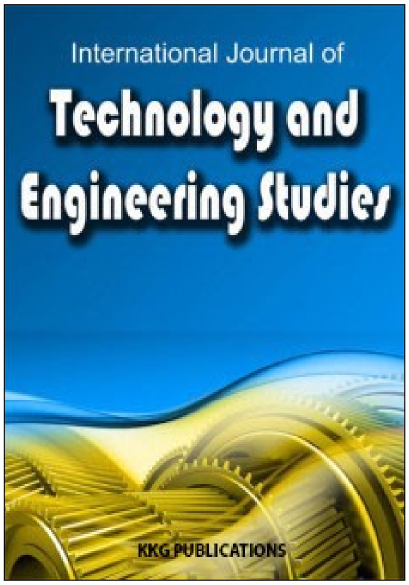

Key Knowledge Generation
Publication details, including instructions for author and subscription information:
http://kkgpublications.com/technology/

Size and Shape Optimization of Structures Using GA-FEA Interface

NAMAN SONI [1], PARTH DAVE [2], VIVEK PATEL [3], VIMAL SAVSANI [4]

[1, 2, 3, 4] Pandit Deendayal Petroleum University, India

Published online: 18 June 2016

To cite this article: N. Soni, P. Dave, V. Patel and V. Savsani, "Size and shape optimization of structures using GA-FEA interface," *International Journal of Technology and Engineering Studies,* vol. 2, no. 3, pp. 76-82, 2016.
DOI: https://dx.doi.org/10.20469/ijtes.2.40003-3

To link to this article: http://kkgpublications.com/wp-content/uploads/2016/2/Volume2/IJTES-40003-3.pdf

International Journal of Technology and Engineering Studies
vol, 2, no. 3, pp. 76-82, 2016

IJTES

SIZE AND SHAPE OPTIMIZATION OF STRUCTURES USING GA-FEA INTERFACE

NAMAN SONI [1] [*], PARTH DAVE [2], VIVEK PATEL [3], VIMAL SAVSANI [4]

[1, 2, 3, 4] Pandit Deendayal Petroleum University, India

Keywords:
Optimization
FEA
Genetic Algorithm
Structural Optimization

Abstract. This study demonstrates the successful application of integration of GA and Ansys for size and shape optimization of a truss structure. Optimizing any truss structure is an important research topic due to the complexity of problems and the benefits to the industry. Furthermore, it is an essential need for any industrial manufacturer to optimize their design with respect to conflicting technical and financial goals. Therefore, the design variables were optimized in order to achieve the minimum weight. The capability of the method was demonstrated by comparing the results with those obtained by HPSO, GSO, and TLBO. It is observed from the comparative result that the present approach is 25.88% lesser in weight than TLBO, 58.23% lesser in weight than HPSO, and 26.66% lesser in weight than GSO.

Received: 08 April 2016
Accepted: 22 May 2016
Published: 18 June 2016

INTRODUCTION

Structural design has always been a concern for engineers in practice. The focus is not only in construction cost, but also in geometry of structures. It is the responsibility of engineers to design structures with high reliability and low cost. The best way to achieve this is to use different optimization algorithm like Genetic Algorithm (GA), firefly algorithm, Teaching Learning Based Algorithm (TLBO), heat transfer search algorithm (HTS) etc.

Without optimization, any structure results in over size, over design and consequently more cost. To incorporate the economic aspect in any design, it is important to optimize the structure. The FEA is used so as to avoid the structure from failure.

Structural optimization problems are grouped into three categories: sizing, shape, and topology [1]. Sizing optimization is typically applied to a truss-type structure to obtain the optimal cross-sectional areas of beams. Shape optimization determines the optimal boundaries of a structure. Topology optimization is able to offer optimum topology along with shape and size. Size optimization is typically applied to a truss-type structure to obtain the optimal cross- section areas of beams. As the area increases, the induced stress in the structure decreases resulting in increased weight. Sizing design variables can be plate thickness and beam cross- sectional areas. Hence, we set these design variables in such a way that we get the minimum possible area for minimum weight. The minimum weight will, however, be constrained by the maximum stress that the material can withstand. Shape optimization determines the optimal boundaries of a structure for the given fixed topology. Design variables are typically spline control points defining the shape of a structure in 2D or 3D. The total number of members in the optimized structure remains the same as the original structure.

There are three popular methods to undertake size and shape optimization [2].

1) Both structural analysis and optimization done using a commercial FEA code (Ansys classic environment);

2) structural analysis done in commercial FEA code but an external optimization code is used (MMA optimizer); and

3) both structural analysis and optimization using MATLAB programming capacities. They have discussed the advantages, disadvantages and the computer running time for the methods by using various examples.

Researchers have adapted various optimization methods on many practical applications like trusses, beams, vehicle structure, etc. [3] have performed size optimization on a tapered cantilever beam. Their governing idea was to minimize the mass of the beam for a given strength. [4] implemented GA optimization by coupling MATLAB with Ansys for optimizing a tapered cantilever beam with an objective to minimize its volume along

[*] Corresponding author: Naman Soni
[†] Email: namansoni36@gmail.com

with maintaining the required strength. [5] performed Size and Shape optimization on a 40-bar truss structure. The method used by them was teaching learning based optimization. The minimized weight obtained was compared with that obtained by other algorithms such as Heuristic Particle Swarm Optimization (HPSO) and group search optimizer (GSO).

The method of using the combination of Ansys and MATLAB has been very popular in recent time. The main reason for its popularity is that the researchers could have a greater control on the different parameters present in the design space and also in return get a better accuracy compared to using only MATLAB for finite element method (FEM) routine. [6] have implemented GA in MATLAB along with Ansys to optimize the structure of a vehicle. They were able to achieve a 4% reduction in weight thereby depicting that the method of integration of both MATLAB and Ansys helps the user to achieve the advantages of both the software with the help of some programming effort. The process flow of this method is discussed in this paper later.

The methodology of carrying out the optimization of trusses using Ansys and MATLAB has successfully been demonstrated by [7]. They have outlined the important constraints required to be applied to the trusses. They have validated their hypothesis by applying it on various benchmark examples and have compared the results with other algorithms.

Interfacing Genetic Algorithm and FEA

This section deals with the method of integration of Genetic algorithm tool box of MATLAB and Ansys. Before looking at the integration of both the software we would be discussing about the individual capacities of both MATLAB and Ansys.

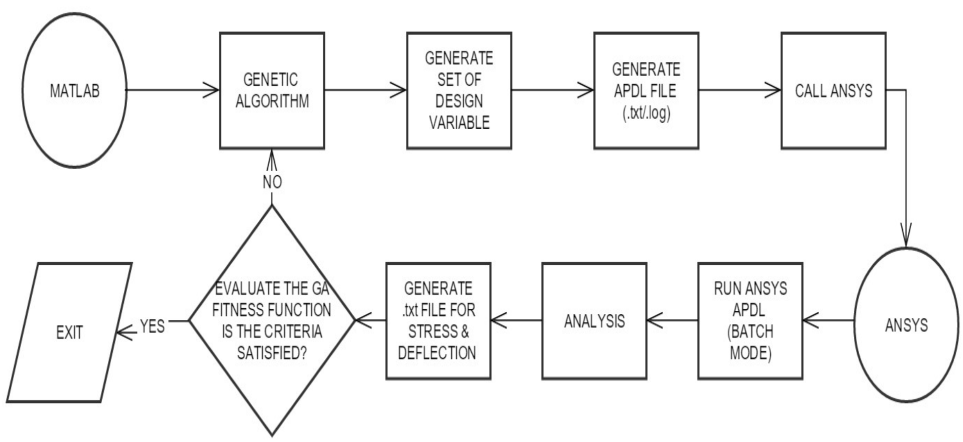

Fig. 1 . Process flow of the optimization loop

Finite Element Analysis (FEA)

Ansys is a commercial finite element software which can solve a variety of problems like structural, thermal, fluid, etc. With the increasing demand from the industry to optimize the designs of their product to satisfy the opposite goals of least cost and strength, Ansys Inc. has released two tools to help them. These are ANSYS Probabilistic Design and the ANSYS DesignXplorer. ANSYS Probabilistic Design is inbuilt into Ansys Mechanical APDL and the ANSYS DesignXplorer is inbuilt into the Ansys Workbench [12]. [8] have reported the advantages and the disadvantages of both the probabilistic design tools.

From past experience and results, for a problem of a few design variables, a commercial finite element method like Ansys is the right choice to optimize the design [13-15]. But if there are large number of design variables then it is experienced that Ansys can spend a huge time in order to reach an optimum point.

Genetic Algorithm

Genetic Algorithm helps in efficiently searching a large space of possible solutions to a problem for an optimal solution, e.g., identifying an optimal order for a number of variables or finding an optimal set of weights and parameters for an experiment [9]. It is based on the model of biological evolution based on the Charles Darwin's theory of natural selection. There are many advantages of genetic algorithms over traditional optimization algorithms, and two most noticeable advantages are: the ability of dealing with complex optimization problems and parallelism. Genetic algorithms can deal with various types of optimization whether the objective (fitness) function is stationary or non-stationary (change with time), linear or nonlinear, continuous or discontinuous, or with random noise [10].

MATLAB is a powerful mathematical computation tool which has inbuilt optimization toolbox. This toolbox has many optimization algorithms like GA, Linear Programming, Quadratic Programming, etc. Researchers have used only MATLAB for carrying out the full cycle of optimization, this is done by programming the FEM codes into MATLAB. The programming is difficult and time consuming so methods of integrating GA and Ansys are developed which incorporate the advantages of both the software.

Integrating GA and FEA

By integrating MATLAB (i.e. GA toolbox of MATLAB) and Ansys, the designer is able to get the best features of both the software. MATLAB is a powerful mathematical computational tool which has many optimization algorithms built in. Ansys is a commercial finite element software which will carry out the structural analysis of the truss. It is worthwhile to note that any other finite element software can be used with MATLAB provided it can accept a txt/log file as input and can run in batch mode.

The problem is defined in MATLAB. The objective function and the constraints are coded in MATLAB. First the genetic algorithm generates a set of variables which generates a truss structure. This truss structure is sent to Ansys for solving it by Finite Element Method (FEM). The maximum stress and deflection are compared with the maximum limit depicted in next section. If the truss exceeds the specified limit, then a penalty is applied to the objective function otherwise the truss is deemed acceptable. This cycle is repeated as per the variables configured in the genetic algorithm till it reaches an exit condition.

Problem Formulation

In the present work, we have taken 40-bar planar truss problem as shown in Figure 2 to demonstrate size and shape optimization method. This problem is adapted from [5]. Their work is on size and shape optimization of truss structure using TLBO.

The problem is solved by integrating MATLAB and Ansys. There are 40 members, which fall into 20 groups on the basis of geometric similarity of the structure, as follows: (A_1) 1, 7; (A_2) 2, 6; (A_3) 3, 5; (A_4) 4; (A_5) 8, 14; (A_6) 9, 13; (A_7) 10, 12; (A_8) 8; (A_9) 15, 22; (A_{10}) 16, 21; (A_{11}) 17, 20; (A_{12}) 18, 19; (A_{13}) 23, 29; (A_{14}) 24, 28; (A_{15}) 25, 27; (A_{16}) 26, 33; (A_{17}) 30, 36; (A_{18}) 31, 35; (A_{19}) 32, 34 ; (A_{20}) 37, 38, 39, 40. Due to the geometrical similarity of the structure the top nodes are constraints as follows: $1 \leq y_9 = y_{16} \leq 5$, $1 \leq y_{10} = y_{15} \leq 5$, $1 \leq y_{11} = y_{14} \leq 5$, $1 \leq y_{12} = y_{13} \leq 5$ (m) where, y is the vertical coordinate of the nodes. The nodes 2, 3, 4, 5, 6 and 7 are acted by Force P, 98066.50 N in negative y direction. The design parameters are listed in Table 1.

Objective Function

In the present work, we have considered the overall weight as the objective function. Mathematically the objective function is described as under considering the density (ρ), length of the bar (l) and area of the bar (A).

Minimize,

$$\sum_{i=1}^{36} (\rho^* l_i^* A_i) + penalty$$

Subject to,

C1 = $S_i - Stress\_l \leq 0$ (i = 1, 2, 3, ... 36)

C2 = $def_i - def\_l \leq 0$ (j = 4, 5)

C3 = $A_{min} \leq A_i \leq A_{max}$ (i = 1, 2, 3, 36)

Where,

Stress_l = Maximum stress that the material can sustain

def_l = Maximum deflection of the nodes

Penalty = Penalty applied in case of constraint violation

TABLE 1
DESIGN PARAMETERS AND SEARCH RANGE

Parameter	Value/Range
Density	7850 kg/m^3
Youngs Modulus	196.13 GPa
Limiting Stress	$\pm$ 156.91 MPa
Limiting deflection at node 4 and 5	$\pm$ 0.035m
A_{min}	0.001 m^2
A_{max}	0.005 m^2

Constraints

Constraint 1. This constraint deals with the limiting stress for each member of the truss. If the stress in a member is greater than limiting stress than a penalty of 5000 is added to the objective function. In case of no violation to this constraint no penalty is applied.

Constraint 2. Node 4 or 5 should not deflect more than 0.0035m. If the deflection is more than 0.0035m, then a penalty of 5000 is applied to the objective function.

Constraint 3. GA variables are real valued coded therefore the upper and lower bounds are automatically taken care of.

The objective function, weight of structure, is minimized under the above mentioned constraints by integration of Ansys and GA toolbox of MATLAB. The next section describes the results-discussion obtained using the proposed methodology.

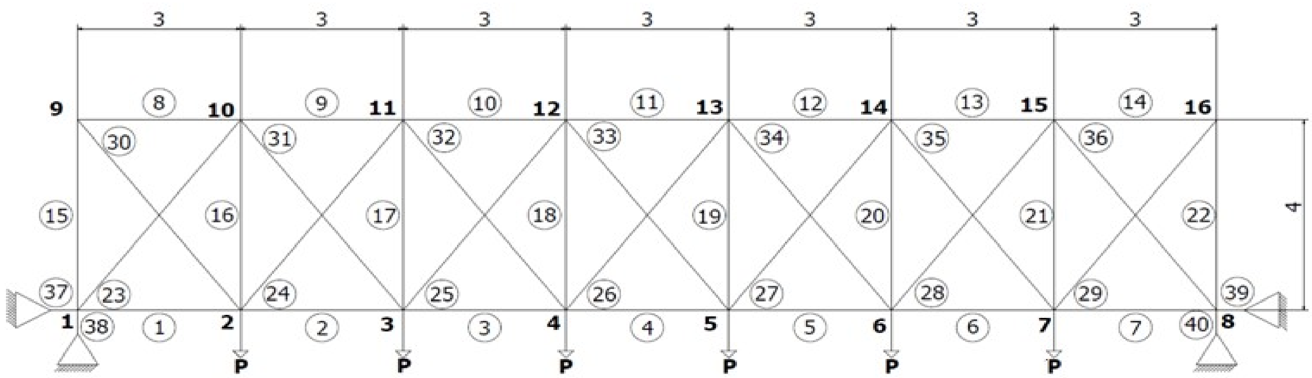

Fig. 2 . The 40 - bar truss structure

RESULTS AND DISCUSSION

The effectiveness of the present approach using integration of MATLAB and Ansys is assessed by analyzing a 40-bar truss structure which was earlier analyzed using HPSO, GSO and TLBO. The specifications, shown in Table 1, are supplied as an input. The limiting values of design variables are also shown in Table 1. After conducting a number of trials, GA is applied with the tuned parameter listed in Table 2. The results obtained using the present approach of integrating the GA and Ansys are compared with the previous results which were obtained using TLBO, HPSO, and GSO in Table 3.

TABLE 2
GA PARAMETERS

Parameter	Value
Population	100
Generations	400
Crossover Probability	0.8
Mutation Probability	0.005

Table 3 shows the optimized design variables obtained using the present approach and its comparison with the earlier approaches. It can be seen that the integration of GA and Ansys results in the minimum area which in turn yields minimum weight of the considered structure. As the weight reduces, the material required reduces and the cost of the structure also reduces. As we minimize the weight, the maximum stress occurring in the truss members may increase but it will stay under the allowable limit of the stress.

It should be noted that the results written in Table 3 are the least weight results found from a batch of 10 complete runs. Also in most of the runs GA has exited because the average change in the objective function is less than the specified tolerance function (1e-6). The mean weight after 10 runs is 1636.45 kg with a standard deviation of 58.35.

TABLE 3
COMPARISON OF OPTIMIZED RESULTS

Variable	TLBO	GSO	HPSO	GA & ANSYS
A_1	0.0010	0.0015	0.0055	0.0011
A_2	0.0010	0.0010	0.0010	0.0010
A_3	0.0010	0.0010	0.0105	0.0017
A_4	0.0010	0.0010	0.0010	0.0028
A_5	0.0010	0.0010	0.0010	0.0027
A_6	0.0030	0.0030	0.0025	0.0024
A_7	0.0035	0.0035	0.0030	0.0032
A_8	0.0035	0.0035	0.0245	0.0036
A_9	0.0010	0.0010	0.0025	0.0019
A_{10}	0.0010	0.0010	0.0010	0.0010
A_{11}	0.0010	0.0010	0.0010	0.0011
A_{12}	0.0010	0.0010	0.0010	0.0014
A_{13}	0.0010	0.0010	0.0010	0.0011
A_{14}	0.0010	0.0010	0.0010	0.0011
A_{15}	0.0010	0.0010	0.0015	0.0011
A_{16}	0.0010	0.0010	0.0050	0.0011
A_{17}	0.0025	0.0025	0.0040	0.0014
A_{18}	0.0010	0.0010	0.0010	0.0011
A_{19}	0.0010	0.0010	0.0010	0.0011
A_{20}[a]	- -	- -	- -	0.0020
Y_9	1.004	1.069	1.006	1.044
Y_{10}	2.412	2.307	2.791	3.127
Y_{11}	2.737	2.851	3.541	3.694
Y_{12}	3.314	3.287	3.396	3.131
Weight (kg)	2058.805	2080.673	3653.010	1525.935

Note. The results for TLBO are from [5]. The results for GSO and HPSO are from [11].

Some critical conclusion can be drawn from Figure 3 and the Table 3. It is observed that the members 3, 5 (A_3), 4 (A_4), 10, 12 (A_7) and 11 (A_8) have a greater area thereby a greater weight to withstand the forces applied to the truss structure, as they are the farthest from the support joints. The areas A3 and A4 are much greater while comparing with TLBO, HPSO and GSO whereas areas A7 and A8 are almost equal to the areas of TLBO, HPSO and GSO. As the structure is optimized there is a significant reduction in area in the members 30 and 36 ($A_1$7) of the order of 44% in TLBO and GSO whereas the percentage reduction in case of HPSO is 65%.

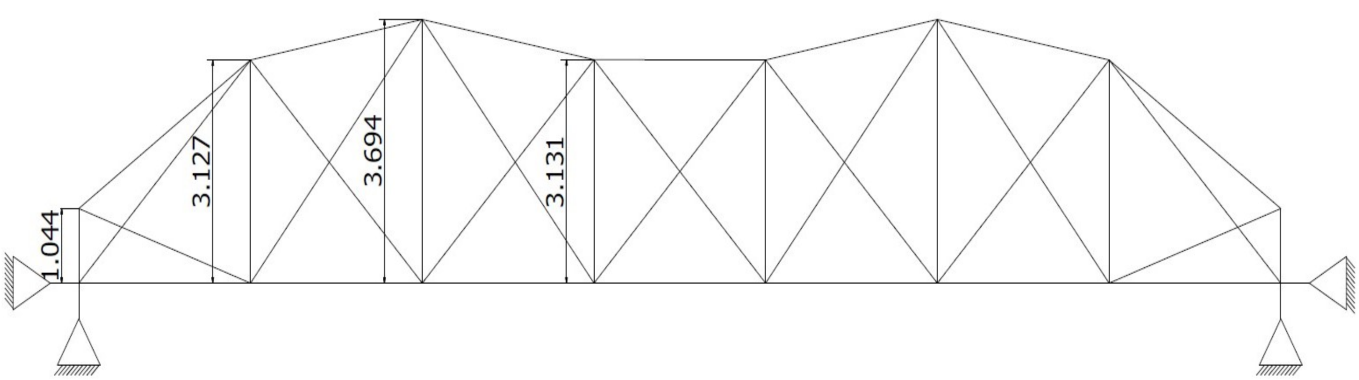

Fig. 3 . The 40 - bar truss structure

Figure 3 shows the optimized shape of the 40-bar truss structure. The structure has a weight of 1525.935 kg with a maximum compressive stress value of 153.05 MPa in members 37 and 40. The maximum deflection is at node 4 with a value of 0.0298 m.

The greater area of members 8, 14 (A_5) and 15, 22 (A_9) depicts that the maximum load is being borne by the outer frame of the truss structure. The areas of the remaining members are approximately equal to the areas obtained by TLBO, HPSO and GSO.

The structure obtained by GA and Ansys is analogous to that obtained by HPSO in the location of the top nodes 9, 10, 11, 12, 13, 14, 15 and 16. It follows an increasing trend when going from left to right then decreasing at the center nodes 12 and 13 and then again increasing till node 15 and decreasing to node 16. The height of the truss structure generated by GA and Ansys is greater when compared with the results from TLBO, HPSO and GSO.

There is a significant reduction of weight when GA and Ansys are used as compared to other optimization algorithms. The weight is reduced by 25.88% when compared with TLBO, 26.66% when compared with GSO and 58.23% when compared with HPSO.

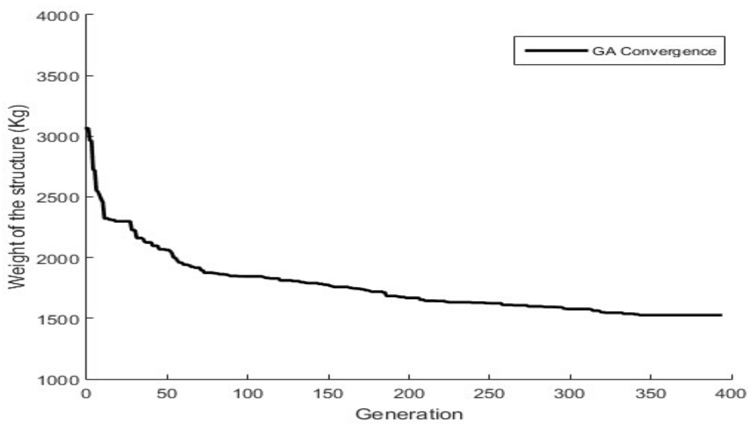

Fig. 4 . Convergance of GA

CONCLUSION AND RECOMMENDATIONS

The convergence of the proposed method is shown in Figure 4. It is observed from the Fig. 4 that the weight of the structure started converging within 298 generations. The genetic algorithm has stopped at 393th generation as the average change in the objective function is less than the tolerance limit (1e-6).

This study demonstrates the successful application of integration of GA and Ansys for Size and Shape optimization of a truss structure. The design variables were optimized in order to achieve the minimum weight. The capability of the method was demonstrated by comparing the results with those obtained by HPSO, GSO and TLBO. It is observed from the comparative result that present approach is 25.88% lesser in weight than TLBO, 58.23% lesser in weight than HPSO and 26.66% lesser in weight than GSO.

Declaration of Conflicting Interests

There are no competing interests.

REFERENCES

[1] H. Kim, O. M. Querin and G. P. Steven, “On the development of structural optimisation and its relevance in engineering design,” *Design Studies,* vol. 23, no. 1, pp. 85-102, 2002.

[2] M. Carvalho, J. L. Vale, L. T. Pinheiro and M. M. Neves, “Techniques adopted in integrating structural analysis with numerical optimization,” in *CD of Proceedings of 5th International Conference on Mechanics and Materials in Design, Porto,* 2005.

[3] S Darshan, V. Akshay, N. K. Anirudh and K. Rahul, “Size and topological optimization of cantilever beam,” *International Journal of Engineering Trends and Technology,* vol. 4, no. 5, pp. 2077-2082, 2013.

[4] K. R. Indu and M. G. Airin, “Optimization of tapered cantilever beam using Genetic Algorithm: Interfacing MATLAB

and ANSYS," *International Journal of Innovative Research in Science, Engineering and Technology,* vol. 4, no. 10, pp. 10145-10150, 2015.

[5] W. Cheng, F. Liu and L. Li, "Size and geometry optimization of Trusses using teaching-learning-based optimization," *International Journal of Optimization in Civil Engineering,* vol. 3, no. 3, pp. 431-444, 2013.

[6] A. Gauchia, B. L. Boada, M. J. L. Boada and V. Diaz, "MATLAB applications for practical engineer," in *Integration of MATLAB and ANSYS for Advanced Analysis of Vehicle Structures,* K. Bennett, Ed., Rijeka: Croatia, InTech, 2014.

[7] K. Deb and S. Gulati, "Design of truss-structures for minimum weight using genetic algorithms," *Finite Elements in Analysis and Design,* vol. 37, no. 5, pp. 447-465, 2001.

[8] S. Reh, J. D. Beley, S. Mukherjee and E. H. Khor, "Probabilistic finite element analysis using ANSYS," *Structural Safety,* vol. 28, no. 1, pp. 17-43, 2006.

[9] T. Manning, R. D. Sleator and P. Walsh, "Naturally selecting solutions: The use of genetic algorithms in bioinformatics," *Bioengineered,* vol. 4, no. 5, pp. 266-278, 2013.

[10] X. S. Yang, *Engineering Optimization: An Introduction with Metaheuristic Applications.* Hoboken, NJ: John Wiley & Sons, 2010.

[11] L. J. Li and F. Liu, *Group Search Optimization for Applications in Structural Design,* Heidelberg, Germany: Springer Berlin Heidelberg, 2011.

[12] Y. A. N. Yun, "Design of structure optimization with APDL," *Journal of East China Jiaotong University,* vol. 4, pp. 52-55, 2004.

[13] K. Zeng, D. Pal, H.J. Gong, N. Patil, N. and B. Stucker, "Comparison of 3DSIM thermal modelling of selective laser melting using new dynamic meshing method to ANSYS". *Materials Science and Technology,* vol. 31, no. 8, pp. 945-956, 2015.

[14] O. A. Negrozov, P. A. Akimov, and I. Y. Lantsova, "Semianalytical structural analysis based on combined application of finite element method and discrete-continual finite element method Part 4: Verification samples." *Procedia Engineering*, vol. 153, pp. 926-932, 2016.

[15] M. T. Bhoskar, M. O. K. Kulkarni, M. N. K. Kulkarni, M. S. L. Patekar, G. M. Kakandikar, and V. M. Nandedkar. "Genetic algorithm and its applications to mechanical engineering: A review," *Materials Today: Proceedings*, vol. 2, no. 4-5, pp. 2624-2630, 2015.

— This article does not have any appendix. —