Topological optimization of the mobile element structure of Gantry type CNC machine using FEA method

L. Petrea and G. Stan
"Vasile Alecsandri" University of Bacau, Department of Industrial Engineering, Calea Marasesti Street, No. 157, 600115, Bacau, Romania
E-mail: septh4000@gmail.com

Abstract. Because of the rapid development key industries like aerospace, energy, rail transportation, engineering machinery and other industries there exist a growing demand for high speed and precise manufacturing of large and heavy parts. CNC Machine Tools with kinematical feed chains with great distance between slides, Gantry type, are the main tool to machine large and heavy parts. In such type of machines, the mobile element (the mechanical coupling between the two kinematical feed chains) has a significant effect upon the static and dynamic characteristics of the machine. Deformation of the mobile element has a significant impact upon machining accuracy. A study of the static proprieties and the deformation that effect the mobile element is extremely useful for improving the machining accuracy and efficiency of the machine. The weight of the mobile element it is extremely important and effects the overall dimension and weight of the machine. Using FEA analysis on a 3D CAD model this papers objectives are mesh optimization and topological optimization of the mobile element. This study will show that the topological optimization preformed on the mobile element has reduce the inertial forces, 10 to 30 %, while maintaining rigidity.

1. Introduction
Due to the growing demand for high speed and precise manufacturing of large and heavy parts, there is a demand for Gantry type CNC machine tools. In such types of machines, the mobile element (mechanical coupling between the two kinematical feed chains) has a significant effect on the static and dynamic characteristics of the machine. Deformation of the moving element has a significant impact on machining accuracy. A study of static properties and deformation affecting the mobile element is extremely useful for improving the machining accuracy and efficiency of the machine. The weight of the mobile element is very important and affects the overall size and weight of the machine. Thus a structural optimization of the mobile element it’s required. In engineering, optimizing an objective function is essentially maximizing or minimizing a constrained problem. Considering the type of design variables and the degree of difficulty, structural optimization can be divided into three types: size optimization, shape optimization and topology optimization corresponding to different stages of product design, namely the three stages of detailed design, basic design and conceptual design. Compared to the first two types of optimization, topological optimization can fundamentally change the structure of the design, the result is the basis for further optimization of the design, and as such it has great significance for the design. Although topology optimization has good prospects, it is recognized as the most difficult task in the field of structural design. Specifically, topology optimization finds optimal models by determining the best locations and geometry of cavities in the
design areas Huang and Xie, [2010] [1]. Topology optimization is performed using the results of a finite element analysis (FEA). In this study, we will use the topology optimization module in ANSYS® to optimize the mobile element.

2. Generalities about topology optimization

2.1. Short history

The development of topological optimization can be attributed Bendsøe and Kikuchi [1988] which proposed a method of homogenization [2], which achieves continuous topological optimization of the structure by dividing the number of structural units of different micro-holes in the design area. A web-based interface for a topology optimization program was presented by Tcherniak and Sigmund [2001] [3]. The study discusses implementation issues and educational issues, as well as statistics and experience with the program. Rahmatalla and Swan [2004] [4] presented a variable node-based design implementation for optimizing continuous structural topology in a finite element framework and explored its properties in the context of solving a number of different design examples. Dadalau et al. [2008] [5] introduced a new penalty system for the SIMP method. An advantage of the present method is the linear density-stiffness relationship which has an advantage for its own weight or Eigen frequency problem. The problem of topology optimization is solved by the method derived from the Optimality Criteria (OC), which is also the method used in this paper.

2.2. Topology optimization

A topology optimization problem aims to minimize compliance, while meeting various constraints, such as a given amount of material, weight, manufacturing requirements, costs, etc., Huang and Xie, [2010] [1]. The most comprehensive goal in topological optimization is to minimize compliance, equation (1).

\[ \min_{\xi} \; c(\rho) = U^T K U = \sum_{e=1}^{N} (\rho^e) u_e^T k^e u_e \]

\[ \text{where:} \quad \frac{v(\rho)}{V_0} = f \]

\[ : KU = F \]

\[ : 0 < \rho_{min} \leq x \leq 1 \]

Where the stiffness matrix for each element can be found in equation (2).

\[ k^e = \int_{\Omega} H^T D H d\Omega \approx \sum_{i=1}^{n-1} \sum_{j=1}^{n-1} w_i w_j / J(\xi, \eta) H D H (\xi, \eta) \]

The Young module does not influence the optimal results of the topology optimization. The overall topology optimization solution is built to minimize compliance using a Lagrange multiplication method equation (3).

\[ L(\rho, \lambda) = f^T d(\rho) + \lambda^T (f - K(\rho)d(\rho)) \]

By assigning \( \lambda = d \) and \( k^e = \int_{\Omega} H^T (E_0 + (\rho^e) P E_1 D) H d\Omega \) derived from the Lagrangian equation in relation to the design variables can be determined as equation (4), Bendsøe și colab., 2004 [6].

\[ \frac{\partial L(\rho, \lambda)}{\partial \rho^e} = -P(\rho^e) P^{-1} d^T K_f^e d^e \]

Using the bi-section algorithm, the densities of the elements in each iteration can be updated by a heuristic scheme. Due to the numerical stability and efficiency, a heuristic algorithm based on the Optimality Criteria (OC) method is proposed.
2.3. Optimality Criteria method

The discrete topology optimization problem is characterized by a large number of design variables, \( D \) in this case. Therefore, it is common to use iterative optimization techniques to solve this problem, e.g. Optimality Criteria method, at each iteration of the Optimality Criteria method, the design variables are updated using a heuristic scheme. The Lagrangian for the optimization problem is defined as [5]:

\[
L(x_i) = u^T Ku + \lambda \left( \sum_{i=1}^{n} x_i v_i - V_o \right) + \lambda_1 (Ku - F) + \sum_{i=1}^{n} \lambda_2^i (x_{min} - x_i) + \sum_{i=1}^{n} \lambda_3^i (x_i - 1)
\] (5)

Where, \( \lambda, \lambda_1, \lambda_2, \) and \( \lambda_3 \) are Lagrange multipliers for various constraints. The optimum condition is given by:

\[
\frac{\partial L}{\partial x_i} = 0, \quad i = 1, 2, 3, ..., n
\] (6)

That result in the Conformity:

\[
L = u^T Ku
\] (7)

By differentiating equation (5) w. r. t., the optimality condition can be written as:

\[
B_i = \frac{\partial c}{\partial x_i} = 1
\] (8)

Conformity sensitivity can be assessed by using equation:

\[
\frac{\partial c}{\partial x_i} = -p(x_i)^{n-1}u^TK_iU_i
\] (9)

According to this equations, the design variables will be updated as follows:

\[
\text{new } x_i = \begin{cases} \max(x_{min} - m), & \text{if } x_i B_i^\eta \leq x_{min}, (x_{min} - m) \\ x_i B_i^\eta, & \text{if } \max(x_{min} - m) < x_i B_i^\eta < \min(1, x_i + m) \\ \min(1, x_i + m), & \text{if } \min(1, x_i + m) \leq x_i B_i^\eta \end{cases}
\] (10)

In which, \( m \) is the the motion limit and represents the maximum permissible change in \( x_i \) in one Optimality Criteria iteration, \( \eta \) represents the numerical depreciation coefficient and is usually considered to be 1/2. The Lagrange volume constraint multiplier \( \Lambda \) is determined at the Optimality Criteria iteration using a bisection algorithm, \( x_i \) represents the value of the density variable at each of the iterations steps, \( u_i \) is determined from the equilibrium equations an it’s the displacement field at each iteration step. In short the structure of the optimization algorithm is:

- The initial design is performed.
- The resulting displacements and strains are calculated by the finite element method.
- The conformity of the project is calculated. If it’s just a marginal improvement in compliance with the last project, stop the iterations. Otherwise, continue.
- The update of the design variable is calculated, based on the scheme presented in equation (10). This step also consists of an internal loop iteration to find the value of the Lagrange multiplier that constrain the volume.
- Repeat the iteration loop.
3. Topological optimization of the mobile element

3.1. Geometric model structure

For this work, a 3D model was created for the mobile element, respecting the technical parameters of an SC14 carousel lathe (figure 1): length 2292mm, width 685mm, thickness 280mm and Gray cast iron material. In order to reduce the calculation time, the 3D model of the slide (the mobile element) has been simplified.

![Figure 1. SC 14 carousel lathe.](image1)

![Figure 2. Simplified mobile element.](image2)

3.2. Optimization and refinement of mesh in ANSYS

ANSYS can generate a generic mesh, see figure 2. To check the quality of the mesh we can use the “Element Quality” function from the “Mesh Metric” tab. As we can see in figure 3 many elements are of low quality (0 - 0.5 plot area in figure 3). This will affect the quality of static analysis and topological optimization. The problem can be solved by generating a new mesh. ANSYS has several tools for optimizing and refining the mesh, to obtain a good result the method "Tetrahedrons" with the parameter "Independent patch" was applied (figure 4). This method refines the interior mesh firstly then the exterior mesh achieving very good results.

![Figure 3. Element quality plot – Generic Mesh.](image3)

![Figure 4. Element quality plot – Tetrahedrons+ Patch independent.](image4)
As we can see most of the elements are grouped on the right side of the element quality plot, with most of them at the 0.90 mark, this represents a very good result.

3.3. Static analysis of the mobile element
According to the technical specification of the SC14 carousel lathe the mobile element can sustain a 51000 N load. Figure 5 shows the distribution of forces that act upon the mobile element during normal machining operations, The 20000 N force on Y axis represent the mass of the vertical slide and spindle.

As we can see in figure 6, the total deformation upon the mobile element its small, under 0.002mm, well within the machine tool precision requirements.

3.4. Topology optimization analysis and results
After preforming the static analysis, we can use ANSYS Workbench to add a topology optimization module to the static analysis results. As mentioned the optimization method will be "Optimality Criteria" (OC), the maximum number of iterations 30, the convergence accuracy 0.1% and the constraint function will be the mass min. 70%.

![Figure 5. Force loading.](image)

![Figure 6. Mobile element deformation.](image)

![Figure 7. Topology Density – Retained region.](image)

![Figure 8. Topology Density – Removed region.](image)
The results of the topology optimization are represented as Topology Density, we can observe in figure 7 the material that has been kept after 12 iterations. In figure 8 we can observe the material that has been removed. As we can observe the result it’s not a geometry that can be easily manufactured, however it dose tell the designer what material need to be removed from the mobile element, without sacrificing rigidity.

Figure 9 and figure 10 represent the objective function plotted course, we can observe that the compliance decreases with the increase of iterations, and that the iterative results are in stable condition after 10th step so that calculation results converge.

4. Conclusions
This study presents the mathematical model of a topological optimization of a structure (the mobile element / slide in this case) while using the ANSYS software platform for optimization calculation to achieve conceptual design that helps to maintain the conditions of rigidity while reducing weight by approximately 25%. Compared to traditional structural designs, this method not only guarantees the reliability of the design, but also shortens the design cycle and increases efficiency.

5. References
[1] Huang, X. and Xie, Y.M. 2010 Evolutionary topology optimization of continuum structures: methods and applications John Wiley & Sons.
[2] Bendsøe M.P. and Kikuchi N. 1988 Generating optimal topologies in structural design using a homogenization method Comput. Meth. Appl. Mech. Eng. 71 197-224.
[3] Tcherniak D. and Sigmund O. 2001 A web-based topology optimization program Struct. Multidisc. Optim. Springer-Verlag 22 179-187.
[4] Rahmatalla S.F. and Swan C.C. 2004 A Q4/Q4 continuum structural topology optimization implementation Struct. Multidisc. Optim. Springer-Verlag 27 130-135.
[5] Dadalau A. and Hafla A. 2008 A new adaptive penalization scheme for topology optimization Stuttgart Research Centre for Simulation Technology (SRC Sim. Tech.), Stuttgart University.Verl.
[6] Bendsøe, M.P. and Sigmund, O 2004 Topology optimization by distribution of isotropic material Topology Optimization, Springer 1-69.