Numerical Simulation and Optimization of the Forging Process

Jean-Loup Chenot, Pierre-Olivier Bouchard, Lionel Fourment, Patrice Lasne

ICFC Proceedings

Numerical Simulation and Optimization of the Forging Process

Jean-Loup Chenot[1], Pierre-Olivier Bouchard[1], Lionel Fourment[1] and Patrice Lasne[2]

[1]CEMEF, Mines ParisTech and CNRS UMR7635, B. P. 207 - F-06 904 Sophia-Antipolis, France, [2]Transvalor S. A., 694, av. du Dr. Maurice Donat, 06255 Mougins Cedex, France

Abstract: The objective of optimizing *automatically* industrial forming conditions in order to achieve a desired objective goal is now possible due to continuous increase of fast and parallel computers. Optimization requires that the computer simulation is accurate enough, that the material behavior is precisely identified and that the optimization parameters are properly selected.
To achieve the first goal, the fundamental mechanical assumptions and the basic principles of three-dimensional finite element discretization are briefly recalled. Several important numerical developments for efficient computation of large plastic deformation are mentioned.
The second requirement is fulfilled not only by experimental tests and identification of the material parameters of the constitutive law. Is it also necessary to predict the possible onset of defects such as cracking by introducing damage modeling..
Before optimization, a parameter sensitivity analysis must be performed in order to select the most important factors: shape of the preform, tools geometry, etc. The practical optimization can be carried out by a evolutionary algorithm technique associated with a surface response method.
Several examples of applications will be presented to illustrate the effectiveness of the procedure with the FORGE3 computer code. The optimization criterion can be on the forming force, on the material weight or on the final strength of the part.

Keywords: Forging, Plastic Deformation, Simulation, Finite Element Method, Optimization

1. Introduction

The forging process is widely used in industry to produce work-pieces which need to fulfil high quality requirements. Design of the forging sequences to assess feasibility is the first concern of engineers and numerical simulation is more and more utilized to replace costly and time consuming trial and error approaches on real materials and production facilities. Numerical modelling of plastic deformation by the finite element method started in the 70's in two dimensions [1-3] and in the 80's in three dimensions [4, 5], mostly at the academic level. It is only in the 90's that real industrial examples could be run successfully (see for example [6]) and commercial codes started to be proposed and begun to be used extensively by engineers in industry.

In addition, since 2000 many numerical improvements have made computations faster and more accurate and now the possibility of process optimization could be investigated practically in 2-D first: see [7, 8], and more recently in 3-D: refer to [9, 12].

Moreover there is an increasing effort to predict the final microstructure obtained at the end of the forming process, summarized in [13], which is also necessary to estimate the final mechanical properties so that an optimization of the strength of the parts will be also considered. In [13] a first approach is presented for predicting void closure by forging.

2. Mechanical and numerical approach

For a more detailed account of the theoretical background, the reader is referred to [14].

2.1 Mechanical formulation

The mechanical law that is often used in metal forming is isotropic with an additive decomposition of the strain rate:

$$\dot{\varepsilon} = \dot{\varepsilon}^e + \dot{\varepsilon}^p \qquad (1)$$

Where $\dot{\varepsilon}^e$ is the elastic strain rate and $\dot{\varepsilon}^p$ the plastic or visco plastic strain rate. The elastic law depends on the Lamé coefficients λ and μ; it is written with Jauman derivative of the stress tensor for material objectivity:

$$\frac{d_J \sigma}{dt} = \lambda \operatorname{tr}(\dot{\varepsilon}^e) + 2\mu\dot{\varepsilon}^e \qquad (2)$$

The plastic law is expressed by a power law:

$$\dot{\varepsilon}^{p} = 1/K \left\langle (\bar{\sigma} - R)/K \right\rangle^{\frac{1}{m}-1} \sigma' \tag{3}$$

Where σ' is the deviatoric stress tensor, $\bar{\sigma}$ is the usual equivalent stress, K is the consistency and m is the strain rate sensitivity.
More general material behaviour can be used:

- Temperature dependant laws,
- Anisotropic law with kinematic hardening,
- Anisotropic law described by e. g. Hill or Barlat equation.

Between work-piece and tools, a unilateral contact with friction modelled by a generalized Coulomb law is considered:

$$\tau = -\mu(\sigma_n)\Delta v / |\Delta v| \tag{4}$$

Where σ_n is the normal stress and Δv the velocity difference between tool and part.

For an incompressible or quasi incompressible flow, it is desirable to utilize a mixed formulation. In the domain Ω of the part, this formulation is written for any virtual velocity field v^* as:

$$\int_{\Omega} \sigma' : \dot{\varepsilon}^* dV - \int_{\Omega} p \, \mathrm{div}(v^*) dV - \int_{\partial\Omega_c} \tau v^* dS = 0 \tag{5}$$

and the mass conservation is imposed with any virtual pressure field p^* by:

$$-\int_{\Omega} (\kappa \, \mathrm{div}(v) + \dot{p}) p^* dV = 0 \tag{6}$$

Equations (5) and (6) are often rewritten in term of displacement and stress increments before the finite element space discretization.

Prediction of in service strength imposes to model the mechanical stress in real condition and to take into account possible failure. Therefore it is necessary to introduce damage evolution, not only in the structural computation, but also during the forming process. Among several damage laws, the isotropic Lemaître approach is a satisfactory compromise between accuracy and complexity; it is formulated in term of rate of a damage parameter w:

$$\dot{w} = \begin{cases} \dfrac{\lambda_{pl}}{1-w}\left(-\dfrac{Y}{S_0}\right)^{b} & if\ 0 \le T_x \quad and \quad if\ \bar{\varepsilon} > \bar{\varepsilon}_d \\ \dfrac{\lambda_{pl}}{1-hw}\left(-\dfrac{Y}{S_0}\right)^{b} & if\ -\dfrac{1}{3} \le T_x < 0 \quad and \quad if\ \bar{\varepsilon} > \bar{\varepsilon}_d \\ 0 & if\ T_x \le -\dfrac{1}{3} \quad or \quad if\ \bar{\varepsilon} \le \bar{\varepsilon}_d \\ 1 & if\ w \ge w_c \end{cases} \tag{7}$$

Where λ_{pl} is the plastic multiplier, T_x is the triaxiality and $b, S_0, \bar{\varepsilon}_d, h, w_c$ are parameters that must be identified with an experimental approach.

2.2. Finite Element Discretization

A mixed incremental displacement and pressure formulation ($P1^+/P1$) is used: the pressure field is discretized with linear tetrahedral elements, and a bubble function is added for the velocity or the displacement field. This formulation is well fitted for meshing and remeshing and the resulting stiffness matrices exhibit a much better conditioning.

The finite element formulation, which is developed at the laboratory level and implemented in the commercial code FORGE3, must be compatible with the following numerical and computational constraints:

- Automatic remeshing based on geometry to avoid element distortion and to follow accurately the tools surface;
- Unilateral contact analysis;
- Iterative solving of very large non linear systems by the Newton-Raphson method with line search;
- Iterative solving of linear systems with efficient preconditioning;
- Parallel computing using parallel domain decomposition;
- Possibility of adaptive remeshing, utilizing error estimation to control accuracy of the computation in term of energy norm;
- Multi body analysis with an improved master and slave method for treating evolving contact;
- Easy transfer of physical internal parameters, when multi physic coupling must be taken into account.

3. Optimization procedure

Starting from an initial design of a sequence of forming, optimization is often necessary to improve quality of the work-piece and hopefully to decrease production cost in order to keep competitiveness. It has been achieved mostly by trial and error, firstly using craftsman experience and utilizing actual equipments and materials. Since the availability of reliable three-dimensional computer codes, optimization can be achieved utilizing numerical simulation, as it appears faster and less expensive for finding a better solution.

More recently it was realized that these optimizations can also be performed by coupling an optimization module with a finite element computer code for simulating the process. First, one needs to define a set of technical parameters which can be varied practically and possibly to prescribe a range of admissible variation. Then a cost function is introduced which represents the objective and the minimization of which will provide the best solution for our problem.

Several methods have been tested in research laboratories to minimize efficiently the cost function. The so-called direct minimization methods and the adjoint state method are relatively fast, but they require the evaluation of complex derivatives of the cost function and may lead to a local minimum only.

To-day the necessity to treat a wide variety of problems, and the availability of relatively cheap parallel computers, allow us to utilize evolutionary algorithms. These methods are easier to implement and test, as they need only the computation of the cost function and therefore they do not require complex coding for the evaluation of derivatives. The number of evaluations of the cost function is generally high, but it can be greatly reduced when it is combined with metamodelling with a Meshless Finite Difference Method. This method was developed by Fourment et al in [12], fully coupled with the FORGE3 computer code and it is shown on several industrial examples that about one hundred simulations are necessary to reach a satisfactory optimum.

4. Examples of process optimization

4.1. Forming optimization of a crankshaft

Crankshafts can be formed using either a cylindrical preshape or a preshape obtained by wedge rolling. It is well known in forging practice that the filling of the dies is possible when the flash is large enough as it shown in Figure 1.

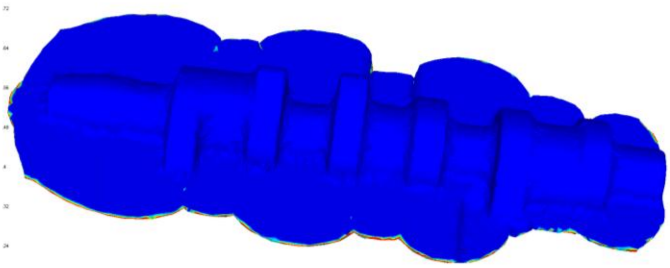

Figure 1: forging of a crankshaft

However it is obvious that too much flash results in a loss of material and an increase of the forging force so that a compromise must be found, by varying the shape of the preform. This can be achieved by introducing a cost function to be minimized as a linear combination of the volume of the preform and of a function representing the lack of filling of the dies. Two sets of optimization parameters are compared:

- The simplest one is corresponds to a cynlindrical preshape. It includes only two parameters: length and diameter, represented with the mesh in Figure 2;

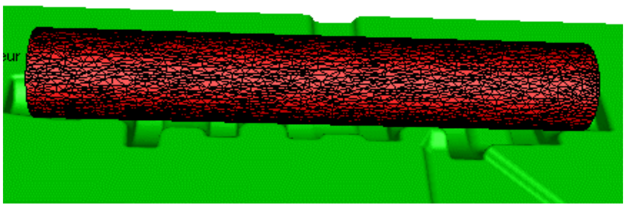

Figure 2: Mesh of the cylindrical preshape

- A more complex one with five parameters: total length and four diameters, corresponds to a wedge rolled preshape, as pictured in Figure 3.

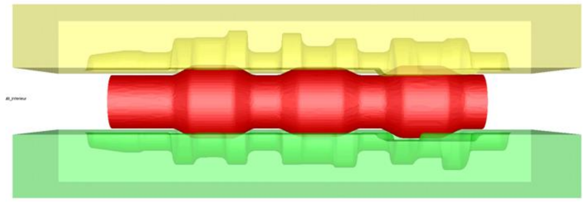

Figure 3: preform with 5 parameters obtained by wedge rolling

After optimization the filling of the dies is well imposed with a significant material saving due to decrease of the flash. In Table 1 the principal features of the computation are summarized.

Table 1

Number of parameters	Number of function evaluations	Number of processors	Total CPU time	Material saving
2	40	2	80h	2%
5	120	20	24h	5%

Similar examples of process optimizations can be found in reference 15.

4.2 Forging of a screw head

The second forging process which is considered is forming of a screw head, where optimization of the geometry will result in an improvement of the strength in service. The initial preform of the screw is shown in Figure 4.

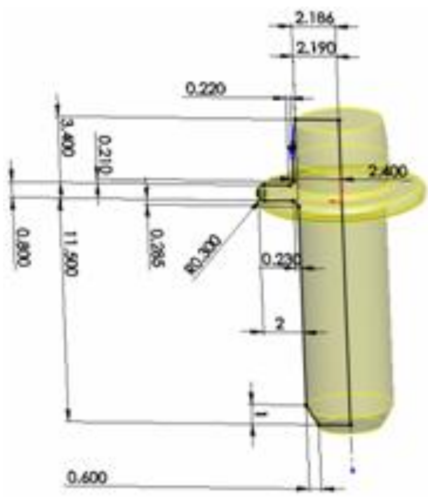

Figure 4: definition of the initial preform of the screw

A simulation of the forging of the Philips head is followed by a virtual screwing to evaluate the effectiveness of the forming. A Hansel Spittel constitutive equation is used for a C15 steel:

$$\sigma = A_1 e^{m_1 T} \bar{\varepsilon}^{m_2} \dot{\bar{\varepsilon}}^{m_3} e^{m_4/\bar{\varepsilon}} \qquad (8)$$

with material parameters defined by: A_1 = 738, m_1 = -0.0011, m_2 = 0.22065, m_3 = 0.01342, m_4 = 0.00065. The elastic parameters are: E = 2 10^5 MPa and ν = 0.3.

For optimization of the head two parameters are defined as shown in Figure 5 and the finite element simulation code Coldform is linked to the CAD system SolidWorks.

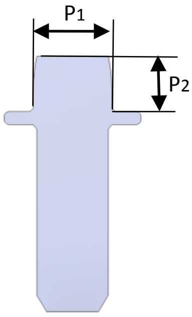

Figure 5: Definition of the two optimization parameters for the preform of the screw

The geometry of the screwdriver is shown in Figure 6. The strength of the head is modelled by imposing a torque to the screwdriver while the remaining of the screw is blocked in rotation.

Figure 6: Philips screwdriver

The progressive deformation of the head of the screw is then observed and pictured in Figure 7.

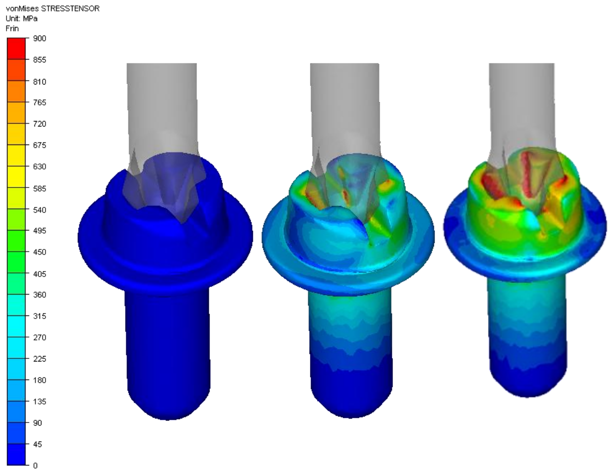

Figure 7: evolution of iso von Mises stress distribution during screwing

The criterion for optimisation is a measure of the gap between the formed head and the Philips. After about 80 evaluations of the cost function, the filling of the upper die is satisfactory and gives the shape pictured in Figure 8. The initial values of the optimization parameters are: p_1 = 5.48mm and p_2 = 3.38mm and after optimization their new values are: p_1 = 5.68mm mm and p_2 = 3.21mm

Figure 8: final Philips shape after forging

The relatively small variations in the optimization parameters induce noticeable changes in the final shape as is shown in Figure 8.

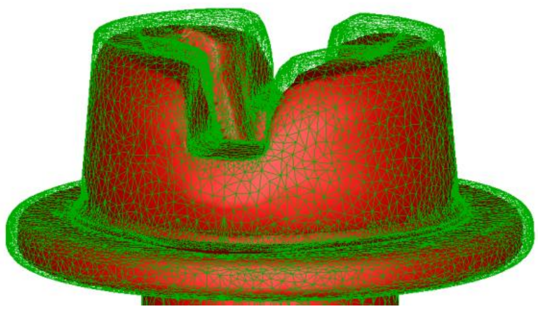

Figure 9: comparison of the head shapes - in red before optimization, in green after optimization.

The consequence of a better filling of the upper die to fit more accurately the Philips shape is illustrated in Figure 10 where the behaviour during screwing shows that optimization provides higher strength of the head.

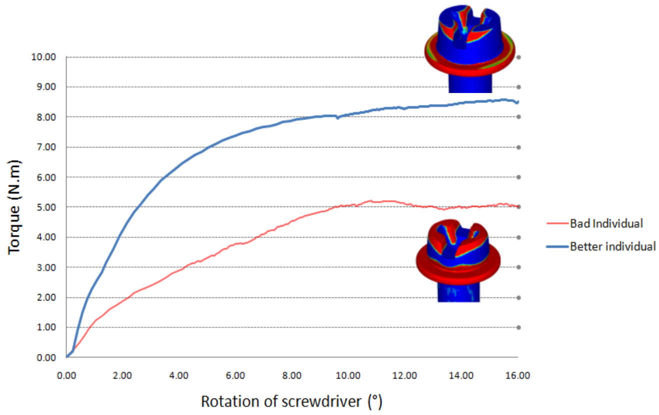

Figure 10: strength of the screw in term of torque vs. rotation angle – Before optimization: lower curve, after optimization: upper curve

4.3 Clinched joint

The clinching process is used for assembling two sheets without any additional rivet or bolt. A cylindrical punch deforms locally the sheets in a die and produces a joint. For our numerical model an isotropic plastic law was utilized with a hardening law of the form:

$$\sigma_0(\bar{\varepsilon}) = \sigma_y + K\bar{\varepsilon}^n$$

where σ_0 is the yield stress, and $\bar{\varepsilon}$ is the equivalent strain, the other parameters were identified using experimental tests as: σ_y = 46 MPa, K = 430 MPa, n = 0.34.

A Lemaître damage law is introduced and possible local failure is modelled using the classical kill element method. Due to the geometry of the process, it is possible to perform the simulations with the axisymmetric version of the code in order to save CPU time. The initial mesh of the two sheets is shown if Figure 11 and the deformed sheets after clinching are pictured in Figure 12.

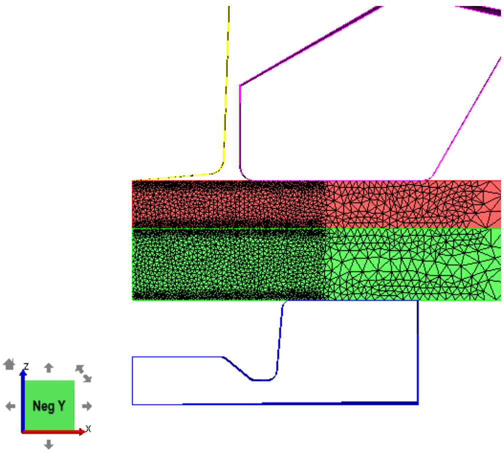

Figure 11: clinching process - initial mesh

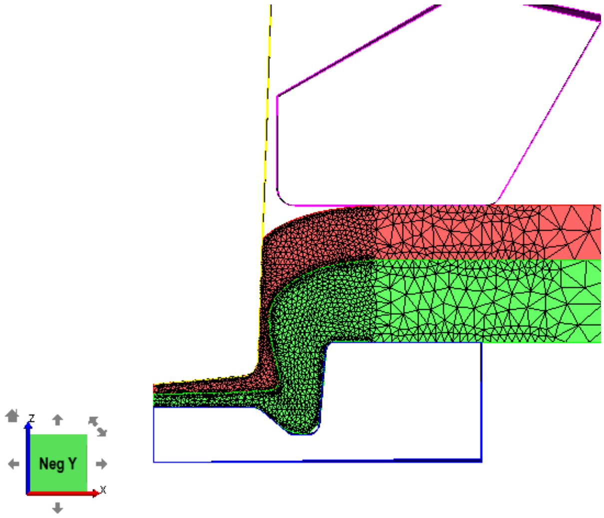

Figure 12: clinching simulation –final mesh

The strength of the joint can be evaluated virtually by submitting it to a mechanical test, for example a tension test as is pictured in Figure 13. It is worthy to remark that the upper sheet is broken during the test.

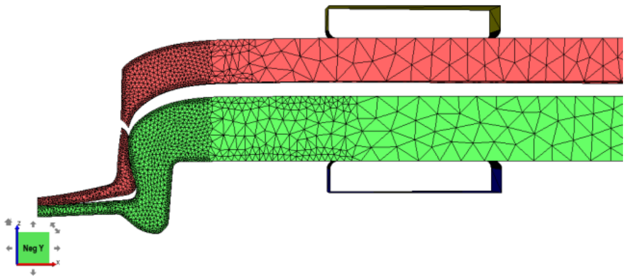

Figure 13: tension test of a clinched joint

The failure of the upper sheet is due to damage occurring during clinching which increases dramatically during the tension test as is shown in Figure 14.

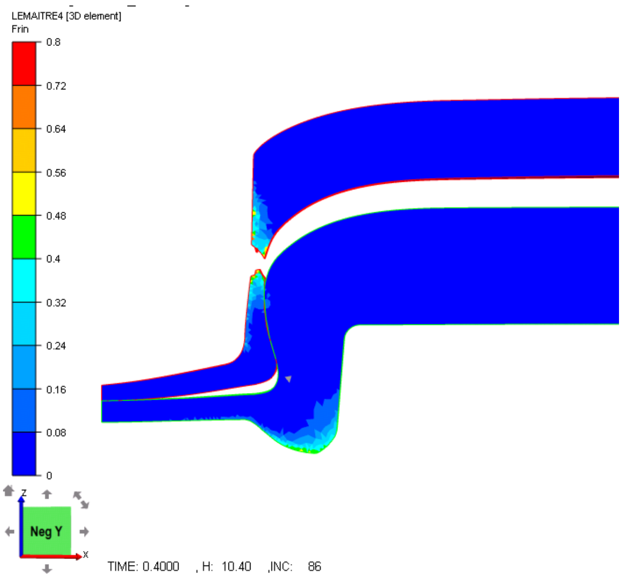

Figure 14: tension test - iso Lemaître damage distribution

In the industrial process, several parameters can be modified in order to improved effectiveness of the joint, especially resistance of the assembly to a tension force. Two geometrical parameters are selected: the radius R_m of the punch and the depth P_m of the die, shown in Figure 15.

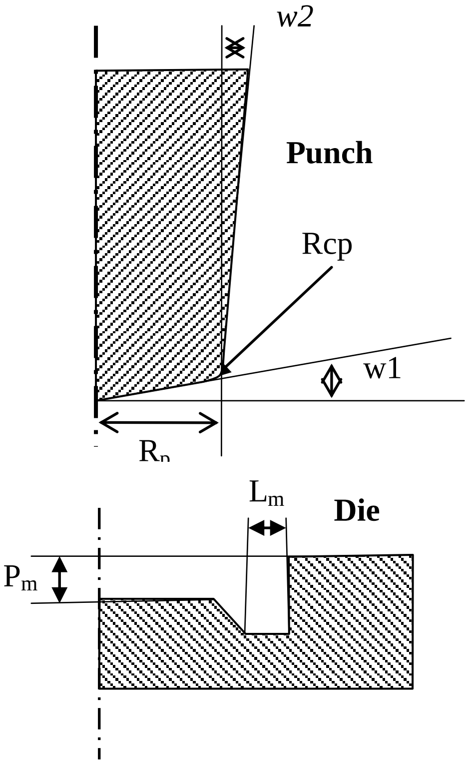

Figure 15: definition of the two geometrical parameters for optimization

Optimization is performed on these two parameters with a cost function which is defined as the maximum tension force on the clinched joint before failure. The surface response is progressively built using cost function evaluations. It is represented in Figure 16 after 30 evaluations of the cost function, i. e. after 30 simulations of the clinching process followed by the corresponding tension tests.

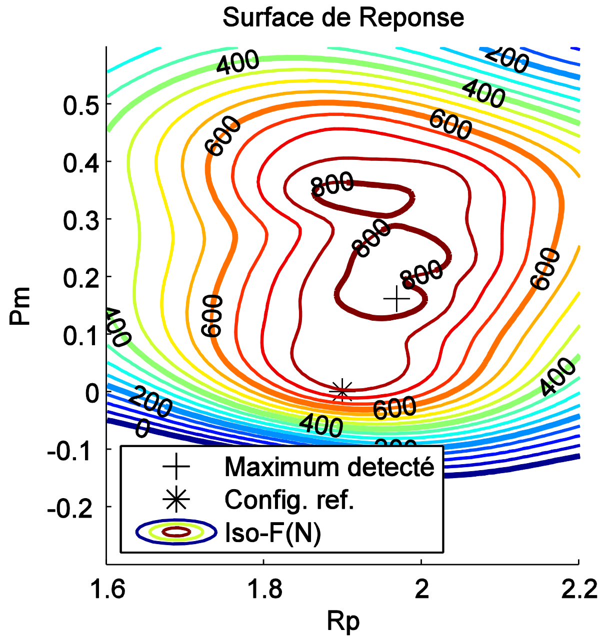

Figure 16: Surface response after 30 computations of the cost function

In Figure 17 the final shapes of the assembly are compared and one can observe that the damage due to forming is lower when the tools are optimized.

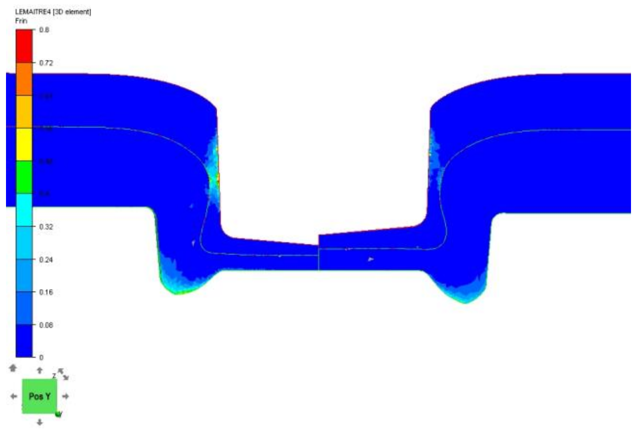

Figure 17: map of iso Lemaître damage after clinching
Left with initial tools; right with optimized tools

The tension test is illustrated in Figure 18 when the sheets are separated; one remarks that damage is lower than in the case of the non optimized configuration so that the upper sheet is not broken.

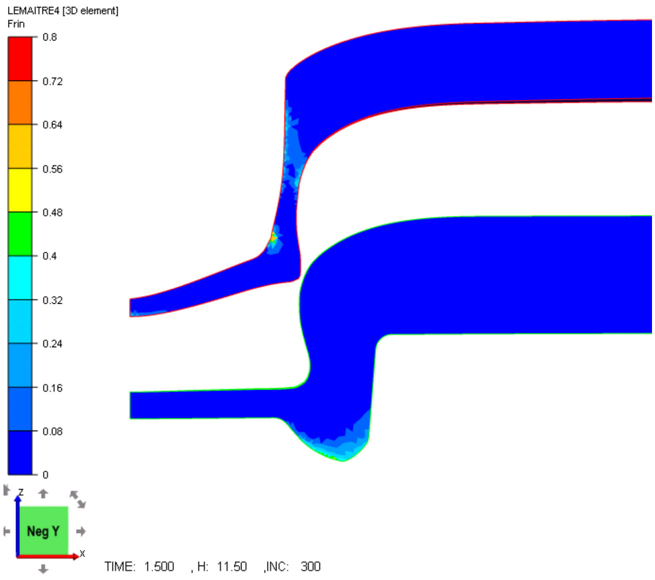

Figure 18: tension test of the optimized clinched joint

In the tension test we observe that the ultimate force, which is equal to 737 N before optimization, goes up to 840 N after optimization, which represents an increase of about 14% with the same material.

5. Conclusions

We have described a general approach for finite element simulation and automatic optimization of metal forming processes, which is implemented in the FORGE3 commercial code. Three examples were briefly analyzed to demonstrate the effectiveness of the procedure. It is expected that this advanced numerical tool will be more and more utilized by engineers to optimize industrial processes in a faster and more economical way and to find better solutions for producing work-pieces.

We have already shown that optimization can performed not only on the forming process itself but also on the in service mechanical behaviour of the parts. In the future, more sophisticated physical models will be implemented to predict and optimize the final properties in the work-piece.

Aknowledgement

The authors are grateful to the Cetim (Centre des Industries Mécaniques) for partial funding of this work, to Mr. Mickael Barbelet from Transvalor for providing the results of calculations in *section 4.2* and to Mr. Emile Roux from CEMEF for the data of *section 4.3*.

References

[1] Zienkiewicz O. C., Valliapan S., King I. P., 1969, Int. J. Num. Meth. Eng. , 1, 75-100.
[2] Lee C. H., Kobayashi S., 1973, Trans. ASME, J. Eng. Ind., 865, 95.
[3] Cornfield G. C., Johnson R. H., 1973, J. Iron Steel Inst., 567, 211.
[4] Surdon G., Chenot J.-L., 1986, International Conference on Numerical Methods in Industrial Forming Processes Numiform'86, ed. by K. Mattiasson et al., A. A. Balkema, Rotterman, 287-292.
[5] Surdon G., Chenot, 1987, Int. J. for Numerical Methods in Engineering, 24, 2107-2117.
[6] Soyris N., Fourment L., Coupez T., Cescutti J.-P., Brachotte G., Chenot J.-L., 1992, Engineering Computations, 63-81, 9.
[7] Kusiak J., Thompson E. G., 1989, International Conference on Numerical Methods in Industrial Forming Processes Numiform'89, ed. by Thompson E. K., A. A. Balkema, Rotterman, 569-574.
[8] Fourment L., Balan T., Chenot J.-L., 1996, International Journal for Numerical Methods in Engineering, 39 (1), 33-66.
[9] Castro C. F., Costa Sousa L., Conceiçao Antonio C. A., César de Sa J. A., 2000, Eccomas Conference, Barcelona.
[10] Laroussi M., 2003, 3-D sensitivity analysis by the adjoint state method – Application to forging, Ph. D. thesis, Mines ParisTech, CEMEF (in French).
[11] Do M. T. T., Shape optimization in 3-D forging, Ph. D. thesis, Mines ParisTech, CEMEF (in French).
[12] Marie S., Ducloux R., Ejday M., Massé T., Fourment L., Bobadilla C., Montmitonnet P., 2009 in 12th ESAFORM conference on material forming. Enschede, 27-29.
[13] Chenot J.-L., Bernacki M., Fourment L. & Ducloux R., 2010, International Conference on Numerical Methods in Industrial Forming Processes Numiform'2010, Pohang, Korea.
[14] Wagoner R. H., Chenot J.- L., 2001, Metal forming analysis, Cambridge University Press, Cambridge.
[15] Ejday M., Fourment L., 2010, 13th ESAFORM Conference on Material Forming, Brescia - Italy - April 7-9, 2010.