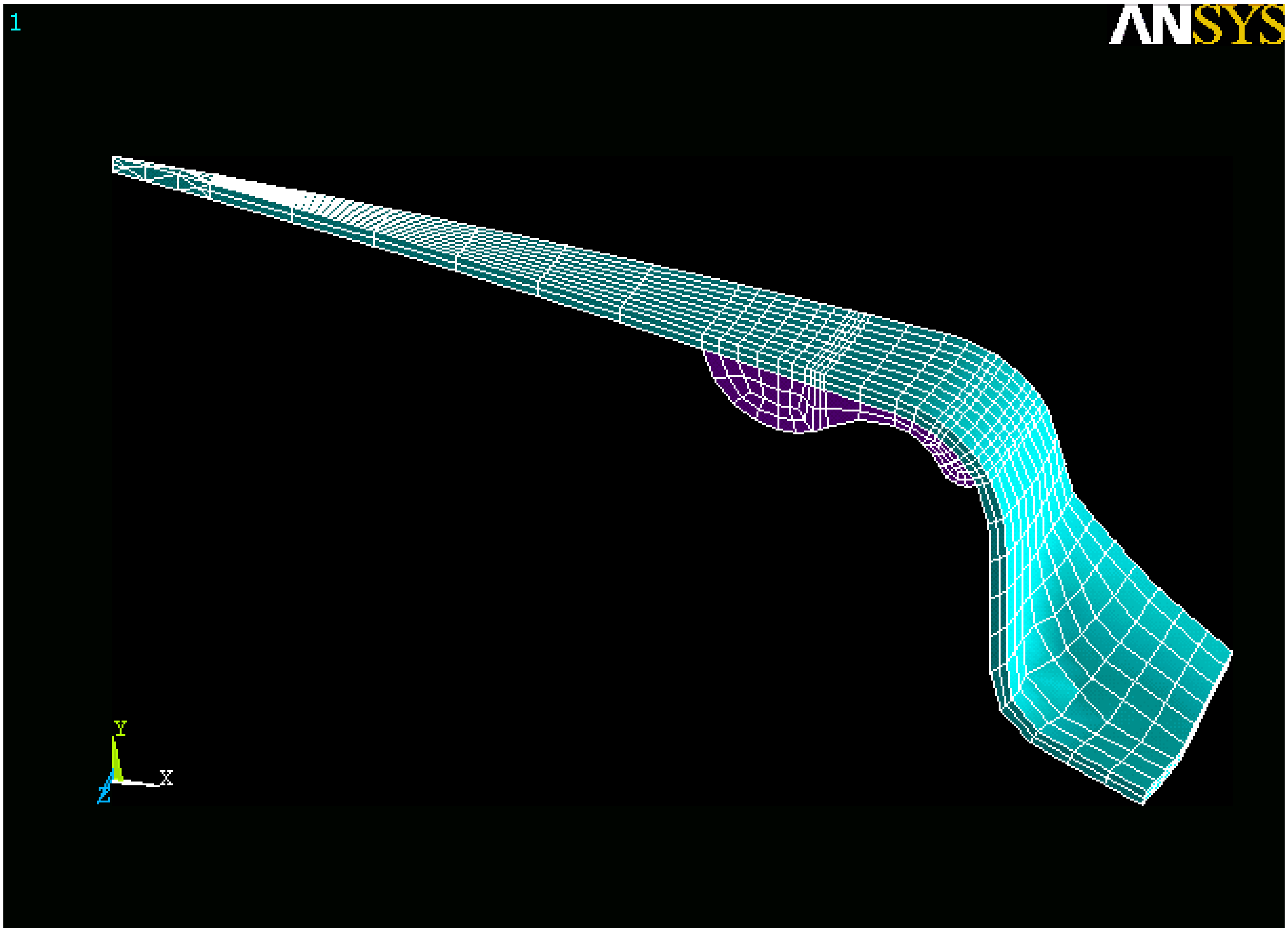
Figure 7. Regular mesh of the cap sector



SIMULATION WITH ANSYS

Now we have to add the bottle and the tools (plunger and crowner) to the model of the cap. In our study we considered that the tools and the glass bottle are rigid in comparison with the cap and of course the gasket. So we use TARGE170 elements for the rigid surfaces of the tools and the bottle. For the flexible surfaces we use CONTA173 elements to put a skin on the gasket and the metal cap. Every rigid surface share a real constant with a flexible surface. Figure 8 shows the mesh of the contact surfaces. Triangular elements are rigid and rectangular element are flexible. A same color for triangular and rectangular elements in figure 8 identifies a contact pair. The proper sizes of the contact areas are the result of a trial and error process. The complete meshed model is the superposition of figure 7 and figure 8. The green link elements in figure 8 are soft springs, used to prevent the cap from rigid displacement.

Figure 8. Mesh of the rigid and flexible surfaces with TARGE170 and CONTA173



In the simulation we have 3 types of contact. One type is between the top of the cap and the plunger. This contact involves little bending of the cap, so that a contact stiffness factor (FKN coefficient) of 0.2 gives good results in terms of penetration and CPU time. Another type of contact is between the gasket and the glass bottle. The material of the gasket is soft in comparison with the steel of the cap and after several adjustments a FKN value of 15 affords convergence satisfaction. The third type of contact is between the crowner and the cap, and between the bottle and the cap. This contact involves more bending of the cap, so that a FKN value of 0.4 gives good results. Since the crowning operation is nearly frictionless we neglect friction in the simulation.

The crowning simulation follows 4 steps: plunger descent, crowner descent, release of the crowner, and release of the plunger. When the crowning operation is achieved, a 5th step consists of applying a pressure on the inner face of the cap. The movement of the tools are simply imposed by the D command in a *DO loop on the selected set of nodes of the appropriate tool. In order to keep robustness for convergence, the displacements are imposed by small increments.

Figures 9 and 10 show the plunger descent. The global imposed displacement is 0.6 mm in 12 steps in order to sufficiently load the cap as in the real crowning operation.

The descent of the crowner is shown in figures 11 to 13. The global imposed displacement on the crowner is 7.5 mm in 75 equal steps. The critical point in term of convergence and CPU time in these steps is when the material begins to yield.

Figure 9. Plunger descent, first step

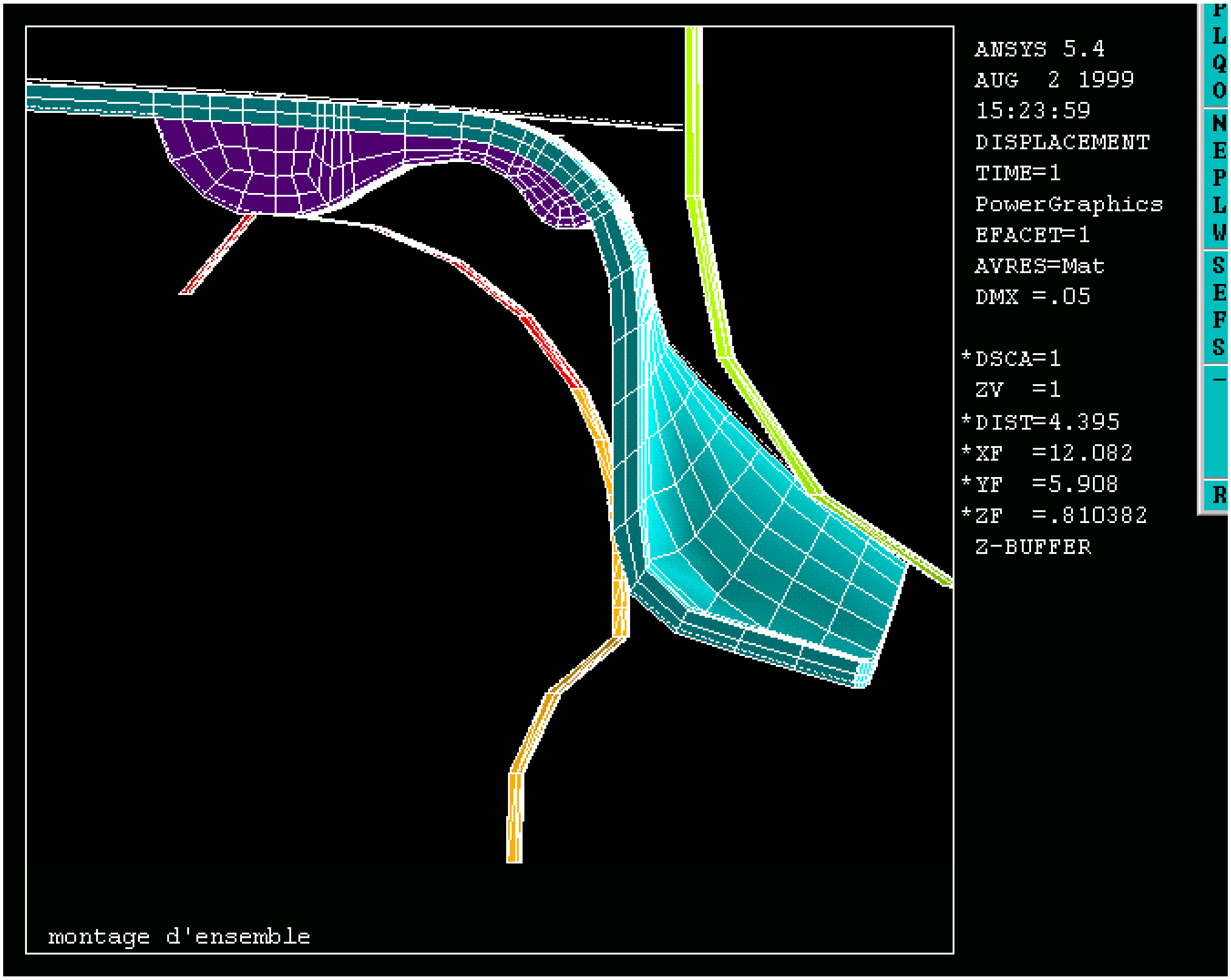


Figure 10. Plunger descent, 0.60 mm

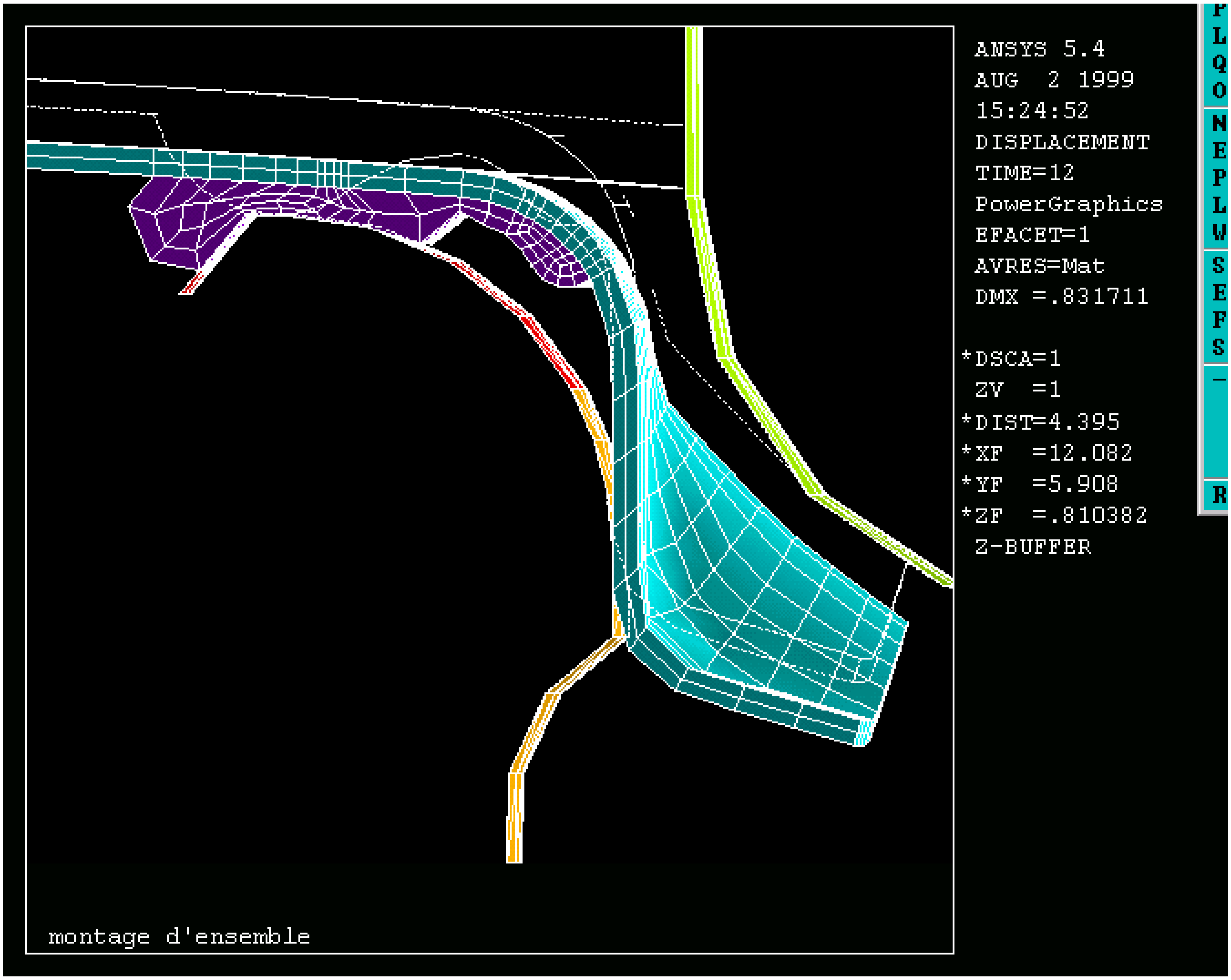


Figure 11. Crowner descent, first step

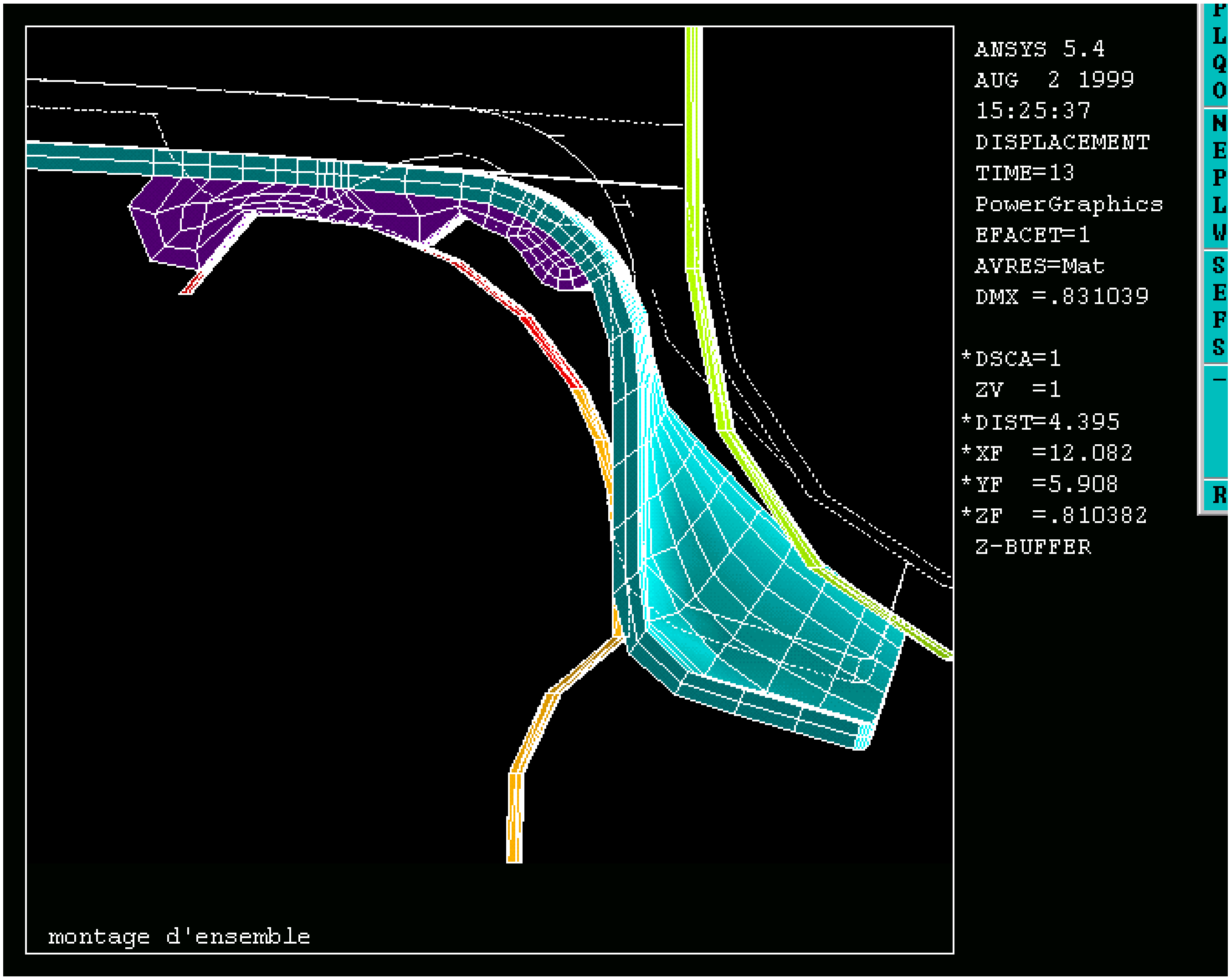


Figure 12. Crowner descent, 2.5 mm

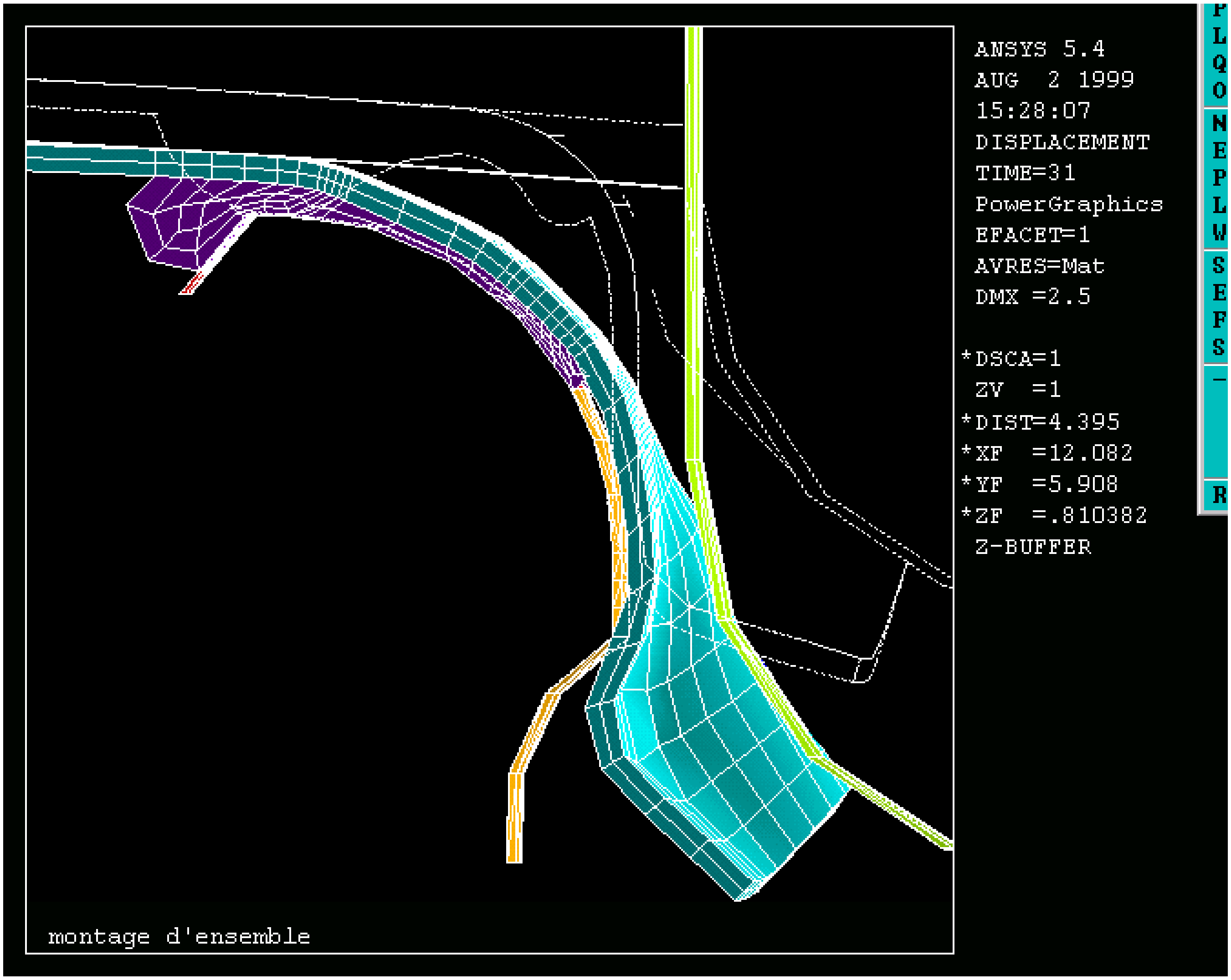


Figure 13. Crowner descent, last step

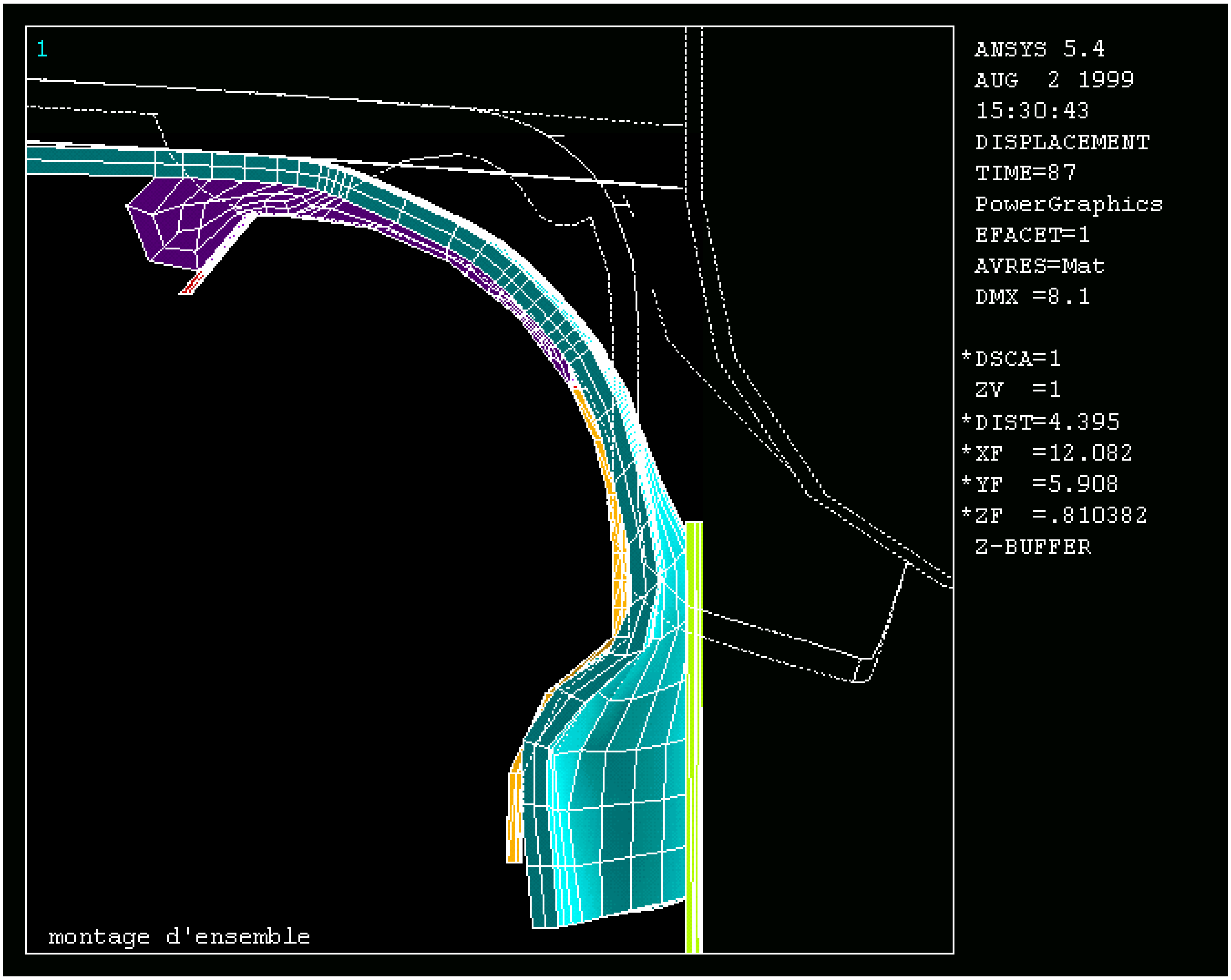
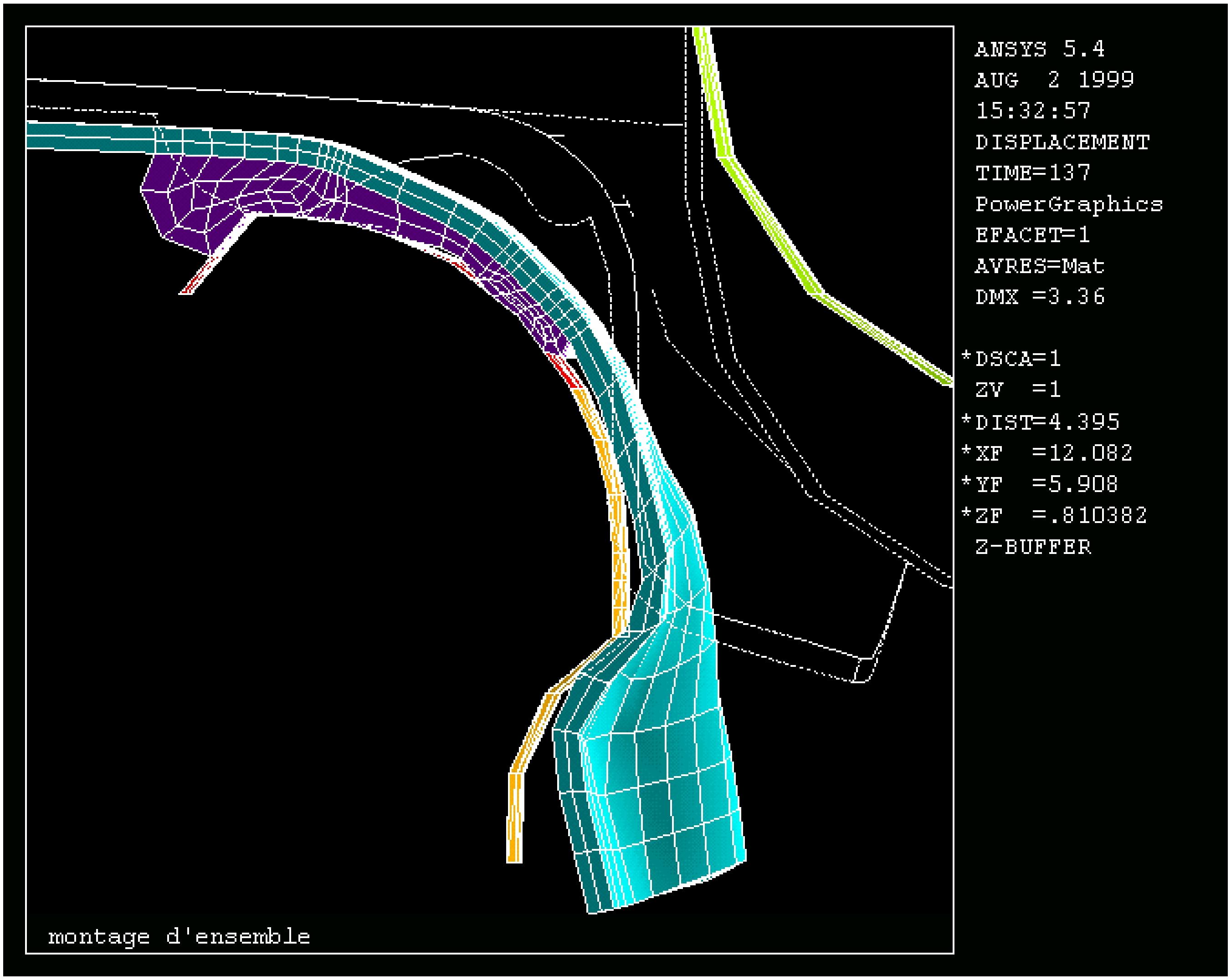


Figure 14. Crowning is complete



The release of the tools follows the order of the real crowning operation: first the crowner retracts in 40 steps of 0.25 mm each, and second the plunger in 9 steps of 0.1 mm each and a supplementary final big step of 1.1 mm in order to give sufficiently space for the deformed cap under pressure. Figure 14 show the cap when the crowning operation is complete.

We can now compare the geometry obtain by the simulation with true geometry. Figure 15 and 16 show good agreement between geometries.

Figure 15. Geometry comparison, between wave

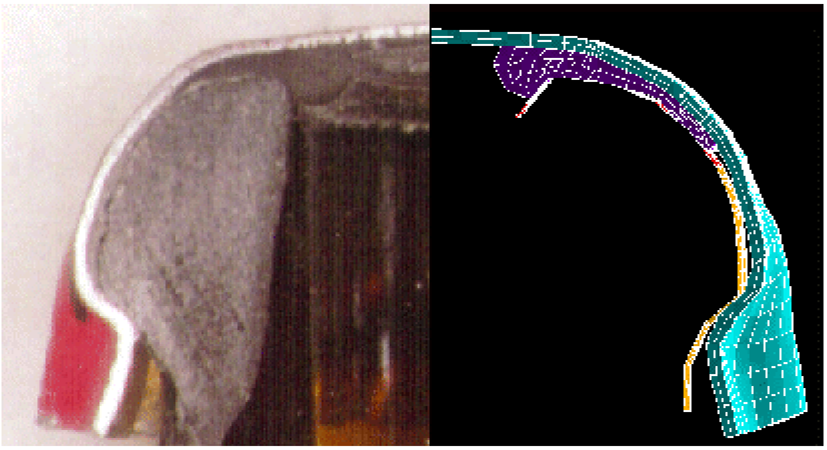
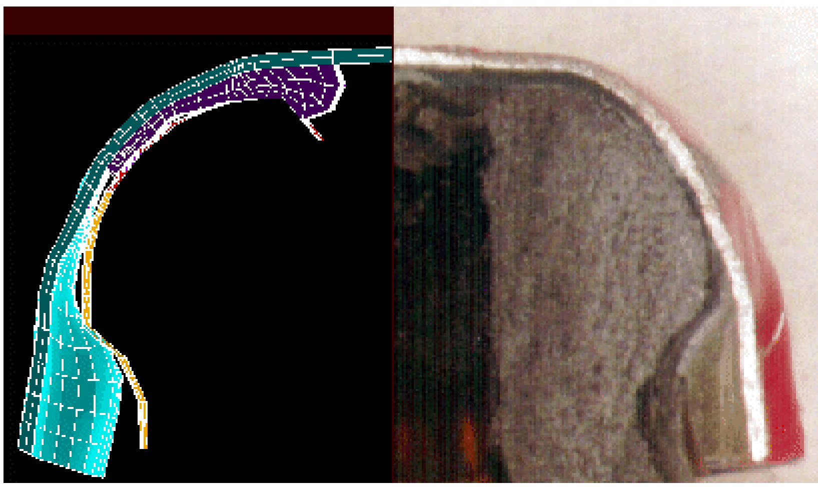


Figure 16. Geometry comparison, on wave



As we have said before, the last thing to do is to apply a pressure on the inner face of the cap. This is done in 14 steps of 0.1 MPa. Figure 17 shows the cap under a pressure of 1.4 MPa.

The global force exerted by the bottle on the gasket, is determined for each pressure increment. This information is used for predicting the leakage pressure, as explained in the next section.

PREDICTED LEAKAGE PRESSURE

To find the characteristic of the gasket we simply compress a cap on the neck of a bottle (without crowning) and record the force exerted on the cap and the leakage pressure using a manometer (see figure 18 for a scheme of the experiment).

With the calibrated strain gage we are able to determine the global force exerted on the cap when leakage occurs. Figure 19 shows experimental results and the calculated regression line.

Figure 17. Pressure, 1.4 MPa

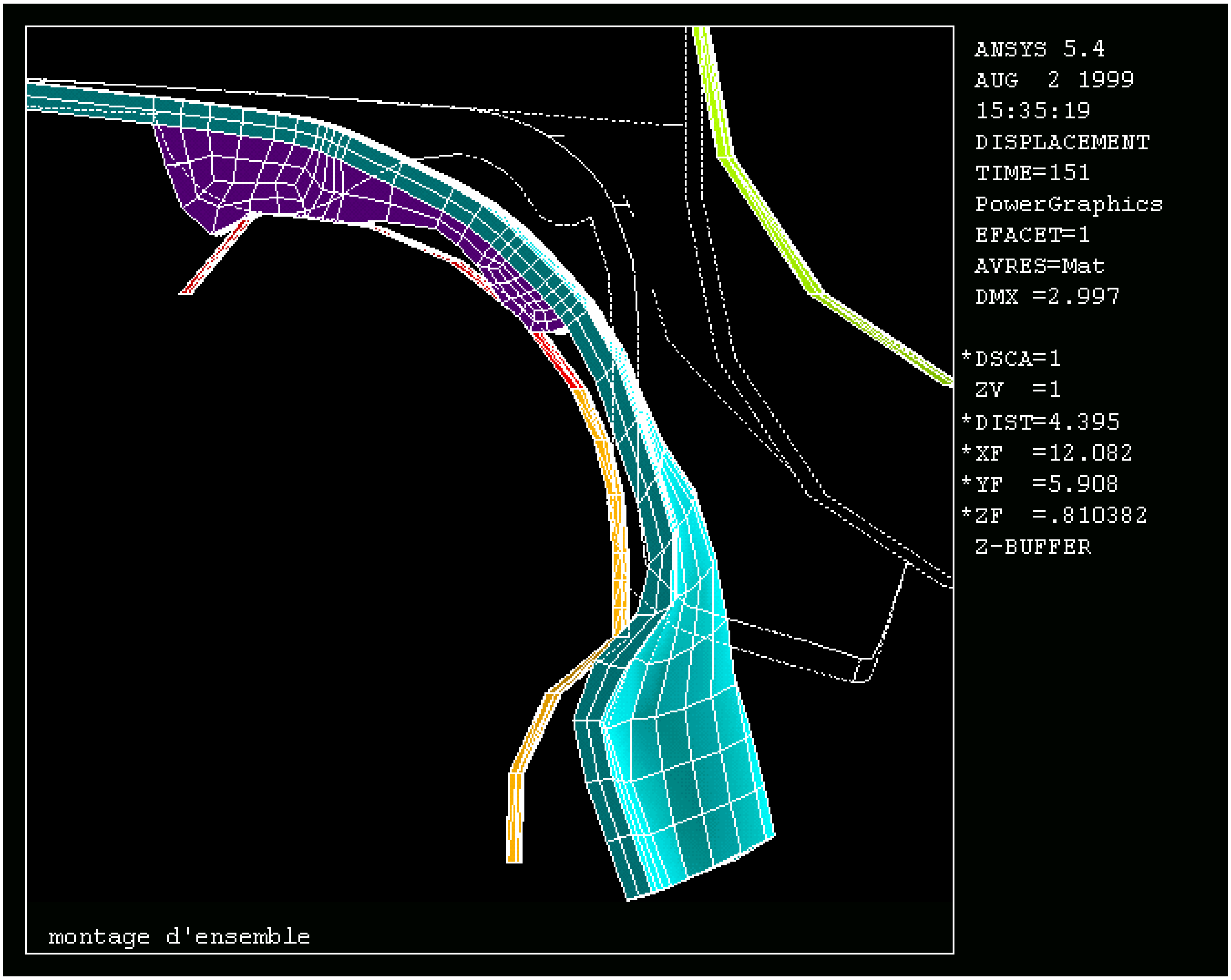
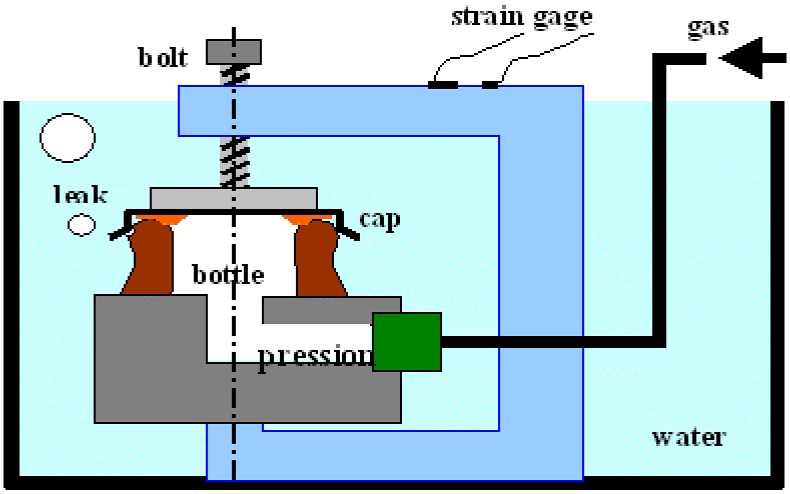


Figure 18. Experiment to determine the leakage pressure of a simply compress cap on a bottle neck



Now we are able to predict the leakage pressure of an assembly. On figure 20 the red line is the calculated force on the gasket obtained by substracting the force exerted on the cap by the pressure from the global experimental response force, shown on figure 19. The blue line on figure 20 is the result of the contact force exerted by the bottle on the gasket in the Ansys® simulation. The leakage pressure of the assembly is simply the pressure at the intersection point of these two lines. This gives as shown on the figure 20 a pressure of 1140 kPa.

The leakage pressure of a real assembly, measured in laboratory, lies between 1200 kPa et 1300 kPa. So, inspite of the coarse mesh of the FE model, the predicted leakage pressure is very close to the range of the real one.

Figure 19. Experimental results

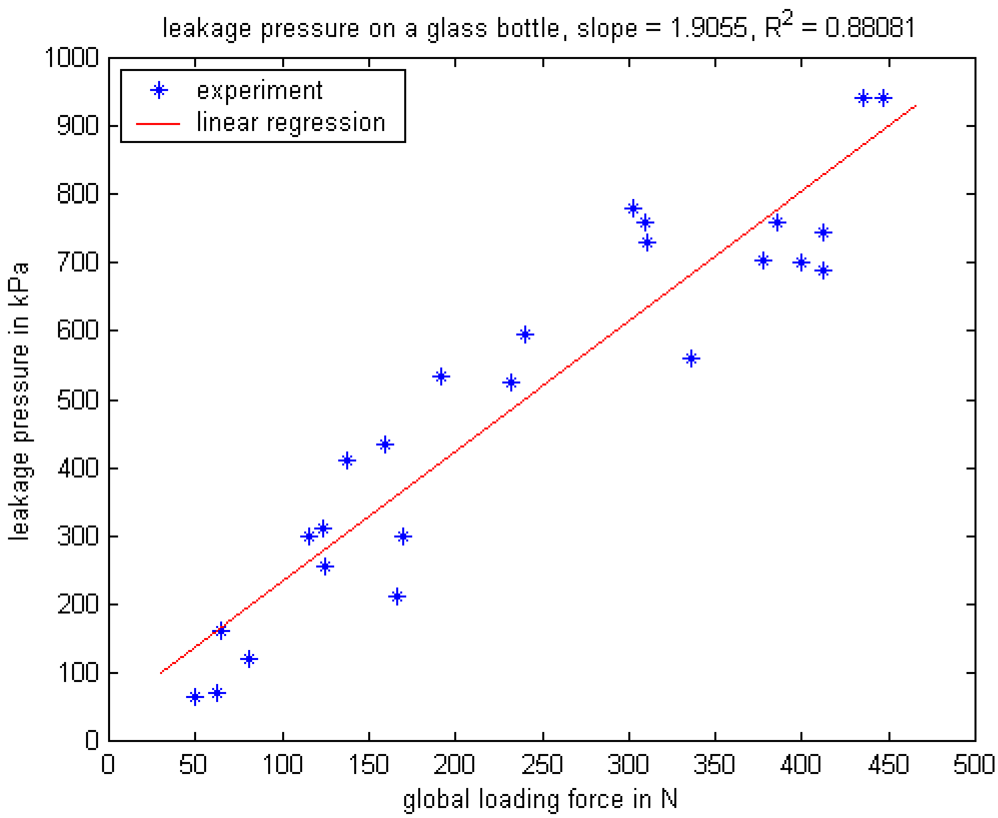
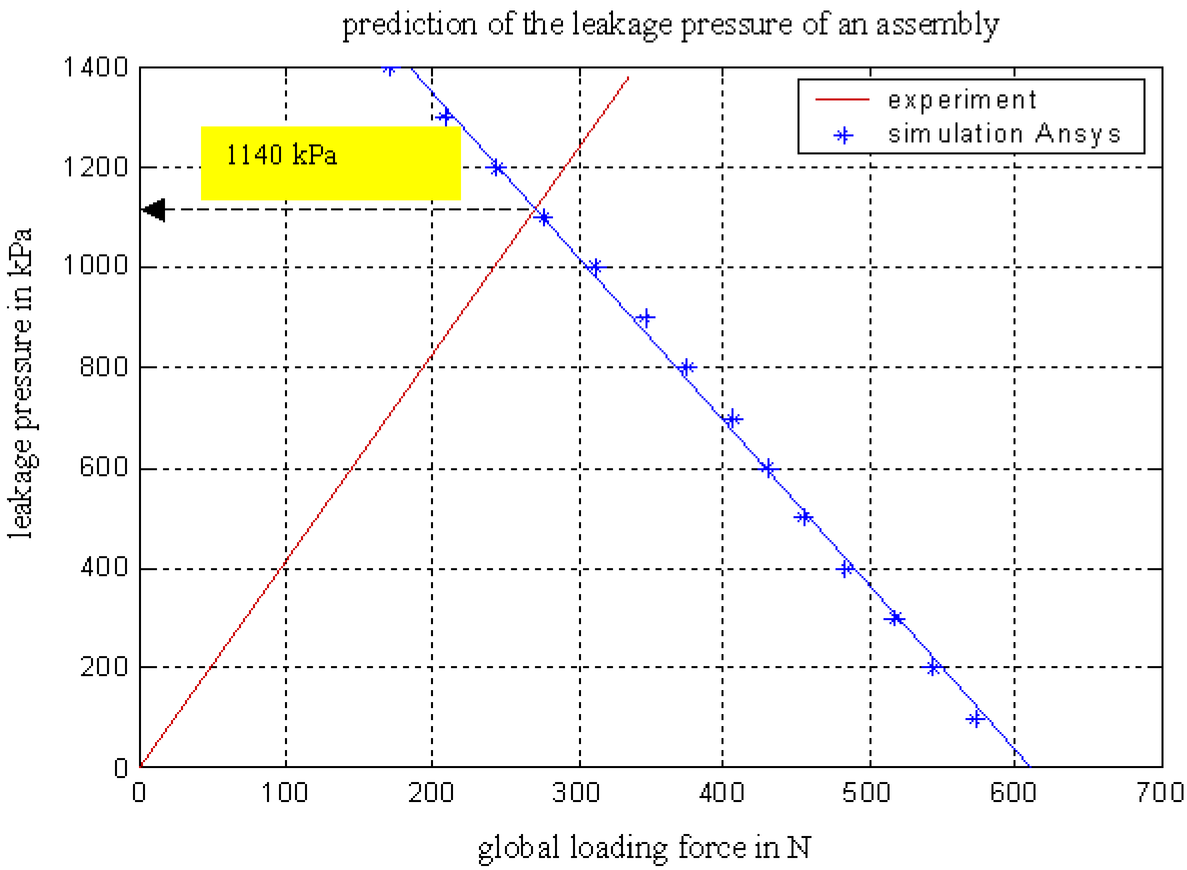


Figure 20. Predicted leakage pressure



CONCLUSION

In this paper we have shown the methodology used to predict the leakage pressure of an assembly by crowning. First we run a 3D FE simulation of the crowning operation in the Ansys® environment. The final simulated geometry of the crowning cap compares very well with a real assembly. In parallel with the simulation, experiment has been conducted to determine the sealing property of the gasket-bottle set. By combining the FE results with the experimental sealing property, we are able to predict the leakage pressure of a given assembly. The predicted result is very close to the leakage pressure of real assemblies.

Some improvements can be made on the simulation. A finer mesh will give more accurate results. In the simulation the material of the metal cap is considered elastic perfectly plastic. A bilinear stress strain relationship with appropriate strain hardening would be another improvement factor. Residual stresses in the metal cap due to punch-and-die operation have been neglected. A more complete study, one should include punch-and-die steps, which would certainly take much longer time.

On the experimental side we can improve our equipment to get better accuracy.

In summary, the results presented here prove the validity of the approach which can be extended for similar cases.

ACKNOWLEDGMENTS

The authors would thank the department of mechanical engineering at École de technologie supérieure for the availability of the equipment, and ZapatA® company for molding the specimen and the laboratory data for the leakage pressure of assemblies.

REFERENCES

1. Champliaud H., Unpublished Ph.D. Thesis, Analyse par éléments finis du sertissage de capsules d'étanchéité', École de technologie supérieure, Montréal, 2000.

2. Ansys® (1997). Ansys Basic Analysis Procedure Guides, release 5.4. Ansys® Inc., September 1997.

3. Barrett, P.R., Zemitis, W.S. (1998). Analysis Methodology for Simulating Battery Cell Assembly. 1998 Ansys Conference Proceedings, (1), pp. 471-481, August 17-19 1998, Pittsburgh.

4. Ansys® (1997). Ansys Theory Reference Manual, release 5.4. Ed. Kohnke P., Ansys® Inc.

5. Mooney, M. (1940). A Theory of Large Elastic Deformation. Journal of Applied Physics, (6), pp. 582-592.

6. Rivlin, R.S. (1984). Forty Years of Nonlinear Continuum Mechanics. Proc. IX, Int. Congress on Rheology, Mexico, pp. 582-592.

7. Lamit, L.G. (1998). Basic Pro/Engineer with references to PT Modeler. PWS publishing.

8. Zeid, I.. (1991). CAD/CAM Theory and Practice. McGraw-Hill.

9. MatLab® (1999). The Language of Technical Computing, Function Reference. (1 et 2), The MathWorks, Inc.

10. Dubé, J-P., Firmin T. (1998). Using Simulation in the Design of a Compliant Tail Connector. 1998 Ansys Conference Proceedings, (1), pp. 541-549, August 17-19, Pittsburgh.

11. Pilcher, M., Burman, Å., (1998). Finite Element Modeling and Testing of Rubber Pipe Seals. 1998 Ansys Conference Proceedings, (1), pp. 521-529, August 17-19, Pittsburgh.