CAD/FEM COUPLING IN SHAPE OPTIMIZATION

P. Morelle[1] , P. Duysinckx[2] , C. Fleury [2].

CAD/FEM coupling is now more and more popular with regard to finite element analysis codes through a lot of classical capabilities:

- interfaces between CAD software and Finite Element software;
- built-in CAD capabilities for data definition (geometry, boundary conditions, material data, etc.) in finite element codes;
- built-in mesh generation capabilities regarding CAD softwares.

The problem is much more complicated as far as shape optimization is concerned because, not only the geometry of the body must be taken into account, but also the relations between the CAD entities (i.e. the geometrical constraints). This is a key point for an efficient shape optimization code.

In this paper, the following aspects are discussed:

- definition of design variables;
- definition of a parametric CAD-file;
- sensitivity analysis when using free mesh generators;
- handling of geometric constraints;
- use of variational geometry in shape optimization;
- use of external CAD softwares;
- general architecture of the code.

It is shown how these problems have been solved with the new optimization module (OPTI) of the SAMCEF system. Several examples of 2-D and 3-D shape optimization problems are given which illustrate the concepts.

---

1 SAMTECH S.A., Bd. Frère Orban 25, B-4020 Liège
2 University of Liège, L.T.A.S., rue E. Solvay 21, B-4000 Liège

DIE KOMBINATION VON CAD UND FEM IN DER FORMOPTIMIERUNG

P. Morelle[1] , P. Duysinckx[2] , C. Fleury [2].

Rechnerprogramme mit der Kombination von CAD und FEM erfreunen sich immer größerer Beliebtheit. Dies ist auf die folgenden Mehrmale zurückzuführen:

- Die Verbindungen zwischen CAD- und FEM-Rechnerprogrammen (interface);
- Die Möglichkeit dar Definition von Daten mittels CAD-Rechnerprogrammen (Geometrie, Randbedingungen, Materialdaten, usw.)
- Die Möglichkeit mittels dem CAD-Rechnerprogramm Netzwerke zu generieren.

Im Bezug auf die Formoptimierung stellt diese Kombination ein außerordentlich kompliziertes Problem dar denn es muß nicht nur die Geometrie der Struktur berücksichtigt werden, sondern auch die Beziehung zwischen bestimmten CAD-Größen (z.B. geometrische Begrenzungen). Dies ist der wichtigste Punkt für eine effiziente Formoptimierung.

In der vorliegenden Veröffentlichung werden die folgenden Gesichtspunkte behandelt:

- Definition der Konzeptionsvariablen;
- Definition eines parameterisierten CAD-Files;
- Analyse der Sensibilitäten im Falle der Verwendung eines beliebigen Netzwerkgenerators;
- Handhabung der geometrischen Begrenzungen;
- Anwendung einer variablen Geometrie in der Formoptimierung;
- Anwendung von externen CAD-Rechnerprogrammen;
- Generelle Architektur des Rechnerprogramms.

Es wird gezeigt, wie diese Probleme im Fall des neuen Optimierungmoduls OPTI von SAMTECH gelöst werden.

Mehrere Beispiele von zweidimensionalen und dreidimensionalen Formoptimierungsproblemen werden gezeigt und ihre Konzepte erklärt.

1 SAMTECH S.A., Bd. Frère Orban 25, B-4000 Liège

2 University of Liège, L.T.A.S., rue E. Solvay 21, 4000 Liège

CAD/FEM COUPLING IN SHAPE OPTIMIZATION

P. Morelle[1] , P. Duysinckx[2] , C. Fleury [2].

1. INTRODUCTION.

1.1 History and general concepts of shape optimization.

As a result of prolific research works, the general concepts of shape optimization were defined in the middle of the eighties. At that time, fundamental contributions were brought by some authors, among which Braibant and Fleury (see [1],[2],[3]), to give to shape optimization a trend towards creation of systems able to handle real-life practical problems. At the University of Liège, those researches led to the creation of the shape optimization module OPTI of SAMCEF. Nevertheless, these works have established the general architecture of commercial and industrial shape optimization software. The structural optimization problem is formulated as a mathematical programming problem:

$$\min_{z} f(z)$$

submitted to

$$C_j(z) \leq \overline{C}_j \qquad (j = 1,m)$$
$$\underline{z}_i \leq z_i \leq \overline{z}_i \qquad (i = 1,n)$$

where z is the design variable vector (of dimension n) and f(z) is the objective function, while the inequalities define the design restrictions (in number m) and the design variables are limited by lower and upper bounds.

Regarding the function evaluation cost (one function evaluation requires one finite element analysis run of the whole problem), this non-explicit and non-linear optimization problem is replaced by an approximate explicit one which is submitted to mathematical programming algorithms. So, a series of approximate subproblems are generated and solved until convergence occurs. Nevertheless, the formulation of the subproblems requires additional information about the gradients: $\nabla f(z)$ and $\nabla c_j(z)$. So, the first task of any optimization package is to perform an accurate sensitivity analysis.

Once the sensitivity information is available, an optimization explicit problem can be formulated. Numerous research efforts were dedicated (see Fleury [13] and [14], Svanberg [15], Haftka [16]), to study and define accurate approximations leading to convergence as quickly as possible. According to their robustness and reliability, convex and conservative approximations are now generally adopted to generate optimization models.

1.2 Problems of interest in this research.

Even if the baselines of shape optimization are already available, even now several difficulties are not completely solved. Some of them stem from the need to update shape optimization techniques to continuing advances in other parallel research fields: e.g. emergence of reliable free mesh generators, advances in C.A.D.[3] , growing importance for interactive and

---

1 SAMTECH S.A., Bd. Frère Orban 25, B-4020 Liège
2 University of Liège, L.T.A.S., rue E. Solvay 21, B-4000 Liège
3 C.A.D. Computer Aided Design

integrated design tools. The purpose of the following developments is to provide a satisfactory answer to some of the problems appearing in modern optimization. One has to note that previously some of them were well described by Braibant in [4].

The first important problem we consider concerns design variable definition. It has been clearly illustrated, e.g. by Braibant and Fleury in [1], that regularized problems of shape optimization are well defined problems. For planar problems, a parametric representation of boundaries by an assembly of splines, Bezier curves, straight lines etc... controlled by master points eliminates "parasitic" or "pseudo" optimal configurations that usually appear when F.E. nodes are chosen as design variables. Nevertheless, when designing industrial pieces, engineers are greatly interested by shapes composed of an assembly of simple curves such as straight lines, circles, ellipses, etc. with continuity requirements. Then, natural design variables are not yet pole positions, but "global" structural sizes. It is much easier and more flexible to define the optimization problem in term of lengths, radii, angles, or positions of special points, than to use exclusively master nodes description. Indeed, technological requirements inherent to manufacturable shapes are taken straight into account when using the appropriate variables. A lot of equality constraints (sometimes too strict to be submitted to optimizers) are naturally eliminated from the suitable formulation. Accordingly, the idea was to introduce a shape optimization tool which can basically consider design variables master node as well as curved global parameters (e.g. radius) and structural sizes. The objective is met by allowing the choice of design variables among the set of parameters which can be defined in any data set of a SAMCEF-BACON model.

A second limitation of the previous shape optimization tools is related to the narrow choice of mesh-generators that was available. The main objective was to allow users to take advantage of free mesh generators and not to restrict themselves to transfinite techniques. First, transfinite mesh generators appear badly conditioned when large geometric variations occur. Major mesh distortions are involved and can seriously deteriorate the quality of the F.E. accuracy as well as the sensitivity analysis. Finally, the obligation to partition the domain into quadrangular macro-elements has been revealed to be somewhat tedious and inflexible in complicated shapes while locally adjusting mesh density can appear almost impossible. Since a few years, new reliable and efficient free meshers have emerged and are now available to yield F.E. grids over complicated domains with variable mesh densities. For planar problems, mesh generators by offset or triangulation (e.g. Delaunay meshers) are now common use while the first 3-D meshers have been already introduced in commercial software. So, it is an interesting alternative to consider these new tools in a modern shape optimization package.

Finally, our third objective is to experiment with an integrated optimization tool within a C.A.D. environment. The guide line was to follow the trend towards integration of design system ranging from modelling to automatic redesign. As these concepts have already been successfully applied in other packages (e.g. CAOS [5]), our wish is to experiment the analysis modules of SAMCEF (ASEF, DYNAM, STABI) and the OPTI optimization package closely integrated with BACON (the modeller). A data basis link was provided for an easy data transfer management.

2. A NEW PARAMETRIC SHAPE OPTIMIZATION IN SAMCEF-OPTI.

2.1 A parametric approach of shape optimization:

The new optimization procedure stems from the capability of defining geometrical models of structural components with parameters when using the input data module BACON of SAMCEF. One can introduce parameters in the BACON module's structural model data set. The optimization procedure allows the designer to declare some of them as design variables and lets OPTI modify their value to meet objectives and constraints previously defined. Any static or geometric characteristics which can be evaluated during the linear F.E. analysis in the ASEF module can be chosen as objective functions or constraints.

The OPTI module is in charge of carrying out a sensitivity analysis and submitting the optimization model to synthesis modulus for redesign. The selected sensitivity analysis method is the discrete form of the semi-analytical approach. Close integration of the geometric modeller of BACON within the OPTI sensitivity procedure is realized to generate shape perturbations. BACON yields to OPTI perturbated geometries respecting every geometrical constraints expressed in the original model. Then, smoothing can be applied to perform mesh flow computation. The redesign analysis is executed by the synthesis moduli. Different types of convex approximations as well as several mathematical programming algorithms are available to solve optimization subproblems; among which we can find the new version of the CONLIN[4] optimizer and its dual solvers, Sequential Quadratic Programming and Inverse Linearization.

After the redesign modulus, a new data set with better values of the design parameters is then produced to be submitted back to BACON and to re-iterate with another design cycle.

2.2 Towards an integrated design tool.

Another goal was to experiment with an integrated design tool in SAMCEF. Several ingredients intervene to create an attractive environment to engineers: a unique data basis, the possibility to define the problem in an interactive and user-friendly way as well as to have a homogeneous description of quantities considered in geometrical and material model, F.E. analysis and optimization procedure.

In the new SAMCEF version, the different modules communicate through a unique data basis: the SAMCEF data basis (D.B.). Information is accumulated along the design process and transferred from one task to another in order to provide a fairly easy data transfer management. So, the use of data bases avoids complicated manipulations of files and the design chain is also simplified.

Another important aspect of an attractive design environment is the possibility to designate every characteristic of the problem in a simple and unified manner. The idea is to provide the capability to define each characteristics of the F.E. analysis or optimization problem by referring to drawing or modelling entities. The whole problem description can be addressed through references to the drawing objects as points, lines, domains, groups etc. Material properties, loading cases and fixations as well as constraints and objective function definitions are specified in terms of their physical meaning in the design model. Then, it must be observed that the entire problem definition can be realized interactively with the graphics tools available in the BACON module.

2.3 The design variables: an independent set of parameters.

As previously explained, the design variables are chosen among the parameters introduced in the SAMCEF-BACON data set to build up the design model. Of course, the design parameters are to be selected independent from eachother. In the first version of the module, it is the designer's responsibility to make this crucial choice properly. Nevertheless, an interactive tool is also available in the BACON module to modify interactively any parameter value and then verify linear dependency of the selected design variables. It is also possible to check the validity of the definition of the design model and thus avoid the shape degenerating during design modifications.

Despite verifying the quality of the parameters, this approach has many advantages and provides great liberty to the design task. Master-slave parameters can be introduced when defining a parameter as a function of other parameters (most of the usual analytic functions are available). One also can take full benefits of the facilities available in the C.A.D. system. The constraints that are explicitly expressed during the geometrical model will be respected throughout the redesign. So, geometrical constraints, such as parallelism, tangency,

---

4 CONvex LINearization

intersections, etc., are considered in an easy and natural way.

These constraints are implicitly taken into account in the model and can then be eliminated from the ones submitted to the optimizer. Indeed, the conditions expressed are respected when perturbating the structural shape. The modeller generates perturbation which is "kinematically admissible" with regard to the defined constraints.

3. SENSITIVITY ANALYSIS WITH FREE MESH GENERATORS.

3.1 A discrete semi-analytical approach

After the linear F.E. analysis in the ASEF module, the sensitivity analysis is performed in OPTI. Among all the approaches available, we selected the semi-analytical method applied to the discretized model.

The discrete formulation of the equilibrium equation of the F.E. structural is:

$$Kq = g \qquad (3.1)$$

where K, q and g are respectively the stiffness matrix, the generalized displacements and forces.

After differentiating equation (3.1), we get the displacements sensitivities with respect to design variable z_i:

$$\frac{\partial q}{\partial z_i} = K^{-1}\left(\frac{\partial g}{\partial z_i} - \frac{\partial K}{\partial z_i} q\right) \qquad (3.2)$$

The semi-analytical approximation consists in evaluating the derivatives of the stiffness matrix and generalized forces by finite differences:

$$\frac{\partial q}{\partial z_i} = \frac{K(z_i + \delta z_i) - K(z_i)}{\delta z_i} \qquad (3.3)$$

$$\frac{\partial g}{\partial z_i} = \frac{g(z_i + \delta z_i) - g(z_i)}{\delta z_i} \qquad (3.4)$$

3.2 The sensitivity analysis problem: determining the velocity field.

To understand the problem of sensitivity analysis with free mesh generators, we have to focus on the derivation of stiffness matrix. Like matrix K itself, it is built by assembling each elementary contribution:

$$\frac{dK}{dz_i} = \sum_{k=1}^{NEL} L_k^T \frac{dK_k^e}{dz_i} L_k = \sum_{k=1}^{NEL} L_k^T \frac{K_k^e(z_i + \delta z_i) - K_k^e(z_i)}{\delta z_i} L_k \qquad (3.5)$$

where L_i is an incident matrix for each element, and NEL is the number of F.E. in the mesh. As we see in (3.5), if we want the sensitivity analysis to be accurate, it is imperative to conserve the same mesh topology during the perturbation process. If a mesh update is performed during perturbation, the mesh topology can change involving a different number of finite elements as well as a modification of F.E. degree of approximation; the result is a complete degradation of sensitivity analysis. So, the mesh topology must be conserved during perturbations. As even small perturbations of the domain can modify the F.E. grid generated by free meshers, the mesh topology has to be frozen and a criterion for positioning the F.E. nodes has to be defined to compute the mesh flow after boundary variation.

By definition, the *velocity field* is the first derivative of the position of any point P(**x**) relative to change of design variable z_i:

$$V_i = \frac{\partial x}{\partial z_i}$$

At first order, after perturbation of z_i, the new position of P becomes:

$$x\left(z_i + \delta z_i\right) = x(z_i) + V_i\, \delta z_i$$

So, the velocity field is the law which gives the first order perturbation of nodes after a small variation of its boundary. Determining V_i or increments of nodal position after perturbation is the same problem. Accordingly, to generate the pertubated stiffness matrices $K(z_i+\delta z_i)$ and loads $g(z_i+\delta z_i)$, one has to determine velocity fields first.

3.3 Determining the velocity field

The velocity field computation can be split into two steps. The first step determines the velocity field on the boundaries and points where positions are fixed by the model description. As a result of coupling BACON to OPTI, the increments of node positions on the boundaries can be computed by submitting from OPTI perturbated data sets to the geometrical modeller of BACON. The drawing models relative to the perturbated set of parameters are built up with a chronology identical to the one which was used to create the original model ("newspaper file").

The second step is to know the new node positions in the inner part of the solid change. The velocity field law is used to link the inner node movements to boundary changes. Several laws have been exposed in literature. The first velocity field used the intrinsic coordinates of Patches (e.g. Coons patches). Belegundu and Rajan in [6] and [7], Zhang and Beckers in [8] and [9] have proposed to use the mechanical behaviour of the structure to determine the velocity field. Other fields can be obtained with relocalization techniques or smoothing procedures. These techniques have been successfully applied in several domains from aero-elasticity to solid mechanics with reliability and efficiency.

3.4 Smoothing procedures to solve the velocity field definition.

The sensitivity analysis needs to determine increments of node positions resulting from a design variable perturbation and shape modification. The choice of "smoothing procedures" can reveal numerous advantages:
- The simplicity of the method and the easiness of applying it to a wide range of problems.
- The complexity of the velocity field problem is drastically reduced. Indeed, the smoothing procedure is a scalar technique (as thermal distribution solution), so planar and solid problems are replaced by a sequence of two or three uncoupled scalar problems (one per coordinate).
- The inherent drawbacks of smoothing procedures are minimized or alleviated. For structural problems, smoothing has revealed to be highly reliable. Since we consider only small increments of node positions and that the basic F.E. grid is of fairly good quality (this condition is generally fulfilled by good mesh generators), mesh degenerescence is highly unprobable during the smoothing process. The element shapes are changed very little. Even for 3-D meshes, smoothed small increments are insufficient to invert F.E. elements or to put a node into the volume of a neighbouring element. Finally, potential movements of nodes to the exterior of the domain in the presence of non-convex local geometry are not possible since the displacements of nodes on boundaries are fixed by the geometric modeller. It can also be added

that those drawbacks, if they appeared, could be minimized by choosing variants of Laplacian techniques.

Before discussing different smoothing techniques, it is interesting to formulate this type of relocalization as a computation of a linear system solution. Each node can be connected to its neighbours by scalar stiffness $k_{i,j}$ so that it is possible to write the linear system:

$$\begin{bmatrix} K_{ii} & K_{ib} \\ K_{bi} & K_{bb} \end{bmatrix} \begin{bmatrix} \delta P_i \\ \delta P_b \end{bmatrix} = \begin{bmatrix} F_i \\ F_b \end{bmatrix}$$

where indices i and b make reference to boundary or inner nodes, while δp and K are node position increments and associated stiffness matrices. δp_b are the boundary node increments which are known after boundary perturbation. No forces are applied to inner nodes. The displacements of the velocity field are then expressed by:

$$\delta P_i = -\left(K_{ii}^{-1} K_{ib}\right) \overline{\delta P_b}$$

This is the law linking the inner node positions to the boundary node movements. The scalar law is applied to each coordinate.

A crucial problem is to select a proper stiffness between the nodes. The simplest technique is the pure Laplacian smoothing where $k_{i,j} = 1$ between nodes supported by common F.E. edges. To the authors' experience, this simple method is, nevertheless, reliable and robust in numerous practical applications. When the element sizes are very different, Robinson et al (ref. [10]) show it is preferable to take into account the element sizes through weighted factor function of the interfaces lengths. A power law has been adopted for stiffness: $k_{i,j} = l^{-p}$. The exponent p which can vary from 2 to 3, increases the stiffness effect on the smallest interfaces. This method exhibits very good results when applied to F.E. meshes presenting very refined zones. In this case, the most important requirements are localized in the biggest elements that are generally situated in the less critical zones, safeguarding the maximum mesh quality during perturbation. Other smoothing procedures using the element area or F.E. estimated error information can also be implemented. As Diaz et al suggested in [11] to use the F.E. error for mesh adaptation between optimization iteration, we can propose to use the element error estimators in weighted smoothing. In this case, the nodes are moved towards the elements with the smallest errors. This procedure is probably the best one since it yields a velocity field providing a uniform error state. Among all the velocity fields, it is the one which gives the smallest degradation of the mesh quality (at first order) in terms of the chosen error estimator.

4. APPLICATIONS.

4.1 Perforated plate under biaxial stress state [1].

This very classical example is treated as a first illustration of the new parametric approach. The objective is to find the perforation shape that minimizes the weight while Von Mises stresses does not exceed a given allowable value. The hole is chosen as an elliptic curve whose unknown parameters are the two principal axe values as well as the angle between the largest principal axis and the horizontal structural axis. The entire model is realized with references to only drawing entities. Stress constraints are selected by considering the F.E. neighbouring the ellipse curve. The CONLIN optimizer is used providing a quick convergence (5 to 6 iterations) towards the analytical solution. Even if CONLIN is sensitive to variable shifting, for that particular formulation of the problem, the solution convergence rate is preserved for a wide range of design variables shifting values.

Figure 1 illustrates the modifications of shape and stress distribution during the optimization loop.

4.2 Optimization of a helicopter rotor mast [17].

The design problem is to find a better design of a helicopter rotor mast (*figure 2*). The load case considered is the constant term of the Fourier development of pressure loads applied on the upper ring of the structure. The objective is to find the rotor profile that minimizes the maximum Von Mises equivalent stress on the outer contour of the structure. The boundaries are made of linear curves joined by a tangent circle. Delaunay free mesh generator is applied to the whole structure avoiding to divide the domain into quadrangular macro-elements as previously. The selected design parameters are the radius of the connection and the sizes of the upper ring. It is "implicitly" imposed that the circle remain tangent to the two straight lines and that the horizontal line remain horizontal after optimization. One more constraint has been added: an upper bound on the total weight which is violated in the initial design. This constraint is active at the optimum (the upper bound is reached).

Four F.E. analyses are needed to find the optimal configuration presented in *figure 3*. *Figure 4* compares the meshes and shapes in the initial and final designs; the stress concentration is reduced by more than 25 % from 60.94 to 46.64 as it can be seen on *figures 5* and *6*.

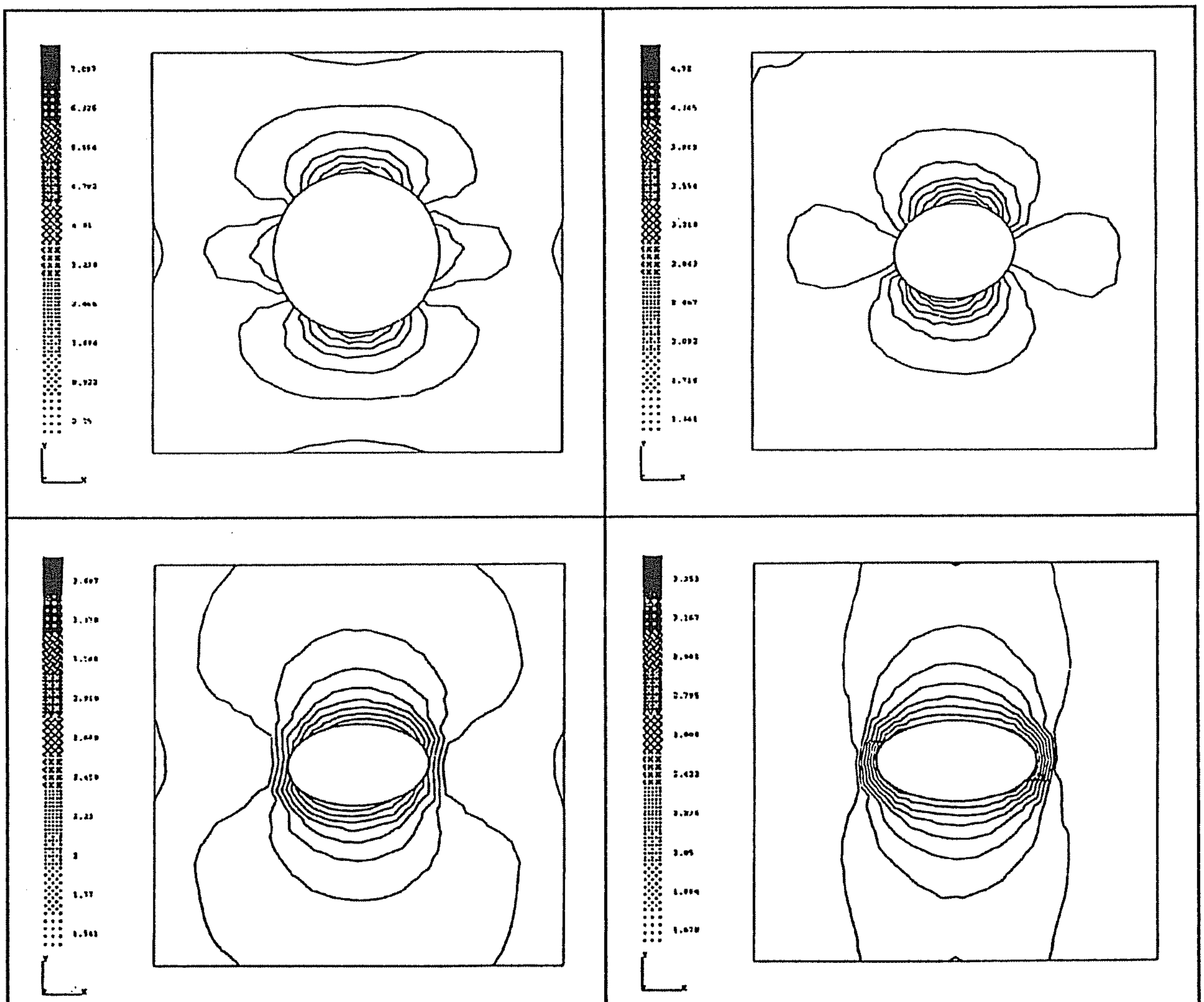


Figure 1 — Evolution of shape and stress distribution during iterations

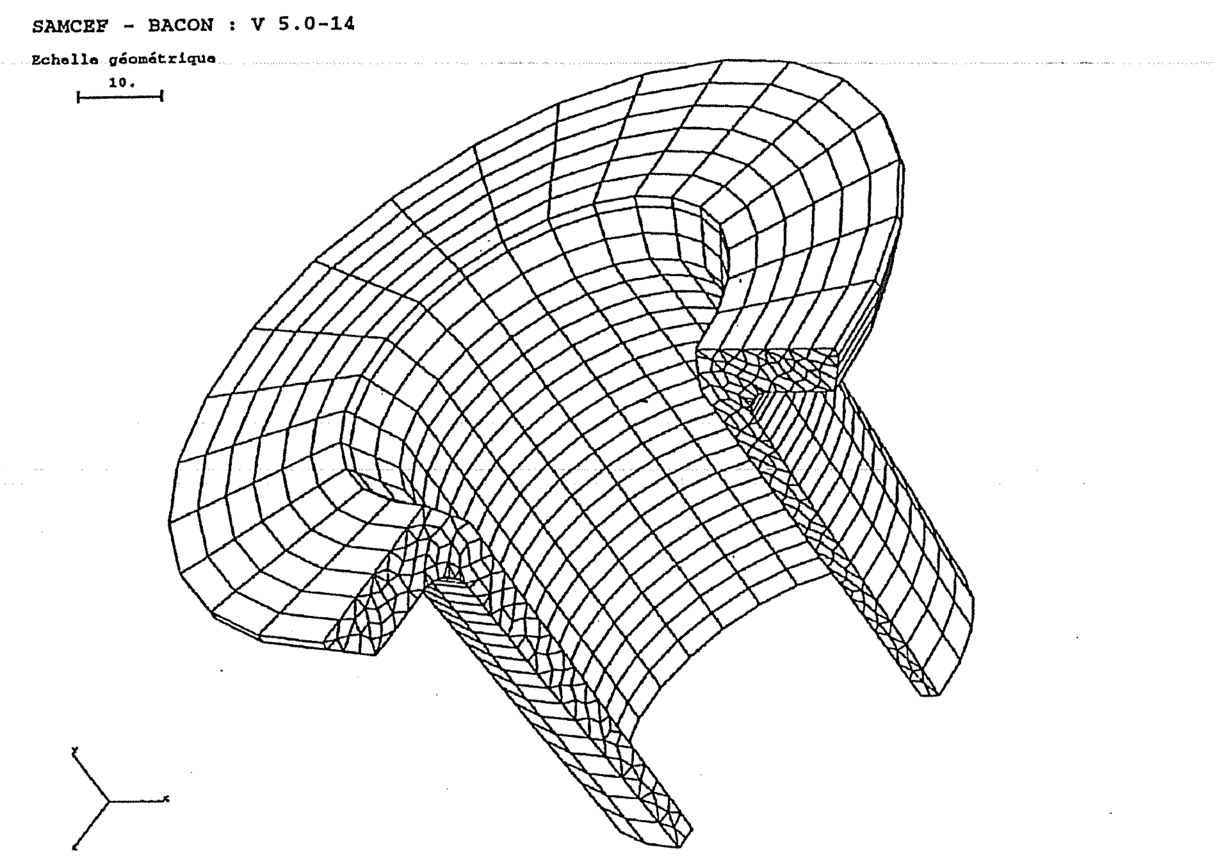


Figure 2

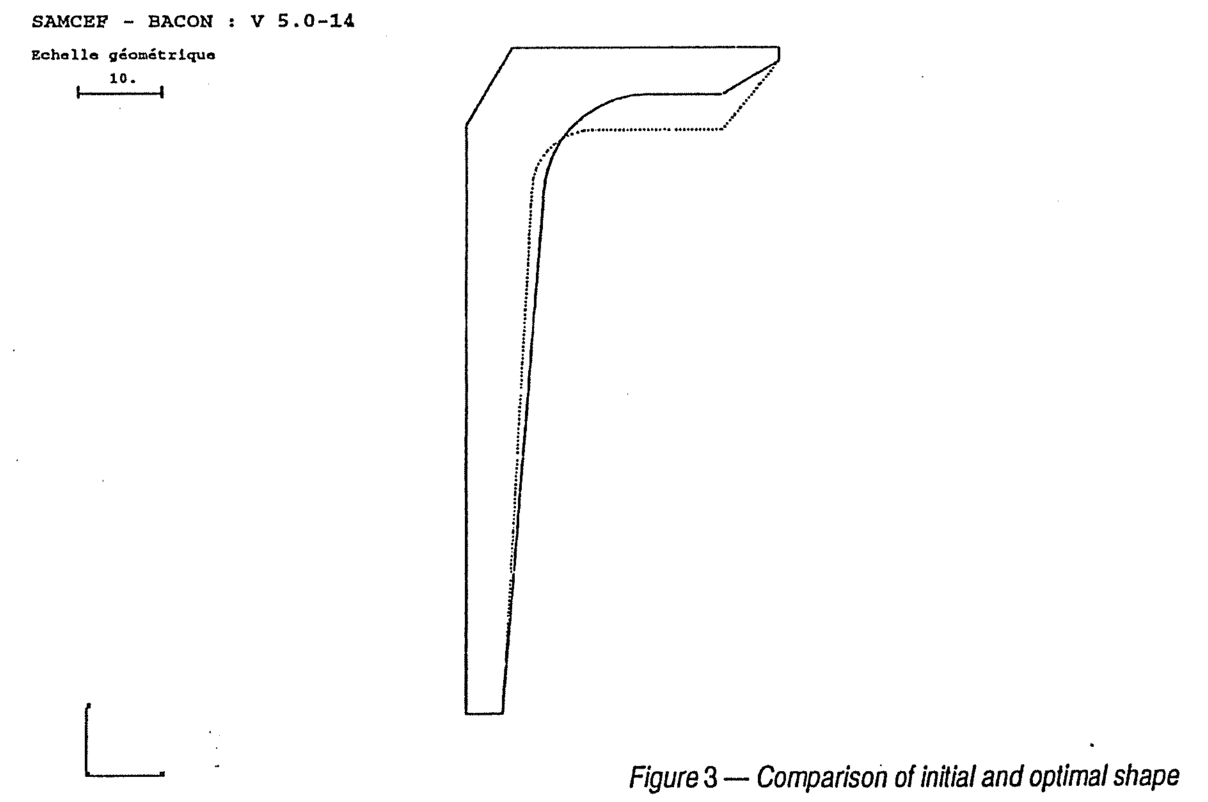
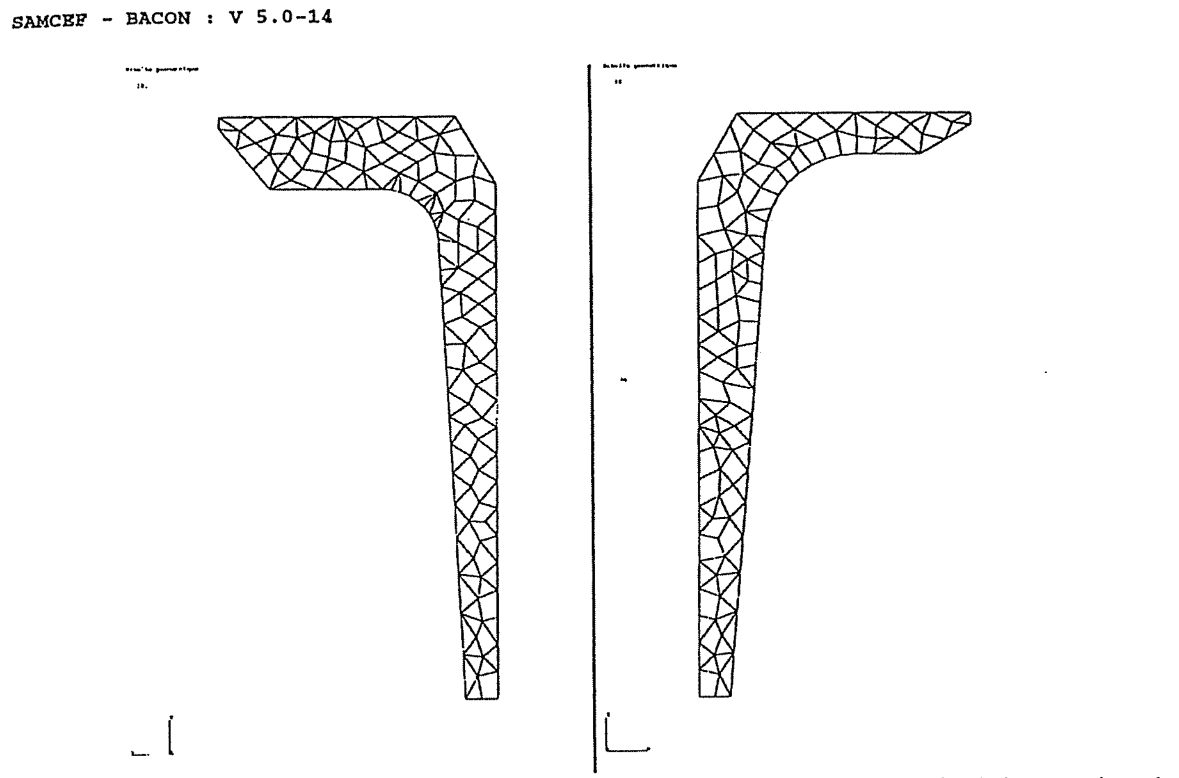


Figure 3 — Comparison of initial and optimal shape



Initial shape and mesh

Optimal shape and mesh

Figure 4

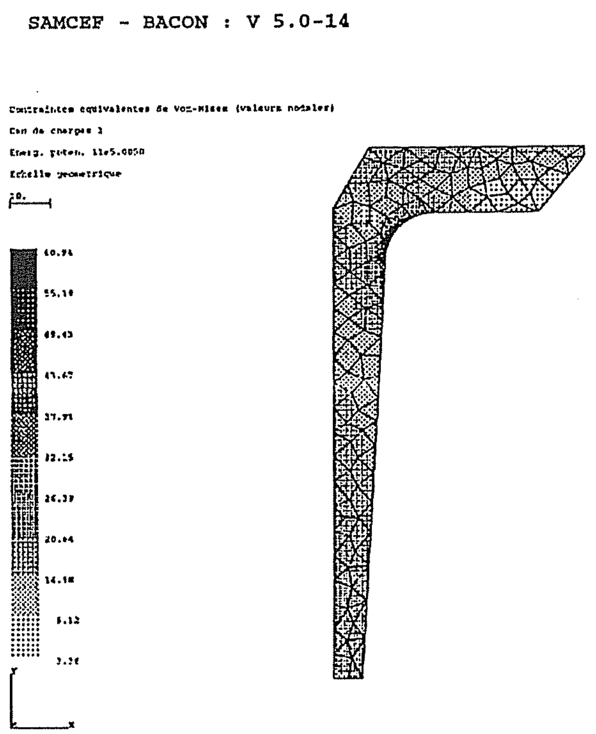


Figure 5 — Initial stress distribution (Von Mises stress at Gauss points)

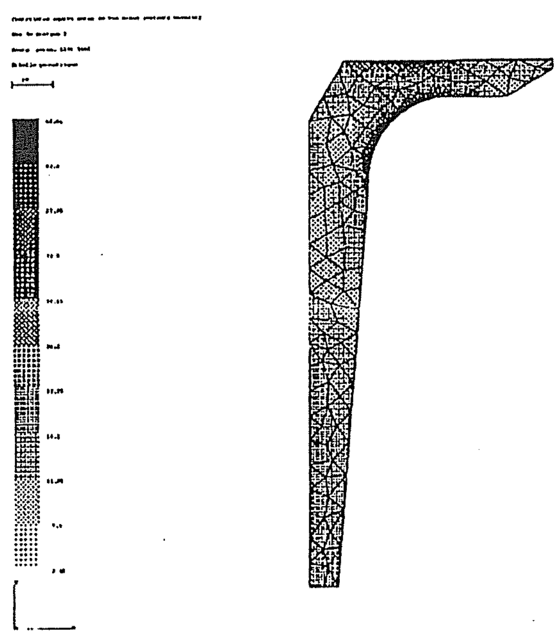


Figure 6 — Optimal stress distribution (Von Mises stress at Gauss points)

References:

[1] BRAIBANT V., FLEURY C.: "*Shape optimal design using B-splines*" Comp. Meth. in Appl. Mech. and Eng. n°44 (1984) pp 247-267.

[2] FLEURY C., BRAIBANT V.: "*Structural Optimization: a new dual method using mixed variables*", Int. Jnl Num. Meth. in Eng. vol 23, (1986) pp 409-428.

[3] BRAIBANT V., Ph.D.Thesis: "*Optimisation de forme des structures en vue de la conception assistée par ordinateur*", University of Liège, Applied Sciences Faculty, Collection of Publications n°102 (1986) (in French).

[4] BRAIBANT V., MORELLE P.: "*Shape optimal design and free mesh generation*", Structural Optimization 2, pp 223-231 (1990).

[5] RASMUSSEN J.: "*The structural optimization system CAOS*", Structural Optimization 2, pp 109-115 (1990).

[6] BELGUNDU A., RAJAN S.: "*A shape optimization approach based on natural design variables and shape functions*", Comp. Meth. in Appl. Mech. & Eng. (1988) pp 87-106.

[7] BELGUNDU A., RAJAN S.: "*Shape optimal design using fictitious loads*", AIAA Jnl vol 27 n°1 (1989) pp 102-107.

[8] ZHANG W.H., Ph.D.Thesis: "*Calcul des sensibilités et optimisation de forme par la méthode des éléments finis*", University of Liège, Applied Sciences Faculty (1991).

[9] BECKERS P.:"*Recent developments in shape sensitivity analysis: the physical approach*", Eng. Opt 1991, vol 18, pp 67-78.

[10] ROBINSON B., BATINA J., YANG H.: "*Aeroelastic analysis of wings using Euler equations with deforming mesh*", Jnl of Aircraft vol 28, n°11, Nov. 1991.

[11] DIAZ A., KIKUCHI N., TAYLOR J.: "*A method of grid optimization for finite element methods*", Comp. Meth. in Appl. Mech. and Eng. n°41 (1983) pp 29-45.

[12] KIKUCHI N., CHUNG K.Y., TORIGAKI T., TAYLOR J.: "*Adaptative finite element methods for shape optimization of linearly elastic structures*", Comp. Meth. in Appl. Mech. & Eng. n°57 (1986) pp 67-89.

[13] FLEURY C.: "*CONLIN an efficient dual optimizer based on convex approximation concepts*", Structural optimization n°1 (1989) pp 81-89.

[14] FLEURY C.: "*Efficient approximation concepts using second order information*", Int. Jnl for Num. Meth. in Eng. vol 28, pp 2014-2058 (1989).

[15] SVANBERG K.: "*The method of moving asymptotes-new method for structural optimization*", Int. Jnl for Num. Meth. in Eng. vol 24, pp 359-373 (1987).

[16] HAFTKA R., KAMAT M.P.: "*Elements of structural optimization*", Dordrecht: Martinus Nijhoff Publishers 1986.

[17] TROMPETTE P., LALLEMENT C., MARCELLIN J., SARLIN P.: "*AXIOPT, Un programme de calcul pour l'optimization de forme des structures axisymétriques*", Actes du 3e Colloque des tendances Actuelles en Calcul des Structures (1985), Bastia, Corsica, pp 731-748.