Naman Jain¹

Topological optimization of the fork-end of a knuckle joint

¹ Department of Mechanical Engineering, G. B. Pant University of Agriculture and Technology, Pantnagar, India, E-mail: namanjainyati@gmail.com. <http://orcid.org/0000-0002-6251-2760>.

Abstract:

Topology optimization is a mathematical approach that optimizes the layout for the given design constraints such as loading and boundary conditions so that the optimum design obtained performs its function. In different types of loading conditions such as single load or multiple load topological optimization result in the best use of a material for a body in given volume constraints. In topological optimization the structural compliance is minimized while satisfying a constraint on the volume of the structure. This paper represents the topological optimization of the fork-end (double eye) of a knuckle joint with the objective to reduce the mass of an existing fork-end of a knuckle joint of an automobile or locomotive by applying the optimization technique. Reducing the weight of an automobile part will result in the overall weight reduction of a vehicle, thus, its energy consumption demands decrease thereby improving its fuel efficiency. The topological optimization was done using a finite element solver, ANSYS. The ANSYS Parametric Design Language was employed for utilizing the topological optimization capabilities of the commonly used finite element solver ANSYS. Solid92 elements were used to model and mesh the fork end of the knuckle joint in ANSYS. The optimality criteria method was used for topological optimizing the fork end of a knuckle joint.

Keywords: finite element method, knuckle joint, SIMP, topological optimization

DOI: 10.1515/jmbm-2018-0022

Introduction

Engineering design refers to converting ideas into reality, that is, obtaining the optimum solution to a specific problem so that demands of society are met. To achieve this, one should study and simulate the design of the components and optimize the material for cost effectiveness. Topological optimization is a process to optimize the material distribution in a given domain of a structure [1] for optimum use of a material along with minimizing displacement and the stresses that develop. In topological optimization function and variables are used to define the problem: the objective function (f): a function that contains the characteristics of the structure such as compliance or stiffness [2]. Design variable (x): this is a variable or vector that describes the distribution of mass through the given design domain. State variable (y): vary depending on the design variable, such as displacement, stress, etc. Here, the fork-end of the knuckle joint was analyzed through the finite element method. A knuckle joint is a mechanical joint used in machines to provide connectivity between two intersecting cylindrical rods, whose axes lie on the same plane. When loading is applied on the cylindrical rods, the load fork end and the eye end adjust their positions about the axis of the knuckle pin. The structure diagram of the fork end of knuckle joint with dimensions (all dimensions are in mm) is shown below in Figure 1, which was used in ANSYS modeling.

Naman Jain is the corresponding author.

©2018 Walter de Gruyter GmbH, Berlin/Boston.

Figure 1: Dimension of the fork end of a knuckle joint.

A general structural problem now takes the form:

$$\begin{cases} \text{minimize } f(x, y) \text{ with respect to } x \text{ and } y \\ \text{subjected to } \begin{cases} \text{behavioral constraints on } y \\ \text{design constraints on } x \\ \text{equilibrium constrain} \end{cases} \end{cases} \quad (1)$$

SIMP stands for the solid isotropic material with penalization method [3], [4]. In this method we define a design variable known as pseudo-density defined as:

$$x_i = \frac{\rho_i}{\rho_0} \quad 0 \leq x_i \leq 1 \quad (2)$$

The design domain first divided into finite part [5] and each part is represented by a pseudo-density the value of which lies between 0 and 1 [6], where 0 represents no material whereas 1 represents 100% volume. In each iteration the value of the pseudo-density variable is optimized. The penalization scheme or the power law approach was applied to the evolution 0–1 scaling. For isotropic material the value of penalization 3 was taken.

The problem in topological optimization is converted into:

$$\begin{cases} \text{Minimize: } C(X) = \{F\}^T \{U\} \\ \quad \quad \quad X = (x_1, x_2, \dots, x_n)^T \\ \text{Subject to: } \begin{cases} V^T x \leq V^* \\ 0 < x_{\min} \leq x_i \leq 1 \\ \{F\} = [K]\{U\} \end{cases} \end{cases} \quad (3)$$

Materials and methods

Finite element modeling

In the knuckle joint there are three basic components namely: eye-end, fork-end and the pin, the dimensions of the fork-end essentially depend upon the load applied in the joints. First, we have prepared the computer-assisted design (CAD) model of the fork-end of a knuckle joint using ANSYS software (Canonsburg, Pennsylvania) as shown in Figure 2A. While preparing the CAD model, we assigned the proper dimensions and the material properties of the abovementioned parts as per Figure 1. The material selected in the present investigation was structural steel. The model of the fork-end of the knuckle joint was meshed with SOLID95 element type as shown in Figure 2B and Table 1 represents the mesh results. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y and z directions. The element may have any spatial orientation. The element has plasticity, creep, stress stiffening, large deflection and large strain capabilities. The fork end was subjected to a horizontal force of P (50 N) to the left and both the inner circle of the fork end constrained with zero degree of freedom in all direction as shown in Figure 3. The sum of all the horizontal force acting on rod-A must be equal to zero. Therefore, there should be a force P to the right acting on the fork end. The force P is divided into two parts, each equal to (P/2) as reaction forces on the fork end.

Figure 2: (A) CAD model (B) mesh model in ANSYS.**Table 1:** Mesh results.

Number of elements	7721
Number of nodes	12595
Total volume	1.3395e-004 m ³
Total weight	1.0515 kg

Figure 3: Loading and boundary conditions for the fork-end of a knuckle joint.

The optimality criterion approach

To solve the optimization problem optimality criterion approach [7], [8], [9] was employed. In the optimality criteria method a Lagrangian function was created as per the design domain and the boundary condition was applied. Two types of problems exist in the topological optimization, either we minimize or maximize the objective function [10], [11], [12], subject to the constraints on the material resource. In the present investigation, compliance [1], [13], [14] is objective function and we are minimizing it. An iterative based optimization technique is applied to solve this problem. The Lagrangian function based on eq. 3 for the optimization problem is defined as:

$$\mathcal{E}(x_i) = \{U\}^T [K] \{U\} + \Lambda (\sum V^T x - V^*) + \lambda_1 ([K] \{U\} - \{F\}) + \sum \lambda_2 (x_{\min} - x_i) + \sum \lambda_3 (x_i - 1) \quad (4)$$

where Λ , λ_1 , λ_2 and λ_3 are Lagrange multipliers for the various constraints. The optimality condition is given by:

$$\frac{d\mathcal{E}}{dx_i} = 0 \quad (5)$$

Now, compliance,

$$C = \{U\}^T [K] \{U\} \quad (6)$$

Differentiating the above equation w. r. t. x_i , the optimality condition can be written as:

$$B_i = \frac{-\frac{dC}{dx_i}}{\Lambda V_i} = 1 \quad (7)$$

The compliance sensitivity can be evaluated as using the following equation:

$$\frac{dC}{dx_i} = -p(x_i)^{p-1} u_i^T k_i u_i \quad (8)$$

Based on these expressions, the design variables are updated and in each iteration the optimum solution is obtained.

Results

In this section the detailed results of finite element analysis and optimization of the fork-end of the knuckle joint have been presented. The static analysis of the fork-end of a knuckle joint structure was carried out for the load cases shown in Figure 3. The material selection depends upon the fact that it could withstand the stresses developed in the joint, and also could deform elastically during its operation. ANSYS software was used to measure the amount of deformation in the x-direction (U_x), the y-direction (U_y) and stress intensity as shown in Figure 4 A, B and C respectively. Maximum displacement in the x-direction is $26.072 \mu\text{m}$ where as in the y-direction it is $3.7032 \mu\text{m}$ with stress intensity of 101.19 N/mm^2 . The optimization of the fork-end of a knuckle joint was carried out using the topology optimization technique as shown in Figure 5. The optimization focused on the uncritical sections which need to be reduced. The main objective was to minimize the weight of the connecting rod as well as the total production cost. It can be seen that the optimized model reduced the weight from the initial design until the value converged as shown in Figure 6 and Table 2. The value of the convergence criteria was 0.0001 and volume fraction of 0.40 was used in the topological optimization of the fork-end of a knuckle joint and the compliance of the fork-end was reduced by 50.4%.

A Displacement in x-direction

B Displacement in y-direction

C Stress intensity

Figure 4: Displacement and stress distribution in the fork end of a knuckle joint.

A Topological optimization of fork-end

B Final shape obtain after material removal

Figure 5: Optimum topology of the fork end of a knuckle joint.

Figure 6: Graph for iteration vs. compliance.

Table 2: Result obtained for topological optimization.

Volume fraction	0.40
Initial compliance (Nmm)	1312.71e3
Final compliance (Nmm)	650.836e3
Percentage reduction (%)	50.4

Conclusion

The modeling of the fork-end of a knuckle joint and finite element analysis has been presented. Topology optimization was analyzed to the fork-end of the knuckle joint and according to the results, it can be concluded that the weight of optimized design was reduced and the maximum stress was also predicted to be lower than the initial design of the fork-end. The results clearly indicate that the new design is much lighter and has more strength than initial design of the fork-end of the knuckle joint. The material optimization approach should be considered for future research for shape optimization and sizing optimization. In shape optimization, the optimum design obtained after topological optimization goes through shape optimization where the shape of the design is determined as per manufacturing considerations.

References

- [1] Chiandussi G, Codegone M, Ferrero S. *Comput. Math. Appl.* 2009, 57, 772–788.
- [2] Swan CC, Kosaka I. *Int. J. Numer. Meth. In Eng.* 1997, 40, 3033–3057.
- [3] Bendsoe MP, Kikuchi N. *Comput. Meth. Appl. Mech. Eng.* 1988, 71, 197–224.
- [4] Thomsen J. *Struct. Multidisc. Optim.* 1992, 5, 108–115.
- [5] Sigmund O, Petersson J. *Struct. Optim.* 1998, 16, 68–75.
- [6] Diaz A, Sigmund O. *Struct. Optim.* 1992, 10, 40–45.
- [7] Tcherniak D, Sigmund O. *Struct. Multidisc. Optim.* 2001, 22, 179–187.
- [8] Sigmund O. *Struct. Multidisc. Optim.* 2001, 21, 120–127.
- [9] Yin L, Yang W. *Comput. Struct.* 2001, 79, 1839–1850.
- [10] Allaire G, Jouve F, Toader AM. *C. R. Math.* 2002, 334, 1125–1130.
- [11] Rahmatalla SF, Swan CC. *Struct. Multidisc. Optim.* 2004, 27, 130–135.

- [12] Rozvany GIN. *Struct. Multidisc. Optim.* 2009, 37, 217–237.
- [13] Hammer VB, Olhoff N. *Struct. Multidisc. Optim.* 2000, 19, 85–92.
- [14] Jain N, Saxena R. *Alexandria Eng. J.* 2017. DOI: <http://dx.doi.org/10.1016/j.aej.2017.01.006>.