

Integrating ANSYS with Modern Numerical Optimization Technology - Part II : A Reverse Parametric Modeling Approach for Structural Shaping Optimization

Shen-Yeh Chen

Vice President, Engineering Consulting Division, CADMEN Group, TADC, Taiwan, R.O.C.

1. Introduction

Structural shape optimization methodologies are fairly mature. For example, the problem has been approached using stochastic methods like the genetic algorithms. [1-3]. However, one of the most important issues in structural shape optimization is still how to mathematically model the shape changes using the design variables. From the history, most of the methods can be categorized into four different general methods.

The first approach uses the nodal coordinates as the design variables directly [4-5]. For this approach the number of design variables would be very large, and the quality of the finite element mesh may be deteriorated. Some researchers also try to add connection and constraints between nodes [6-7] in order to keep better mesh quality as well as reduce the total number of design variables. This approach can be built upon the existing FE packages without CAD modeling tools available, but intensive user input and judgement is necessary.

The second approach, the geometry-based approach, uses the controlling parameters in CAD or geometric model as design variables, and relates the nodal coordinates with the blending interpolation or parameter of the geometry [8-12]. Some researchers also consider the locations of the boundary nodes only for design variables [13-15]. For getting better mesh quality, there are also algorithms available to re-distribute the internal nodes. Hwang and Fleury [16], for example, re-located the node based on optimizing the shape quality of each element. Another approach is to use the Laplacian formulation for re-distributing the internal node [17-18]. Only small changes on the boundary are allowed. The approach mentioned above will require internal information from the CAD system, therefore intensive programming effort may be necessary. Also a proper parametric model on the CAD level is crucial. This approach sometimes suffers from the problem of inconsistent mesh pattern after re-meshing, unless a mapping mesh procedure or re-distribution of mesh nodes is implemented. Sometimes controlling the error estimate of the FE analysis was used to minimize the inconsistency caused by re-meshing [19].

The other two methods seem to be popular in the commercial structural optimization packages, like MSC/NASTRAN [20], and GENESIS [21]. The first method, which is called the fictitious load method or the Natural Shape Function approach [22], uses an auxiliary structure, and applies fictitious load as the design variables. The structural shape and mesh is then updated according to the deformation of the auxiliary structure. This approach has relatively good flexibility in the variation of structural shape, but the selection of the fictitious load and boundary conditions depends on the user's experience. An improved version of this approach, called the Hybrid Natural Approach [23], replaces fictitious load with displacements, and couples with geometry parameters to simplify the process of deciding design variables. Also the concept of velocity field matrix [24] was used to avoid real-time calculation of the deformed auxiliary structure. The improved version of this approach will require parametric information from the CAD model. Similar modeling methods can also be found in the field of computational fluid dynamic optimization, which constructs analog truss elements on the edges of the element [25].

The remaining approach uses domain elements or super elements to define the perturbed mesh domain with a few controlling key nodes [26-27]. The perturbed basis was calculated beforehand and the selection of the design variables was simplified. However, the flexibility of the structural shape variations will depend on the pre-defined mathematical model of the domain elements and sometimes the mesh still deteriorates near the intersecting boundary of elements.

Most of these methodologies mentioned have a few things in common. First, attempts have been made over the years to reduce the effort of input needed for controlling and deciding the final shape of the structure. These attempts were achieved by allowing the users to pick a few nodes only for altering the coordinates, and interpolating the coordinates of the rest of the nodes in some special formulation. The

nodes that are assigned as the design variables directly are called the *Controlling Nodes* in this paper. All other nodes, which are affected by the *Controlling Nodes*, are called *Passive Nodes*.

2. The Contour Natural Shape Function

2.1 Difficulty of the Geometry-Based Approach

The goal of this research was to develop a methodology that would take advantage of the existing methods. The tasks were then to minimize the user input effort but still retain reasonable flexibility in the design variables. The algorithm was also expected to use finite element meshes only without CAD information. The reason for minimizing the user input data and retaining reasonable flexibility is obvious. The reason for limiting the available information to finite element mesh only will be explained below. Note that these comments are suitable for the author's working environment but may not be universally true for all situations.

As mentioned before, the geometry-based approach requires a robust parametric CAD model which can be very difficult to make, if not impossible. There are several reasons for this.

1. Usually in the aerospace industry a surface-based modeler will be used. This increases the difficulty of making a parametric geometry model.
2. The CAD model is usually made to include all details. For a meaningful mesh to be created, these models need to be de-featured, otherwise the computational expense for analysis will be unaffordable. It is questionable whether the parametric CAD model can still exist after the de-featuring.
3. Some CAD software today allows the meshing to be done within the CAD software. However rarely can all the necessary boundary conditions and loads be complete applied in the CAD module. More frequently the geometry will be exported into the FEA package, modified, and meshed. Then the boundary conditions and loads will be applied and the analysis performed. Or the user may create the mesh with the CAD software and export it into the FEA package. Then the loads and boundary conditions are applied and the problem solved. Either way means interruption in the process.
4. A parametric CAD model requires the modeling parameters be defined when building the model. However these controlling parameters and geometry feature may no longer be suitable after entering the detailed design.

It is obvious that, unless the CAD package can have the same analysis strength as the FEA packages, or the geometric modeler inside the FEA packages is as powerful as the sophisticated CAD, hoping for a robust and easy integration between these two modules is not realistic at this time.

There are other reasons not to favor the geometry approach too. Figure 1 shows the traditional concept of a closed-looped optimization, where the parametric CAD model is closed inside the optimization loop. Figure 2, however, shows a slightly different version of an optimization loop, where there is an iterative loop inside the loop of the parametric CAD model. The advantage of the later methodology, is that right before and after the modification of the CAD model, engineering judgement is allowed. Also, if the scope of the optimization loop is reduced, effort on maintenance of the code is greatly reduced, and so is the risk of unexpected termination. Coding the mesh perturbation algorithms inside the FE package also makes it easier to access the analysis response database. This approach takes advantages of a good parametric CAD model and also reduces the restriction of the CAD modeling as well.

For the reasons stated above, the geometry-based approach was not considered during this research.

2.2 Objectives of the New Algorithm

Table 1 shows the comparison of the techniques indicated earlier between non-CAD based algorithms for structural shaping optimization. A few interesting facts can be observed. First, while the user usually expects less input required for the optimization task, it is obviously conflicting with the demand for flexibility on design shapes. However, since in practical design there is usually limitation on the changeable feature, eventually the flexibility of design shapes is pre-defined. Therefore a predefine flexibility of design shapes is actually acceptable.

Second, if a pre-defined flexibility is acceptable, then the Hybrid Natural Approach [23] seems to be the best choice. However, getting detailed information from a sophisticated CAD system is usually difficult, unless the CAD system itself supports this feature. Also, the analysts usually need several iterations to build a satisfactory mesh which compromises both accuracy and computer power. When the analysis is sophisticated, the association between the parametric CAD model and the mesh is likely to be destroyed, since it will become more and more frustrating for the analysts to compromise between meshing and modeling capability of the package. From the author's experience, there is usually only finite element mesh existing after some iterations. The original CAD model, even if carefully built, does not play a significant role anymore.

The objective here is to overcome this obstacle but still take advantages of the Hybrid Natural Approach. This is called the Contour Natural Function approach. The desired features are compared in Table 1. The objective of the current research can be summarized as follows, and the details will be explained in the following sections.

1. Reserve the flexibility of the candidate shape in Natural Shape Function approach.
2. Reduce the number of user-input nodes.
3. Reduce the number of design variables compared with the original Natural Shape Function approach, therefore also to reduce the dimension of the design space.
4. Build parametric model without CAD data or information. Thus the design variables should be set up in a CAD-like fashion. It should be easily build on any existing finite element packages without accessing or interference with the internal database and code.
5. Allow different parametric model setting upon one preliminary design without excessive effort.

	Node-Based	Natural Shape Function	Domain Elements	Hybrid Natural Approach	This Paper
Controlling Nodes	All Nodes	Boundary	Boundary	Boundary	Boundary
Passive Nodes	NONE	NONE	All Nodes	NONE	NONE
Domain Division	NONE	NONE	YES	NONE	NONE
Design Variables Basis	NONE	NONE	YES	YES	YES
Continuity Across the Whole Structure	NO	YES	NO	YES	YES
Design Shape Flexibility	Infinite	Infinite	Pre-defined	Pre-defined	Pre-defined
CAD Information	NO	NO	NO	YES	NO

Table 1 Comparison of Modeling Methods of Interests

2.3 The Mesh Perturbation Algorithms

The basic concept of the Hybrid Natural Approach for controlling the design shape, is first to divide the boundaries of the model into several connected segments. Each segment is then controlled by one CAD geometry entity and a few key points, and the perturbation of all other nodes are calculated by proper interpolation. The Natural Shape Function approach was then invoked to calculate the perturbation on the whole model.

Now assuming the CAD geometry information does not exist, it is then natural to consider rebuilding some geometry information based on the defined boundary nodes. For example, in a 2D problem, the user may pick a segment of nodes on the boundary, and then hope to define variation of these nodes based on only, say, 3, controlling nodes, for a parametric quadratic perturbation. If all nodes perfectly lie on this line, proper parameters can be assigned to each node easily, and many formulations can be used for calculating perturbation on the passive nodes based on given perturbation on the controlling nodes.

However, calculation of the parameters for each node on the curve will encounter difficulty, if the nodes do not lie perfectly on the reconstructed quadratic line. This situation is illustrated on Figure 3.

The problem is then how to assign a parameter on each passive node assuming that the line (or surface) does not perfectly match the contour. For a 2D finite element model, assuming the parametric line is to be interpolated, for example, by the Lagrange Interpolation as

$$x = \sum_{j=1}^N N_j(\xi) \cdot x_j \quad (1)$$

$$y = \sum_{j=1}^N N_j(\xi) \cdot y_j \quad (2)$$

with (x_j, y_j) the coordinate of the j th controlling node, and $N_j(\xi)$ the corresponding shape function of parameter ξ . The problem of assigning the parameter ξ to a node with the coordinate (x_i, y_i) is then solved by a proprietary algorithm. The parametric value of this node is then assigned to the corresponding passive node.

2.4 Calculating the Perturbation on the Passive Nodes

Once the curve or surface parameters of the passive nodes are decided, the perturbation on each *Passive Node*, based on the perturbation on the *Controlling Nodes*, can be obtained from modifying (1) and (2) as

$$\Delta x = \sum_{j=1}^N N_j(\xi) \cdot \Delta x_j \quad (4)$$

$$\Delta y = \sum_{j=1}^N N_j(\xi) \cdot \Delta y_j \quad (5)$$

Where Δx and Δy are the coordinate perturbation on the calculated *Passive Nodes*, and Δx_i and Δy_i are the coordinate perturbation of the *Controlling Nodes* on the curve that the *Passive Node* lies on.

With the perturbation defined on the controlling nodes and the ones on the passive nodes available, the Hybrid Natural Approach can be readily used. That is, when the new contour of the structural shape is decided, the final modified structural shape can be calculate by a static structural analysis with specified displacement on all nodes of the contour.

2.5 Design Sensitivity Analysis

The analytical design sensitivity of the current method is essentially the same as the one explained in the Hybrid Natural Approach [23]. The details will not be explained here and the interested readers can refer to the previous publication. Currently only the numerical design sensitivity is used since the access to the finite element analysis code is not possible.

2.6 Advantages of the Contour Natural Shape Function Approach

The advantages of this approach are

1. The flexibility of the candidate shape in Natural Shape Function approach is preserved.
2. The number of user-input nodes is greatly reduced.
3. The number of design variables can be greatly reduced compared with the original Natural Shape Function approach, therefore the dimension of the design space is also reduced.
4. This method allows easy combination with both the domain element approach and the natural shape function approach.
5. No CAD data is necessary. Thus the design variables are set up in a CAD-like fashion. It can also be easily built on any existing finite element package without accessing or interference with the internal database and code.
6. It is easier to define different parametric models upon the existing one preliminary design without much effort.

These characteristics of the approach are summarized in the first right column of Table 1.

3. Implementation

3.1 Finite Element Analysis Module

The commercial ANSYS package [28] was used for the FE meshing, modeling and analysis module. The algorithm was first developed based on the available functionality and features in ANSYS. However, the selection of the package becomes more and more reasonable later. The advantage of using ANSYS as a basic FEA package for structural optimization includes the following :

1. It integrates pre-processing, post-processing and analysis solution in one package seamlessly.
2. It is a multi-physics analysis package. Coupled analysis is possible.
3. It provides named parameters for conveniently connecting variables and response between the optimizer and itself.
4. It provides very powerful macro language. It is enough for accessing almost any database and response in the package. This language also makes it possible to define flexible constraints and the objective function, when connected with the named parameters in the package.
5. It provides interface to call the external programs in the macro language. This makes the interaction between the optimizer and the analysis easier.

The conclusion is the flexibility of this package makes it reasonable for structural optimization purpose. The definition of the constraints and the objective function can be almost infinite, since the macro language itself allows very complicated mathematical expression like the FORTRAN programming language. Graphic User Interface can also be easily implemented for special requirement.

Unfortunately, the package itself does not provide efficient design sensitivity information, therefore numerical design sensitivity is necessary. However, this is almost true in all commercial packages. For those with analytical design sensitivity available, the availability is usually pre-defined.

3.2 The Contour Natural Shape Function

The above-mentioned characteristics of ANSYS allows the original setting of the FE model to be repeatedly used, as long as the user is familiar with the flow and the macro language of the package. All existing macros and programs can still be used without much modification.

The Contour Natural Shape Function approach is also implemented in an external program, and ANSYS APDL is used as an interface to communicate with this program. Through this approach, it can be used in the batch mode execution of ANSYS in a parametric fashion.

3.3 The Structural Optimization Process

The overall structural optimization process is executed in the following architecture.

1. The user creates an input file for batch mode execution in ANSYS. The design variables, objective functions and constraints then can be linked with named parameters in ANSYS.
2. The optimization program reads in the input file and detects the relationship between the design variables, objective function, constraint values and the named parameters inside ANSYS. The communication protocol is then established in this step.
3. When the program proposed a new design, the value of the associated named parameters in the input was changed.
4. The optimizer then feeds the input file into ANSYS, and the desired response is saved in parameters and written to a file. The optimizer gets the response by reading in this file.
5. The optimizer calculates sensitivity of the new design, and repeats step 2 to 4, until a convergence is reached.

Since the contour natural shape function is implemented in macro form of ANSYS, it only appears as a regular command in the input file. Therefore the linking between the design variables and the perturbation of the controlling nodes is easy. This is also true for all other existing macros.

The similar approach is actually used in many general-purpose commercial optimization packages. The advantage is the flexibility. The disadvantage, however, is the overhead time for communicating between different programs through external files. This can be even more significant when the operating system is not every efficient in handling disk input/output.

The optimization process can be easily implemented and expanded since the programs do not require physical connection and linking. Most of the programming was actually done for file IO and data structure definition. The programming interface can be easily re-used since it took advantage of the macro

languages. Multiple optimization algorithms can also be easily implemented. Currently the Feasible Direction Method [29] is used for the optimization routine.

Finally, all programs were written in ANSI FORTAN 77 with minor extension from FORTAN 90. The program was compiled by HP Fortran 90 operating on a Unix system, and can be ported to other platforms easily. The architecture of this program allows all existing finite element analysis resource to be used as usual.

4. Numerical Examples

4.1 Square Plate with A Hole in the Center

This example was commonly used to test the shaping optimization algorithms. It is a square plate subjected to uniform and equal distributed load on all edges. Figure 4 shows the one quarter symmetric finite element model and the dimension with loads and boundary conditions.

The thickness of the plate is 0.0305 inches. The distributed load $P=10\text{lb/in.}$, $E=30\times 10^6$ psi, $\nu=0.3$, density $\rho=0.734\times 10^{-3}$ lbm/in³. Limit on the Von Mises stress is 1000 psi. The objective of this problem is to minimize the weight of the plate without violating the stress limitation. The changeable contour is defined as straight lines AB and FG, and quadratic lines BCD and DEF. There are totally 8 design variables. They are the x and y coordinates of C, D and E, the y coordinate of B and the x coordinate of F. (The number of design variables can be further reduced to four if symmetric linking is implemented).

Figure 5 shows the final optimal design. The problem took 14 design cycles to converge. Numerical gradients were used and a total of 119 function evaluations were taken (which includes 31 function evaluations and 11 numerical gradient evaluations). The final maximum Von Mises stress was 1000 psi. The final shape of the hole is a circle, which is the same as the analytical optimal solution. The weight was reduced by about 22%, and the radius of the circle is 3.82 inches, which is almost exactly the analytical optimum. This problem took 4 minutes and 21 seconds on a HP C3000 Unix workstation with 2GB RAM. This example in the previous publication [22] uses 12 design variables. It took 20 design cycles, 64 function evaluations, and 19 analytical gradients in the previous publication to achieve the same result.

4.2 Turbine Disk Shape Optimization

Figure 6 shows the finite element mesh of the preliminary design of a turbine disk in a gas turbine engine. The model is a two-dimensional axial-symmetric cross-section, with X representing the rotational axis. The disk is subjected to the centrifugal load. The initial design did not satisfy the stress constraints and reduction of the weight was required. The objective is to minimize the weight by changing the shape and to reduce the stress under allowable limits. Figure 6 also shows the controlling key nodes 1 through 14. The x and y coordinates of nodes 2 to 5 and 8 to 14 are defined as the design variables.

Figure 7 shows the final design after the optimization process. The weight is reduced by 22 percent of the original weight, and the maximum stress was reduced to 5 percent below the allowable value. This problem took about 5 minutes on a HP C3000 Unix workstation with 2GB RAM.

5. Conclusion

An algorithm for mathematically modeling and optimizing structural shape changes was developed. Special attention was paid to reduce the user input effort but still maintain reasonable flexibility. The examples show that the proposed algorithm is able to achieve the same result with less design variables in some cases, yet reduce the size of the design space. The application of the natural shape function approach provides less element distortion during the design process as demonstrated in the numerical example 1. The modular development of the methodology also provides flexibility to combine with the existing finite element analysis package even when the access to the internal database is limited. Using the proposed approach, a CAD-like parametric model for design optimization is possible even though a CAD parametric model is not available. Therefore this approach can be applied into the existing industrial design process without re-inventing the whole analysis and design system.

6. References

1. S.D. Rajan, Sizing, Shaping and Topology Design Optimization of Trusses Using Genetic Algorithm. ASCE J. Struct. Eng. 121 (1995), 1480-1487.
2. S-Y. Chen and S.D. Rajan, S.D., A Robust Genetic Algorithm For Structural Optimization, Struct. Eng. & Mech. J. 10 (2000), 313-336.
3. S-Y. Chen, An Approach For Impact Structure Optimization Using The Robust Genetic Algorithm, Finite Elements in Analysis and Design 37 (2001), 431-446.
4. O.C. Zienkiewicz and J.C. Campbell, Shaping Optimization And Sequential Linear Programming, in : Optimum structural design, edited by Gallagher (1973) , 109-126.
5. C.V. Ramakrishnan, A. Francavilla, Structural Shape Optimization Using Penalty Functions, J. Struct. Mech. 3 (1975), 403-422.
6. E.J. Huang, K.K. Choi, J.W. Hou and Y.M. Yoo, Shape Optimal Design Of Elastic Structures, in : Proc. Int. Symp. Optim. Struct. Design (1981).
7. L. Holzleitner, K.G. Mahmoud, Structural Shape Optimization Using MSC/NASTRAN And Sequential Quadratic Programming, Comp. & Struct. 70 (1999), 487-514.
8. K.H. Chang and K.K. Choi, A Geometry-Based Parameterization Method For Shape Design Of Elastic Solid, Mech. Struct. Mach. 20 (1992), 257-264.
9. D.A. Tortorelli, A Geometric Representation Schemes Suitable For Shape Optimization, Mech. Struct. Mach., 21 (1993), 95-121.
10. S. Chen, D.A. Tortorelli, Three-Dimensional Shape Optimization With Variational Geometry, Struct. Optim., 13 (1997), 81-94.
11. E. Hardee, K.-H. Chang, J.Tu, K. K.Choi, I. Grindeanu and X. Yu, A Cad-Based Design Parameterization For Shape Optimization Of Elastic Solids, Advances in Eng. Software 301 (1999), 185-199.
12. W. Annicchiarico, M. Cerrolaza, Finite Elements, Genetic Algorithms And β -Splines : A Combined Technique For Shape Optimization, Finite Element in Analysis and Design 33 (1999), 125-141.
13. H.G. Seong, K.K. Choi, Boundary Layer Approach To Shape Design Sensitivity Analysis, Mech. of Struct. Mach. 15 (1987), 241-263.
14. M.E. Botkin, Three-Dimensional Shape Optimization Using Fully Automatic Mesh Generation, AIAA J. 30 (1992), 1932-1934.
15. X. Song, J.D. Baldwin, A Novel Node-Based Structural Shape Optimization Algorithm, Comp. & Struct. 70 (1999), 569-581.
16. Y-T. Hwang, C. Fleury, A Mesh-Shape Sensitivity Synthesis For Boundary-Formulated Shape Optimal Design, AIAA 93-1532-CP (1993).
17. S. Kodiyalam, K. Virendra, P.M. Finnigan, Constructive Solid Geometry Approach To Three-Dimensional Structural Shape Optimization, AIAA J., 30 (1992), 1408-1415.

18. T. Lindby, J.L.T. Santos, Shape Design Sensitivity Analysis And Optimization With An Existing Associative CAD System, Proc. 5th AIAA/NASA/USAF/ISSMO Symp. Mult. Struct. Optim. (1994), 1483-1490.
19. N.V. Banichuk, F.-J. Barthold, A. Falk, E. Stein, Mesh Refinement for Shape Optimization, Struct. Optim. 9 (1995) , 46-51.
20. MacNeal-Schwendler Corporation, MSC/NASTRAN Version 69 Encyclopedia, (Los Angeles, CA, USA, 1996).
21. Vapderplaats Research & Development, Inc., GENESIS Version 6.0, User Manual, (Colorado Springs, CO, USA 2000).
22. A.D. Belegundu, S.D. Rajan, A Shape Optimization Approach Based On Natural Design Variables And Shape Functions, Comp. Meth. Appl. Mech. Eng. 66 (1988), 87-106.
23. S.D. Rajan, S.-W. Chin, L. Gani, Toward a Practical Design Optimization Tool, Micro. Civil Eng. 11 (1996), 259-274.
24. K.K. Choi, T.M. Yao, 3D Modeling And Automatic Regridding In Shape Design Sensitivity Analysis, Presented at the Symposium on Sensitivity Analysis in Engineering, (NASA Langley Research Center Hampton, VA 1986).
25. E.J. Nielsn, W.K. Anderson, : Aerodynamic Design Optimization Using The Navier-Stokes Equations, AIAA J. 37 (1999), 1411-1419.
26. S. Kodiyalam, G. N. Vanderplaats, H. Miura, Structural Shape Optimization With MSC/NASTRAN, Comp. & Struct. 40 (1991), 831-829.
27. M.E. Botkin, Shape Optimization Of Plate And Shell Structures, AIAA J. 20 (1982), 268-273.
28. ANSYS Inc., ANSYS Basic Analysis Procedures Guide, Release 6.0, (Canonsburg, PA, USA 2001).
29. G.N. Vanderplaats, CONMIN User's Manual, (NASA Technical Memorandum X-62282, Moffett Field, USA 1978).

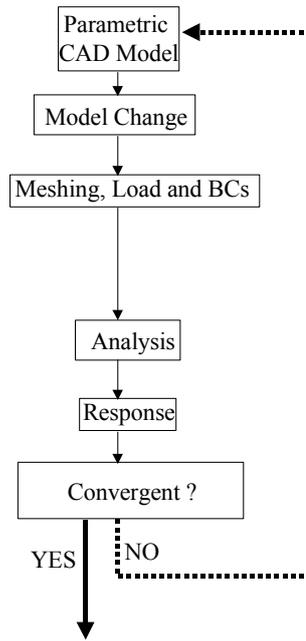


Figure 1 Traditional Design Loop with Parametric CAD Model.

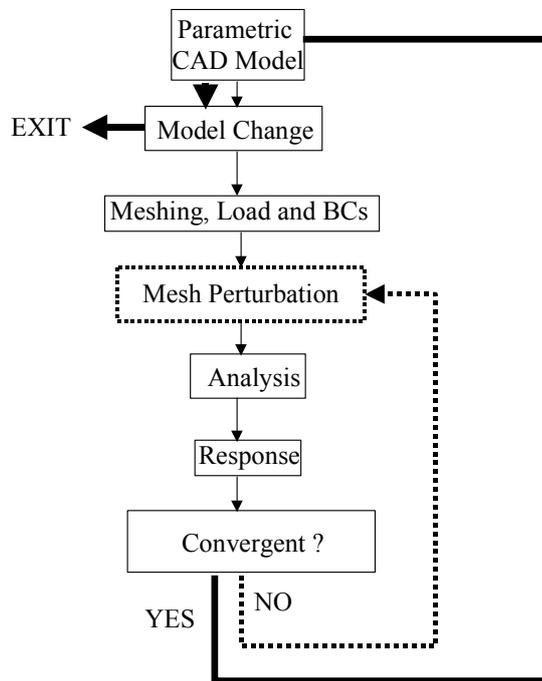


Figure 2 Design Loop with Parametric CAD Model and Mesh Perturbation

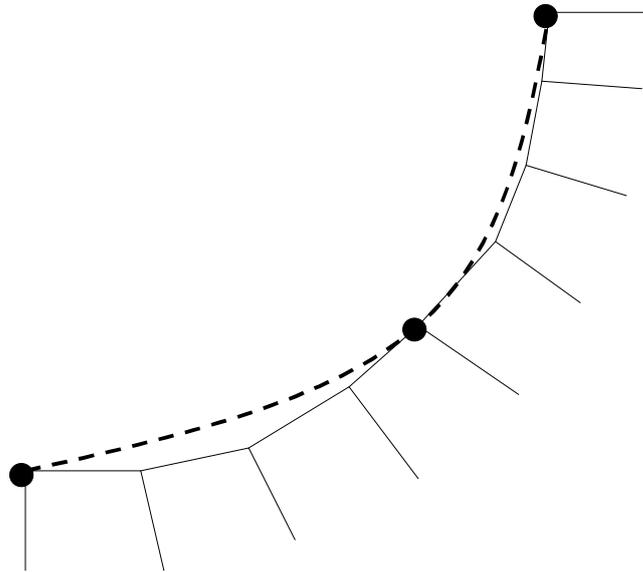


Figure 3 The Re-Built Geometry Curve (the Dash Line) and the Actually Mesh Contour.

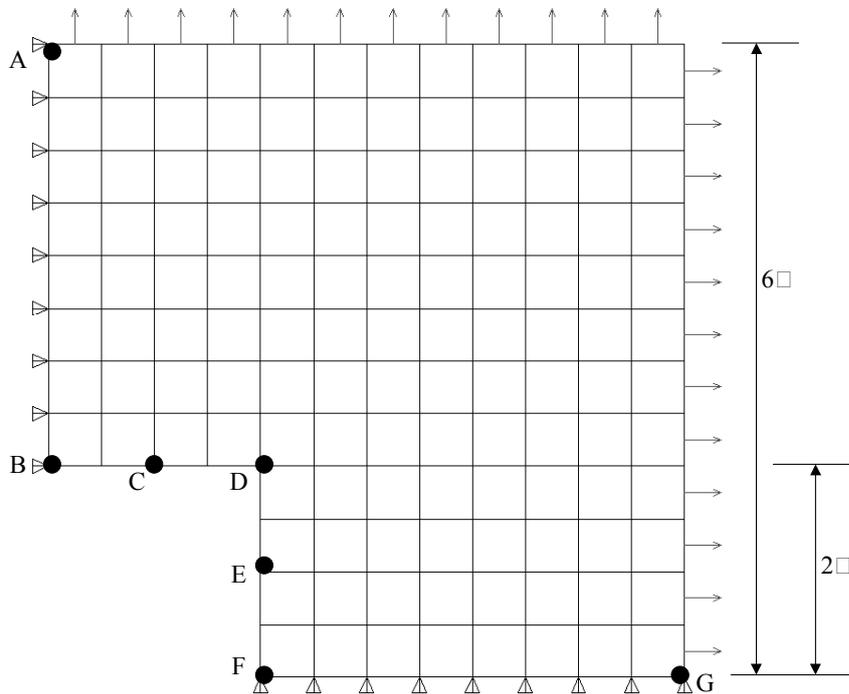


Figure 4 Geometry, Finite Element Mesh, Loading and Boundary Conditions for Example 2

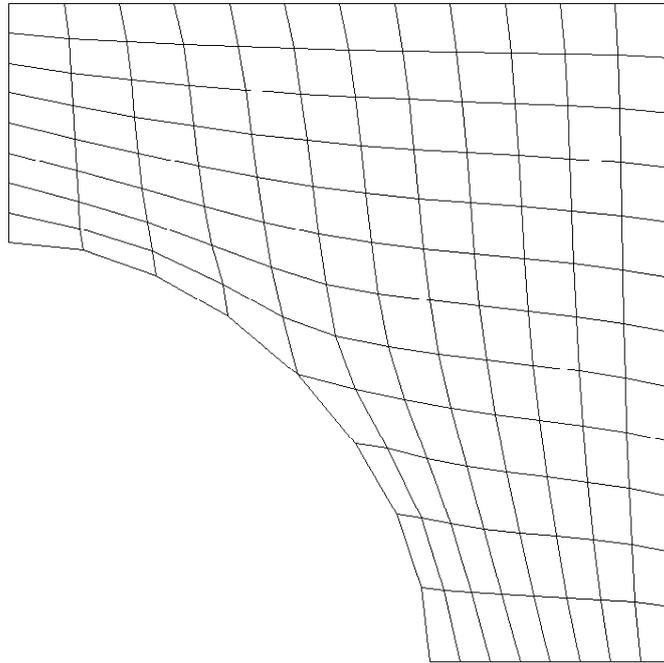


Figure 5 Final Optimum Design of Example 2

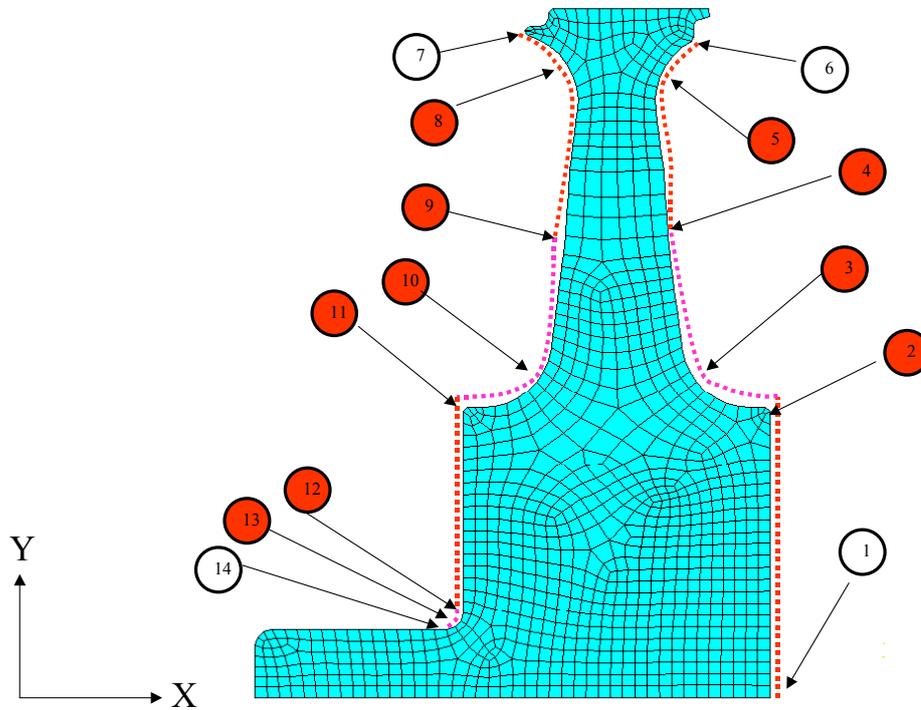


Figure 6 Initial Design and Controlling Nodes of the Turbine Disk

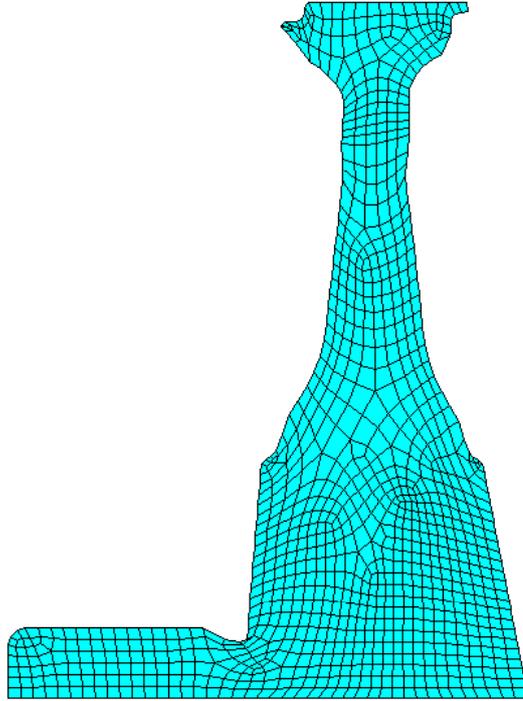


Figure 7 Final Optimum Design of the Turbine Disk