TOPOLOGY OPTIMIZATION OF CONTINUUM STRUCTURES USING OPTIMALITY CRITERION APPROACH IN ANSYS

Dheeraj Gunwant & Anadi Misra
Department of Mechanical Engineering,
G. B. Pant University of Agriculture and Technology, Pantnagar, India

ABSTRACT

Topology optimization is an important category of structural optimization which is employed when the design is at the conceptual stage. Generally, the topology optimization deals with finding the optimal material distribution in a design domain while minimizing the compliance of the structure. In this work, focus has been kept on the topology optimization of five benchmark plane stress models through a commercially available finite element software ANSYS. ANSYS employs topology optimization using the Solid Isotropic Material with Penalization (SIMP) scheme for the penalization of the intermediate design variables and the Optimality Criterion for updating the design variables. The results of the ANSYS based Optimality criterion are validated and compared with the results obtained by Element Exchange Method.

KEYWORDS: *Topology Optimization, Pseudo-densities, Compliance minimization, Optimality Criterion, SIMP.*

I. INTRODUCTION

Designers are many times faced with the problems of deciding the optimal layout (distribution of material) or topology of the design. They have to make trade-offs between various factors to achieve a sensible design, which satisfies the performance criteria imposed on it satisfactorily. While doing this he has to examine a large number of candidate solutions and find a globally optimal solution which satisfies the boundary conditions imposed on it. The task of searching globally optimal solutions is more cumbersome when the design is at conceptual stage.

Therefore, in an optimisation problem, different candidate solutions are compared with each other, and then the best or optimal solution is obtained which means that solution quality is fundamental. In engineering, the optimisation of an objective function is basically the maximisation or minimisation of a problem subjected to constraints.

Optimisation can basically be categorised into three types namely: a) sizing (mass), b) shape and c) topology (layout). Refer figure below.

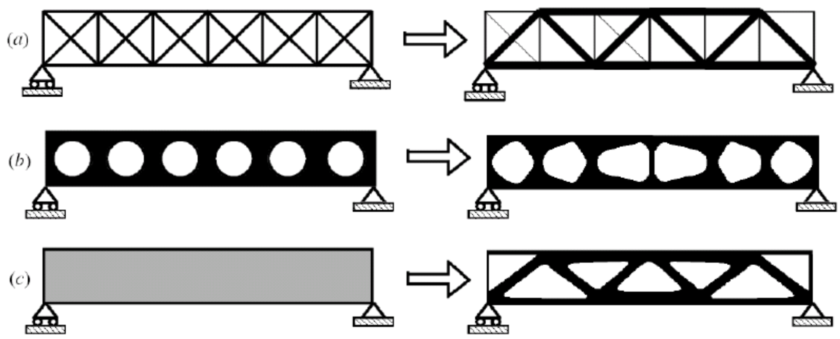


Fig. 1: (a) Sizing, (b) Shape and (c) Topology optimization

This paper basically focuses on topology or layout optimisation, so that will be discussed in detail.

1.1. A word about topology optimization

Topology optimization is perhaps the most difficult of all three types of structural optimization. The optimization is performed by determining the optimal topology of the structure. Hence, the design variables control the topology of the design. Optimization therefore occurs through the determination of design variable values which correspond to the component topology providing optimal structural behaviour. While it is easy to control a structure's shape and size as the design variables are the coordinates of the boundary (shape optimization) or the physical dimensions (size optimization), it is difficult to control the topology of the structure.

In this problem, the design domain is created by assembling a large number of basic elements or building blocks. By beginning with a set of building block representing the maximum allowable region (region in space which the structure may occupy) each block is allowed to either exist or vanish from the design domain, a unique design is evolved. For example in the topology optimization of a cantilever plate, the plate is descritized into small rectangular elements (building blocks), where each element is controlled by design variables which can vary continuously between 0 and 1. When a particular design variable has a value of 0, it is considered to be a hole, likewise, when a design variable has a value of 1, it is considered to be fully material. The elements with intermediate values are considered materials of intermediate densities.

The development of topological optimization can be attributed to Bendsøe and Kikuchi [1988]. They presented a homogenization based optimization approach of topology optimization. They assumed that the structure is formed by a set of non-homogenous elements which are composed of solid and void regions and obtained optimal design under volume constraint through optimization process. In their method, the regions with dense cells are defined as structural shape, and those with void cells are areas of unnecessary material. The maximization of the integral stiffness of a structure composed of one or two isotropic materials of large stiffness using the homogenization technique was discussed by Thomsen [1992]. Numerical results are presented at the end of the paper. Application of Genetic algorithm for topology optimization was made by Chapman [1994]. Given structure's boundary conditions and allowable design domain, a descritized design domain is created. The genetic algorithm then generates an optimal structure topology by evolving a population of chromosomes, where each chromosome, after mapping into the design domain creates a potentially-optimal structure topology.

Diaz and Sigmund [1995] computed the effective properties of strong and weak materials. It is shown that when 4-noded quadrilateral elements are used, the resulting topology consists of artificially high stiffness material which is difficult to manufacture. This material appears in specified manner and is known as the checker board pattern due to alternate solid and void elements. Swan and Kosaka [1997] investigated a continuous topology optimization framework based on hybrid combinations of classical Reuss (compliant) and Voigt (stiff) mixing rules. To avoid checker boarding instabilities, the continuous topology optimization formulation is coupled with a novel spatial filtering procedure. Sigmund and Petersson [1998] summarized the current knowledge about numerical instabilities such as checkerboards, mesh-dependence and local minima occurring in applications of the topology optimization method. The checkerboard problem refers to the formation of regions of alternating solid and void elements ordered in a checkerboard-like fashion. The mesh-dependence problem refers to obtaining qualitatively different solutions for different mesh-sizes or diseretizations. A local minimum refers to the problem of obtaining different solutions to the same descritized problem when choosing different algorithmic parameters.

A web-based interface for a topology optimization program was presented by Tcherniak and Sigmund [2001]. The program is available over World Wide Web. The paper discusses implementation issues and educational aspects as well as statistics and experience with the program. Allaire *et al.* [2002] studied a level-set method for numerical shape optimization of elastic structures. The approach combines the level-set algorithm of Osher and Sethian with the classical shape gradient. Although this method is not specifically designed for topology optimization, it can easily handle topology changes for a very large class of objective functions. Rahmatalla and Swan [2004] presented a node-based design variable implementation for continuum structural topology optimization in a finite element framework and explored its properties in the context of solving a number of different design examples. Since the implementation ensures C^0 continuity of design variables, it is immune to element wise checker boarding instabilities that are a concern with element-based design variables. The objective of maximizing the Eigen frequency of vibrating structures for avoiding the resonance condition was considered by Du and Olhoff [2005]. This can also be achieved by maximizing the gap between two consecutive frequencies of the given order. Different approaches are considered and discussed for topology optimization involving simple and multiple Eigen frequencies of linearly elastic structures without damping. The mathematical formulations of these topology optimization problems and several illustrative results are presented.

Sigmund and Clausen [2007] suggested a new way to solve pressure load problems in topology optimization. Using a mixed displacement–pressure formulation for the underlying finite element problem, we define the void phase to be an incompressible hydrostatic fluid. Rozvany [2008] evaluated and compared the established numerical methods of structural topology optimization that have reached the stage of application in industrial software. Dadalau *et al.* [2008] presented a new penalization scheme for the SIMP method. One advantage of the present method is the linear density-stiffness relationship which has advantage for self weight or Eigen frequency problem. The topology optimization problem is solved through derived Optimality criterion method (OC), which is also introduced in the paper. Rouhi *et al*. [2010] presented a stochastic direct search method for topology optimization of continuum structures. In a systematic approach requiring repeated evaluations of the objective function, the element exchange method (EEM) eliminates the less influential solid elements by switching them into void elements and converts the more influential void elements into solid resulting in an optimal 0–1 topology as the solution converges. For compliance minimization problems, the element strain energy is used as the principal criterion for element exchange operation. Gunwant *et. al.* obtained topologically optimal configuration of sheet metal brackets using Optimality Criterion approach through commercially available finite element solver ANSYS and obtained compliance versus iterations plots for various aspect ratio structures (brackets) under different boundary conditions.

1.2. Topology optimization using ANSYS

The goal of topological optimization is to find the best use of material for a body such that an objective criterion (i.e. global stiffness, natural frequency, etc.) attains a maximum or minimum value subject to given constraints (i.e. volume reduction).

In this work, maximization of static stiffness has been considered. This can also be stated as the problem of minimization of compliance of the structure. Compliance is a form of work done on the structure by the applied load. Lesser compliance means lesser work is done by the load on the structure, which results in lesser energy is stored in the structure which in turn, means that the structure is stiffer.

Mathematically,

$$Compliance = \int_V \ fu\, dV + \int_S \ tu\, dS + \sum_i^n F_i u_i \qquad 3.1$$

Where,

u = Displacement field

f = Distributed body force (gravity load etc.)

F_i = Point load on i^{th} node

u_i = i^{th} displacement degree of freedom

t = Traction force

S = Surface area of the continuum

V = Volume of the continuum

ANSYS employs gradient based methods of topology optimization, in which the design variables are continuous in nature and not discrete. These types of methods require a penalization scheme for evolving true, material and void topologies. SIMP (Solid Isotropic Material with Penalization) is a most commonly penalization scheme, and is explained in the next section.

1.3. The SIMP method

The SIMP stands for Solid Isotropic Material with Penalization method. This is the penalization scheme or the power law through which is the basis for evolution of a 0-1 topology in gradient based methods.

In the SIMP method, each finite element (formed due to meshing in ANSYS) is given an additional property of pseudo-density, x_j where $0 \le x_j \le 1$, which alters the stiffness properties of the material.

$$x_j = \frac{\rho_j}{\rho_o} \quad 3.2$$

Where,
ρ_j= Density of the j^th^ element
ρ_o= Density of the base material
x_j= Pseudo-density of the j^th^ element

This Pseudo-density of each finite element serves as the design variables for the topology optimization problem. The Pseudo-density of j^th^ element K_j depends on its Pseudo-density x_j in such a way that,

$$K_j = x_j{}^p K_o \quad 3.3$$

Where,
K_o = Stiffness of the base material
$p > 1$ = Penalization power
As is clear from equation 3.3,
For, $x_j = 0$
K_o= 0, which means no material exists
For, $x_j = 1$
K_o= 1, which means that material exists.
In SIMP p is taken to be greater than 1 so that intermediate densities are unfavourable in the sense that that the stiffness obtained is small as compared with the volume of the material.
In other words, specifying a value of p higher than 1 makes it uneconomical to have intermediate densities in the optimal design.
As a matter of fact, for problems where the volume constraint is active, experience shows that optimization does actually result in such designs if one chooses p sufficiently large (in order to achieve complete 0-1 designs, $p > 1$ is usually required).
In ANSYS, the standard formulation of topology optimisation problem defines the problem as minimising the structural stiffness and maximising the fundamental frequency while satisfying a constraint on volume of the structure. Another problem is the maximisation of natural frequency of the structure subjected to dynamic loading, while satisfying a constraint on the volume of the structure.
The objective function (function which is to be minimized in topology optimization) is generally the compliance of the structure. A constraint on usable volume is applied on the structure. As the volume reduces, the structure's stiffness also reduces. So the volume constraint is of opposing nature.
The Compliance of a descritized finite element is given by,

$$c(x) = F^T u \quad 3.4$$

The force vector (which is a function of the design variables x_j) is given by,

$$K(x)u = F \quad 3.5$$

Therefore, $c(x)$ can be written as,

$$c(x) = u^T K u = \sum_{j=1}^{n} u_j^T K_j(x_j) u_j \quad 3.6$$

$$subject\ to \sum_{j=1}^{n} x_j v_j \le V_o \quad 3.7$$

$$0 < x_{min} \le x_j \le 1 \quad j = 1, 2, 3 \dots \dots . . n$$

A lower bound on the design variables has been applied to avoid singularity in the stiffness matrix.
We have used a gradient based, heuristic approach the Optimality Criterion approach in this work. The Optimality Criterion method is described in the next section.

II. MATERIALS AND METHODS

2.1. The Optimality Criterion approach

The discrete topology optimization problem is characterized by a large number of design variables, N in this case. It is therefore common to use iterative optimization techniques to solve this problem, e.g. the method of moving asymptotes optimality criteria (OC) method, to name two. Here we choose the latter. At each iteration of the OC method, the design variables are updated using a heuristic scheme. The Lagrangian for the optimization problem is defined as:

$$\mathcal{L}(x_j) = u^T Ku + \Lambda\left(\sum_{j=1}^{n} x_j v_j - V_o\right) + \lambda_1(Ku - F) + \sum_{j=1}^{n} \lambda_2^j\left(x_{min} - x_j\right) + \sum_{j=1}^{n} \lambda_3^j\left(x_j - 1\right) \qquad 3.8$$

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

$$\frac{\partial \mathcal{L}}{\partial x_j} = 0 \qquad j = 1, 2, 3 \ldots . n \qquad 3.9$$

Now, Compliance,

$$C = u^T Ku \qquad 3.10$$

Differentiating eq. 3.8 w. r. t. x_j, the optimality condition can be written as:

$$B_j = \frac{-\frac{\partial C}{\partial x_j}}{\Lambda v_j} = 1 \qquad 3.11$$

The Compliance sensitivity can be evaluated as using eq.:

$$\frac{\partial C}{\partial x_j} = -p(x_j)^{p-1} {u_j}^T k_j u_j \qquad 3.12$$

Based on these expressions, the design variables are updated as follows:

$$\begin{aligned} x_j^{new} &= \left\{\max(x_{min} - m)\,, if\ x_j B_j^{\eta} \leq (x_{min}, x_{min} - m\right\} \\ &= \left\{x_j B_j^{\eta}, if\ \max(x_{min} - m) < x_j B_j^{\eta} < \min(1, x_j + m\right\} \\ &= \left\{\min(1, x_j + m, if\ \min(1, x_j + m) \leq x_j B_j^{\eta}\right\} \end{aligned} \qquad 3.13$$

Where, m is called the move limit and represents the maximum allowable change in x_j in a single OC iteration. Also, η is a numerical damping coefficient, and is usually taken to be $1/2$. The Lagrange multiplier for the volume constraint Λ is determined at OC iteration using a bisection algorithm. x_j is the value of the density variable at each iteration step. u_j is the displacement field at each iteration step determined from the equilibrium equations.
The optimization algorithm structure is explained in the following steps:
- Make initial design, e. g. homogenous distribution of material.
-For this distribution of density, compute by finite element method the resulting displacements and strains.
-Compute the compliance of the design. If only marginal improvement in compliance over last design, stop iterations. Else, continue.
-Compute the update of design variable, based on the scheme shown in eq. 3.13. this step also consists of an inner iteration loop for finding the value of Lagrange multiplier Λ for the volume constraint.
-Repeat the iteration loop.
This paper considers the maximization of static stiffness through the inbuilt topological optimisation capabilities of the commercially available FEA software to search for the optimum material distribution in five plane stress structures as used by [15]. The optimum material distribution depends upon the configuration of the initial design space and the boundary conditions (loads and constraints). The goal of the paper is to minimize the compliance of the bracket while satisfying the constraint on

the volume of the material reduction. Minimizing the compliance means a proportional increase in the stiffness of the material. A volume constraint is applied to the optimisation problem, which acts as an opposing constraint. To visualize, more the volume of material, lower will be the compliance of the structure and higher will be the structural stiffness of the structure. For implementation of this, APDL codes for various brackets modelling and topological optimisation were written and run in ANSYS.

2.2. Specimen Geometry and Boundary Conditions

In the present investigation, five specimen geometries and boundary conditions applied have been used as shown in the figures below. The specimens are taken from the work of **Rouhi *et al.* [2010].** All the models are under plane state of stress.

Model 1: *Messerschmitt-Bolkow-Beam (MBB):*

This is a simply supported beam of dimensions 6mmx1mmx1mm. The beam is supported by a roller support on the right hand side support and on the other end; it is supported by fixed support. The beam is acted upon by a central load of 1N. Due to symmetry of the model, only the right half of the model has been used in this study.

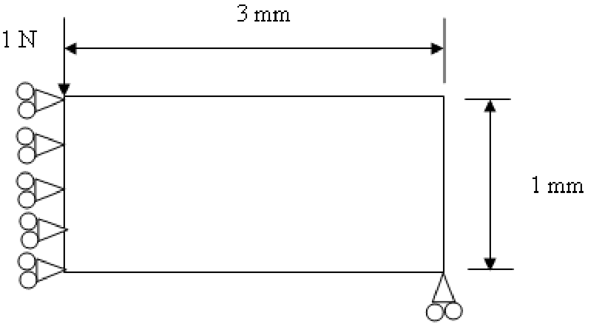


Fig 2: Geometry and boundary conditions for Model 1 (Symmetric model)

Model 2: *A cantilever with load at bottom tip*

In this case a cantilever of dimensions 8mmx5mmx1mm and loaded with a load of 1N on the bottom tip and in a state of plane stress is considered. The left hand edge is supported with fixed support as shown in figure 3 below.

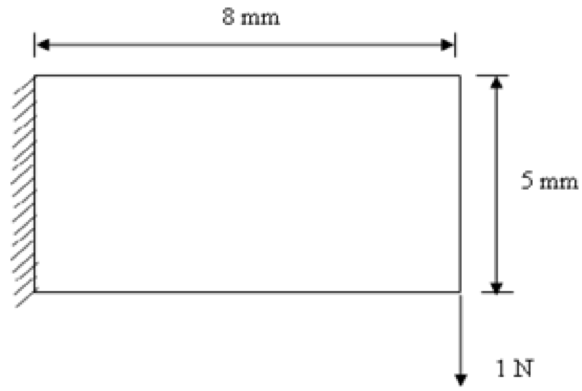


Fig 3: Geometry and boundary conditions for Model 2

Model 3: *A cantilever with load at the centre of right edge*

Figure 4 below shows the geometry and boundary conditions for model 3. The model is a 2mmx1mmx1mm structure loaded with a unit load at the middle of the right hand side edge. The left hand side edge is supported by fixed support.

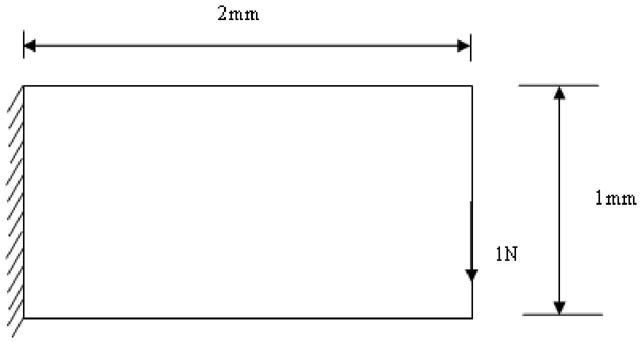


Fig 4: Geometry and boundary conditions for Model 3

Model 4: *A cantilever with load at the centre of right edge*

Figure 5 shows a short cantilever of dimensions 2mmx1mmx1mm. It is centrally loaded with a unit load at the middle of right hand side edge and is under a state of plane stress. The left hand edge is fixed.

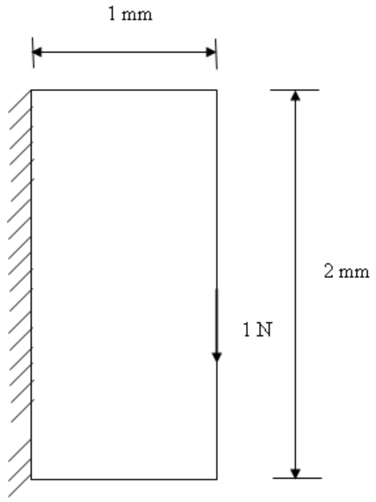


Fig 5: Geometry and boundary conditions for Model 4

Model 5: *A cantilever with load at the centre of right edge*

Figure 6 below shows the geometry and boundary conditions for the model 5. The structure is of dimension 1mmx1mmx1mm. It is subjected to a unit load at each of upper and bottom tip of the right edge and is also under a state of plane stress. The left edge is fixed.

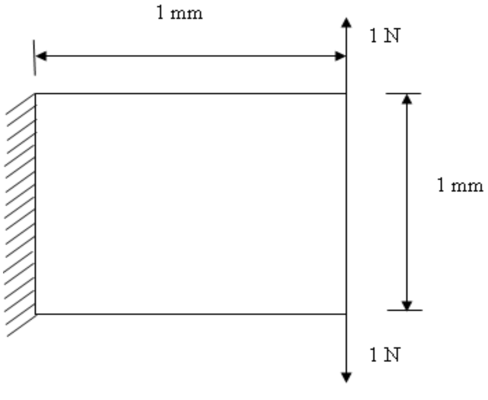


Fig 6: Geometry and boundary conditions for Model 5

The material properties used are given in the table below:

III. RESULTS

In this section, final compliance and optimal shape of the models obtained with the help of gradient based ANSYS based Optimality Criterion have been compared with a non-gradient based Element Exchange Method (EEM) in the work of **Rouhi *et al.* [2010]**. This is important in the sense that the gradient based methods are prone to entrapment in local optimum instead of the global optimum. Whereas, non-gradient based methods do not suffer with this problem. Comparison and validation is necessary to prove that this method is not giving results that are sub-optimal.

Each of the five models are characterized by the finite element descritization in x(n_x) and y(n_y) directions and the volume usage fraction(V_0) used. The material properties used are given in the table below:

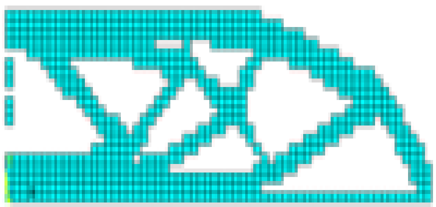
Table 1: Material Properties used

Young's Modulus (E) $(\frac{N}{mm^2})$	Poisson's ratio (ν)
1	0.3

Model 1: The beam is in the state of plane stress with a thickness of 1 mm. The beam is optimized for minimum compliance. Due to symmetry of the model, only half of the model is considered with symmetry boundary conditions as it is symmetric about the vertical axis. The beam is supported by a roller support at the bottom right corner and symmetric boundary conditions are applied on the right edge (fig. 2.) Table 2 shows the final compliance obtained in the case of ANSYS based OC and EEM.

Table 2: Comparison between OC and EEM for model 1

(n_x, n_y, V_0)	(60, 20, 0.5)	
Method	ANSYS based OC	EEM
Compliance(Nmm)	182.20	187.00
Iterations	46	210
Percentage difference in compliance values	2.56	



(a)

(b)

Fig 7: Optimal shapes obtained by (a) ANSYS based OC and (b)EEM for $(n_x, n_y, V_0) = 60, 20, 0.5$ for model 1

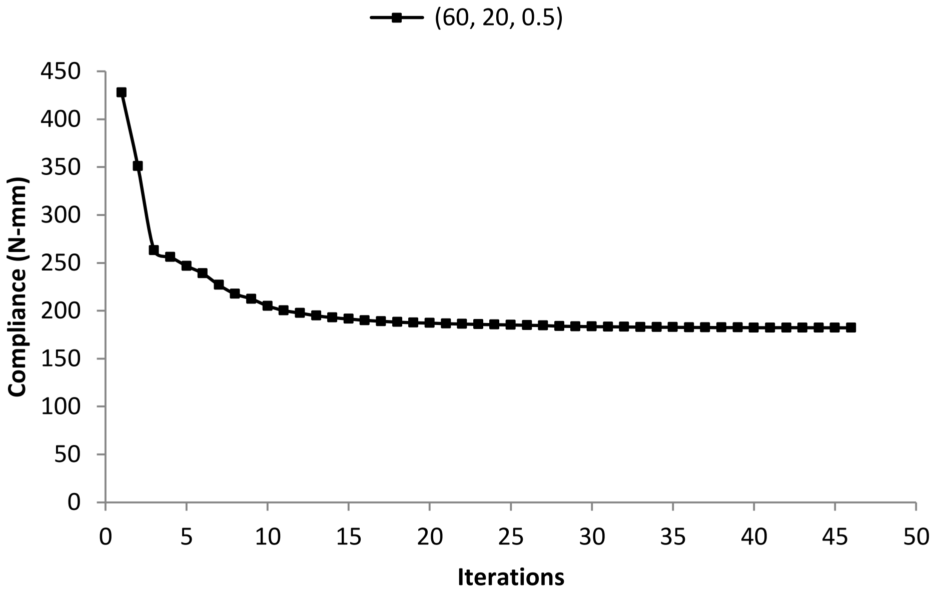


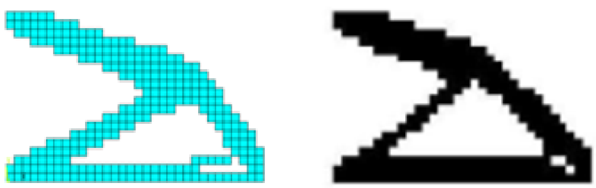
Fig 8: Convergence plot for ANSYS based Optimality criterion for model 1

Initial compliance obtained in the first iteration is 427.93 Nmm, which drops to 350.86 Nmm in second iteration and 263.00 Nmm in the third iteration. The final compliance obtained with the help of ANSYS based OC is 182.20 Nmm after 46 iterations. On the other hand, the EEM gives a final compliance of 187.00 after 210 iterations. ANSYS based OC reaches a more optimal solution in lesser number of iterations.

Model 2: A cantilever beam of dimensions 8mm x 5mm and thickness 1mm is considered in this case. The cantilever is under a state of plane stress and supports a concentrated load of magnitude 1N at the bottom right corner. The left hand side edge is fixed (fig 3). Table shows the final compliance obtained with ANSYS based OC and EEM for mesh densities of 32, 20 and 64, 40 and a volume usage fraction of 40%.

Table 3: Comparison between OC and EEM for model 2

(n_x, n_y, V_0)	Coarse mesh (32, 20, 0.4)		Fine mesh (64, 40, 0.4)	
Method	ANSYS based OC	EEM	ANSYS based OC	EEM
Compliance (Nmm)	52.22	53.60	53.41	57.00
Iterations	39	178	30	174
Percentage difference in compliance values	2.57		6.3	



(a) (b)

Fig 9: Optimal shapes obtained by (a) ANSYS based OC and (b)EEM for $(n_x, n_y, V_0) = 32, 20, 0.4$ for model 2

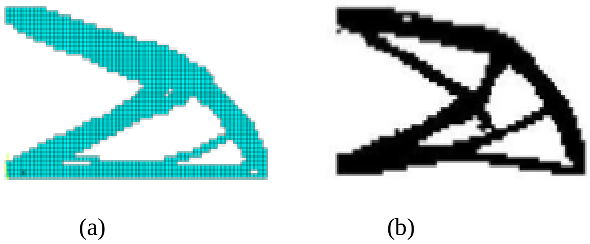


Fig 10: Optimal shapes obtained by (a)ANSYS based OC and (b)EEM for $(n_x, n_y, V_0) = 64, 40, 0.4$ for model 2

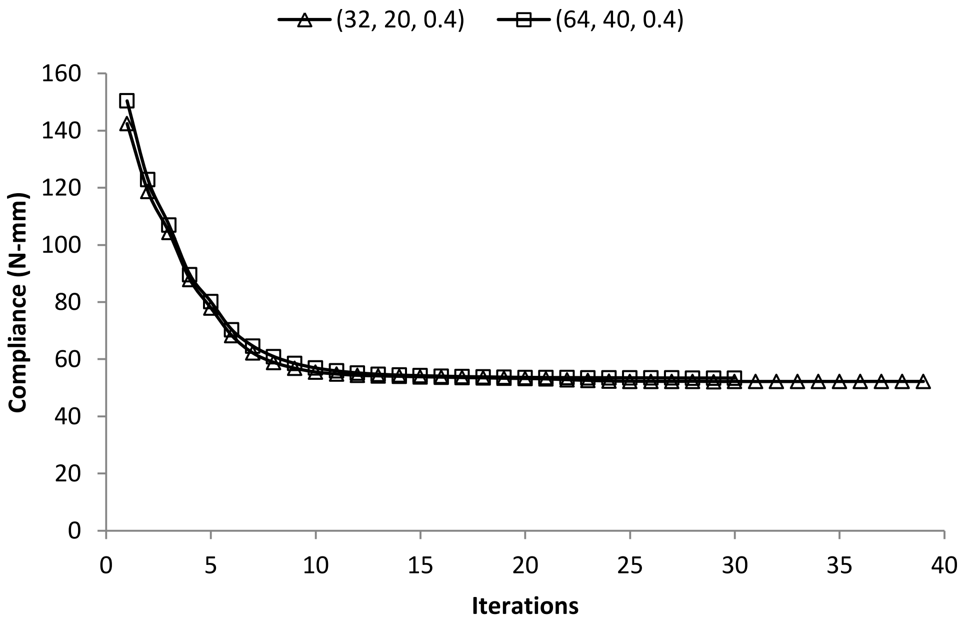


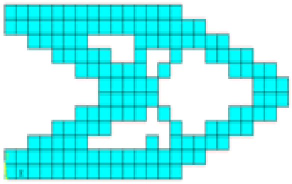
Fig 11: Convergence plot for ANSYS based OC for model 2

For a mesh size of 32, 20 and volume usage fraction of 40%, the initial compliance obtained is 142.43 Nmm in the first iteration, which drops to 118.68 Nmm and 104.43 Nmm in second and third iterations respectively. The final compliance obtained is 52.22 Nmm after 39 iterations. EEM gives a minimum compliance of 53.60 Nmm after 178 iterations. The optimal shapes, as shown in the Fig. 10 (a) and (b) are almost same.
When the mesh size is changed to 64, 40 and volume usage fraction of 40%, the initial compliance obtained is 150.39 Nmm in the first iteration, which drops to 122. 86 Nmm and 106.97 Nmm after second and third iterations. Final compliance obtained is 53.41 Nmm after 30 iterations in case of ANSYS based OC and in case of EEM it is 57.00 Nmm after 174 iterations. Optimal shapes are shown in Fig. 11(a) and (b) are different.

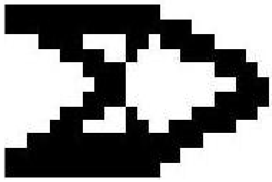
Model 3: A cantilever beam of dimensions 2mm x 1mm and thickness 1 mm is considered in this case. The cantilever beam is in the state of plane stress and is subjected to a load of 1 N in the center of the right edge as shown in Fig 4. Final compliance obtained using ANSYS based OC has been compared with those reported by EEM and Genetic algorithm and are given in the table 4 below.

Table 4: Comparison between ANSYS based OC, EEM and GA for model 3 for different mesh densities

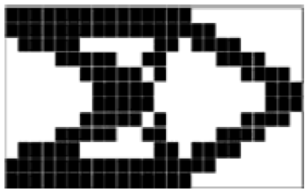
(n_x, n_y, V_0)	Coarse mesh (24, 12, 0.5)			Fine mesh (48, 24, 0.5)	
Method	ANSYS based OC	EEM	GA	ANSYS based OC	EEM
Compliance(Nmm)	63.20	66.10	64.40	62.73	63.50
Iterations	99	150	$4x10^4$	33	250
Percentage difference in compliance values	4.4			1.2	



(a)

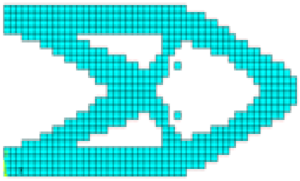


(b)

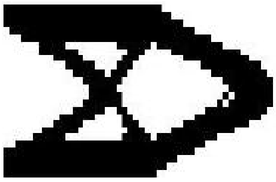


(c)

Fig 12: Optimal shapes obtained by (a) ANSYS based OC and (b) EEM and (c) Genetic Algorithm for $(n_x, n_y, V_0) = 24, 12, 0.5$ for model 3



(a)



(b)

Fig 13: Optimal shapes obtained by (a) ANSYS based OC and (b) EEM for $(n_x, n_y, V_0) = 48, 24, 0.5$ for model 3

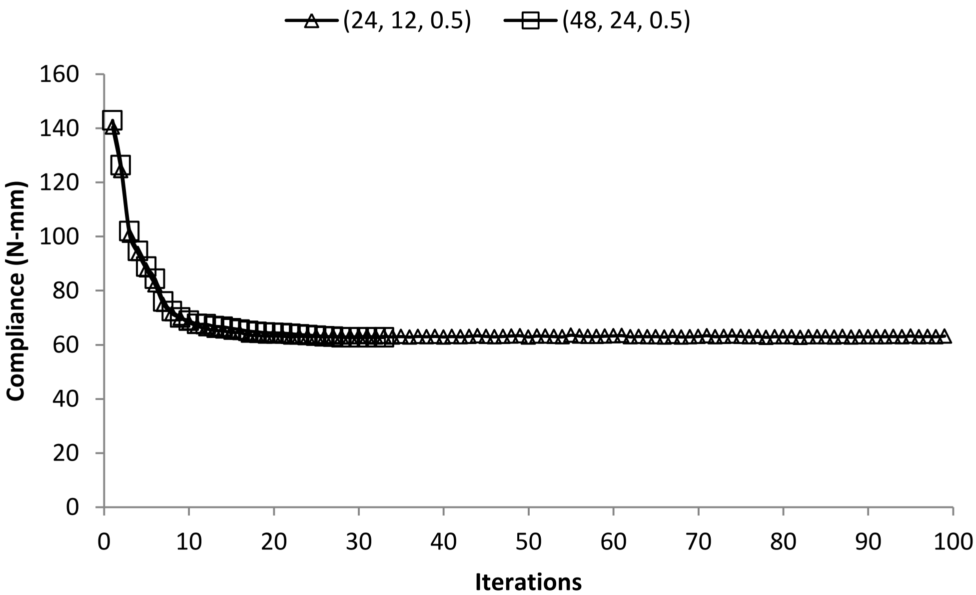


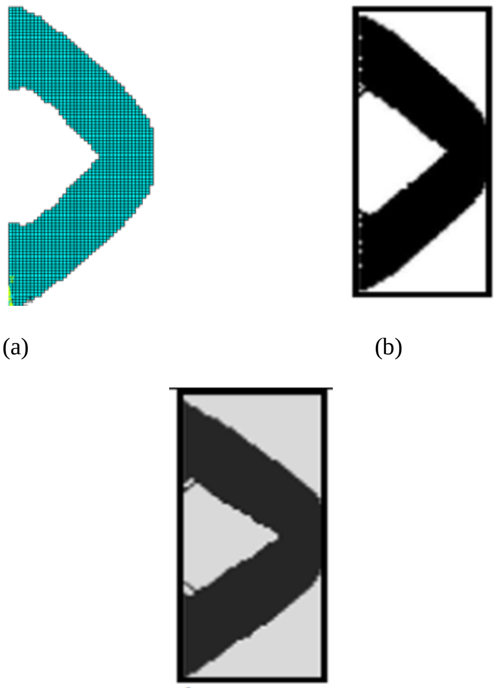
Fig 14: Convergence plot for ANSYS based OC for model 3.

In the case when mesh density is 24, 12, the initial compliance is 140.39 Nmm in the first iteration, which drops to 124.38 Nmm in the second and 100.45 Nmm in the third iteration. The final compliance is 63.20 Nmm in this case after 99 iterations. When the mesh density is 50, 50, the final compliance is 65.89 Nmm after 28 iterations. In the case of EEM, the mesh densities of 24, 12 and 48, 24 yield a final compliance of 66.1 Nmm and 63.50 Nmm respectively after 150 and 250 iterations respectively. GA yields a final compliance of 64.40 Nmm for a mesh density of 24, 12 after $4x10^4$ iterations.

Model 4: A short cantilever with dimensions 1mm x 2mm and thickness of 1mm is considered in this case. The cantilever is in the plane state of stress and is acted upon by a unit load at the center of the right edge as shown in Fig 5 above. Table 5 below lists the final compliances of the model obtained by ANSYS based OC, EEM and Particle Swarm Optimization (PSO) for the mesh densities of 20, 47 and 40, 94.

Table 5: Comparison between OC, EEM and PSO for model 4

(n_x, n_y, V_0)	Coarse mesh (20, 47, 0.5)			Fine mesh (40, 94, 0.5)		
Method	ANSYS based OC	EEM	PSO	ANSYS based OC	EEM	PSO
Compliance (Nmm)	4.77	2.96	Not reported	5.40	5.10	Not reported
Iterations	12	100	10^5	15	103	10^3
Percentage difference in compliance values	37.9			5.55		



(a) (b)

(c)

Fig 15: Optimal shapes obtained by a) ANSYS based OC and b) EEM and (c) PSO for $(n_x, n_y, V_0) = 40, 94, 0.5$ for model 4

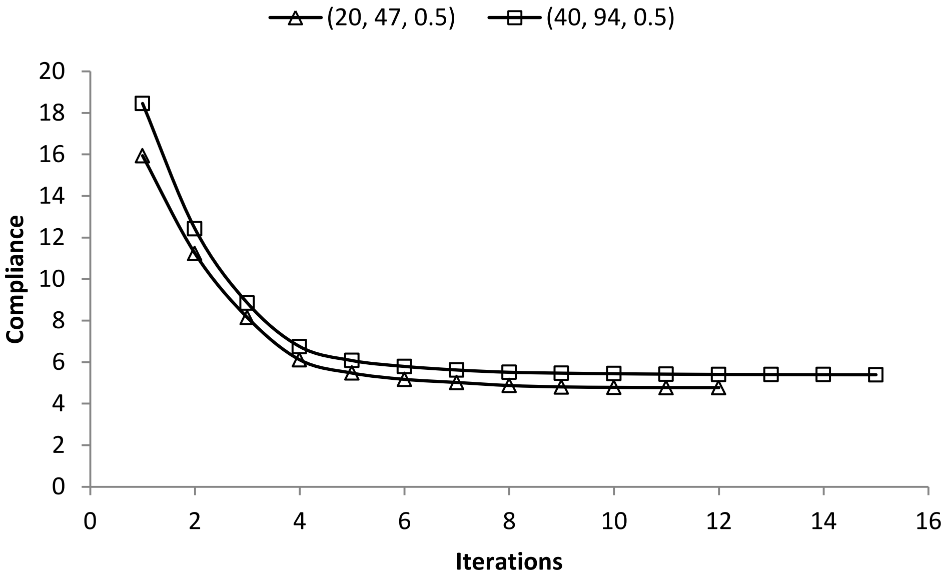


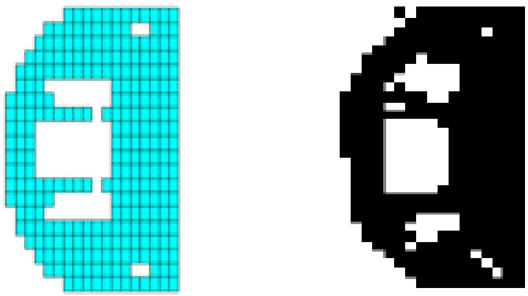
Fig 16: Convergence plot for ANSYS based OC for model 5

It is clear from the figure that ANSYS based OC converges to results very fast and within 20 iterations for both mesh sizes. For mesh density 20, 47, the initial compliance is 15.94 Nmm which drops to a final compliance of 4.77 Nmm. For a mesh density of 40, 94, the ANSYS based OC reached optimal solution in 23 iterations. Initial compliance being 15 Nmm and final compliance is 5.38 Nmm. Although there is difference in the optimal structures at mesh densities of 20, 47, but the optimal shape obtained in the case of 40, 94 mesh densities are almost same. It is also clear that ANSYS based OC reaches optimal solution in lesser number of iterations.

Model 5: In this case a doubly loaded cantilever of dimensions 1mm x 1mm and plane thickness of 1mm is considered. The cantilever is loaded with a unit load at each of the bottom and top corner points as shown in the Fig 6. The compliance values obtained by ANSYS based OC at different mesh densities are compared with EEM. Table 6 shows the final compliance obtained for each mesh densities and number of iterations taken be ANSYS based OC and EEM.

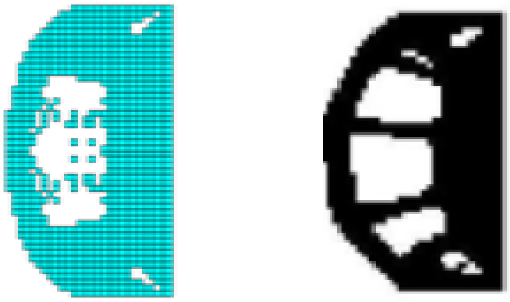
Table 6: Comparison between OC and EEM for model 5

(n_x, n_y, V_0)	Coarse mesh (32, 20, 0.4)		Fine mesh (50, 50, 0.4)	
Method	ANSYS based OC	EEM	ANSYS based OC	EEM
Compliance (Nmm)	15.23	17.48	17.29	19.70
Iterations	15	73	17	37
Percentage difference in compliance values	12.8		12.22	



(a) (b)

Fig 17: Optimal shapes obtained by (a) ANSYS based OC and (b) EEM for $(n_x, n_y, V_0) = (32, 20, 0.4)$ form model 5



(a) (b)

Fig 18: Optimal shapes obtained by (a) ANSYS based OC and (b) EEM for $(n_x, n_y, V_0) = (50, 50, 0.4)$ for model 5

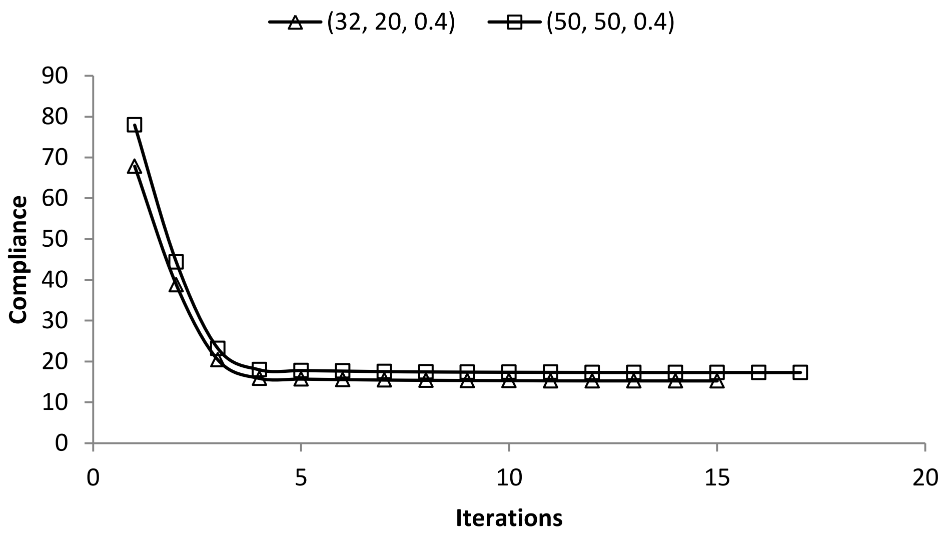


Fig 19: Convergence plot for ANSYS based OC for model 5

The initial compliance obtained in this case for a mesh density of 32, 20 is 67.85 Nmm which eventually drops to 15.23 Nmm after 15 iterations. For a mesh density of 50, 50 Nmm, the initial and final compliances are 77.93 Nmm and 17.30 Nmm. The final compliance is attained in this case after 17 iterations. EEM yields 17.48 Nmm and 19.70 Nmm after 73 and 37 iterations for mesh densities of 32, 20 and 50, 50 respectively.

IV. DISCUSSION

Of all the stages of the design process, the *conceptual design (topological optimization)* phase is considered to be the most critical. It is an early stage of the design process, yet decides much of the structure's final design. Because, the design revisions are expensive at the later stages of the design, design decisions made in the conceptual phase must be planned and executed thoroughly. Unfortunately, till date very less importance is given to the conceptual design phase and most of the decisions about the form of the design are left to the designer's intuition. The shape and pattern of the holes (locations from where material is to be removed) are usually left to the intuition of the design engineer. Time constraints typically do not permit multiple iterations which can result in non-optimized designs. In an attempt to aid the designer with the conceptual design stage, this investigation uses a commercially available finite element solver ANSYS for the form finding of some benchmark structure in the literature. Shape and topology optimization by ANSYS based Optimality Criterion was reviewed as a tool to converge on the ideal hole configuration. Through this paper, we emphasize that topology optimization is a very important and the relatively toughest part of the design optimization studies. Therefore, there appears the need of studying topology optimization separately. No amount of sizing and shaping optimization can rectify mistakes committed in finding optimal distribution of material in the design domain (topology optimization).

V. CONCLUSIONS

Following conclusions can be drawn from the above studies:
1. The results of ANSYS based Optimality Criterion which is a gradient based method are compared with those obtained by Element Exchange Method which is a non-gradient based method. The non-gradient based methods guarantee a globally optimal solution, which implies that the results obtained by ANSYS based Optimality Criterion are also global.
2. Compliance values obtained by ANSYS based Optimality Criterion are lower by **1.5** to **13 %** than the Element Exchange Method, Genetic Algorithm and Particle Swarm Optimization in the work of **Rouhi *et al.* [2010]**, except in the case of model 4, when the mesh size is 20, 47. Moreover, it takes lesser number of iterations to reach optimal results as compared with the Element Exchange Method, Genetic Algorithm and Particle Swarm Optimization.
3. As the mesh density is increased there is a **2-3%** decrease in the compliance values for every model.
4. As compared to the EEM, Optimality Criterion approach provides symmetrical results for model 5.

VI. FUTURE SCOPE

Topology optimization being the primary stage of structural optimization, the above plane stress structures can be considered for shape optimization and sizing optimization. In shape optimization, the design variables can be considered to be the coordinates of the nodes and in sizing optimization, any physical dimensions, such as thickness etc can be considered. The objective variable in both cases can be the volume of the structure.

REFERENCES

[1] M. P. Bendsøe, and N. Kikuchi, (1988) "Generating optimal topologies in structural design using a homogenization method" *Comput. Meth. Appl. Mech. Eng.*, vol: 71: 197-224.
[2] J. Thomsen, (1992) "Topology optimization of structures composed of one or two materials" *Struct. Multidisc. Optim.*, vol: 5: 108-115
[3] C. D. Chapman, (1994) "Structural topology optimization via the genetic algorithm", Thesis, M. S. Massachusetts Institute of Technology, America.
[4] A. Diaz and O. Sigmund, (1995), "Checkerboard patterns in layout optimization" *Struct. Optim.*. Vol: 10: 40-45
[5] C. C. Swan and I. Kosaka, (1997) "Voigt-Reuss topology optimization for structures with linear elastic material behaviors", Int. *J. Numer. Meth. In Eng.* Vol: 40: 3033-3057
[6] O. Sigmund and J. Petersson, (1998) "Numerical instabilities in topology optimization: A survey on procedures dealing with checkerboards, mesh-dependencies and local minima", *Struct. Optim.*. Vol 16: 68-75
[7] D. Tcherniak and O. Sigmund, "A web-based topology optimization program" *Struct. Multidisc. Optim. Springer-Verlag* 2001, Vol 22: 179-187
[8] G. Allaire, F. Jouve and A. M. Toader, (2002) "A level set method for shape optimization" *C. R. Acad. Sci. Paris.*
[9] S. F. Rahmatalla and C. C. Swan, (2004) "A Q4/Q4 continuum structural topology optimization implementation", *Struct. Multidisc. Optim. Springer-Verlag*, Vol 27: 130-135
[10] J. Du and N. Olhoff, (2005) Topology optimization of continuum structures with respect to simple and multiple Eigen-frequencies. *6th World Congr. Struct. Multidisc. Optim. Brazil*,.
[11] O. Sigmund and P. M. Clausen, (2007) "Topology optimization using a mixed formulation: An alternative way to solve pressure load problems" *Comput. Meth. Appl. Mech. Eng.*, Vol 196: 1874-1889
[12] G. I. N. Rozvany, (2008) "A critical review of established methods of structural topology optimization." *Struct. Multidisc. Optim. Springer-Verlag*.
[13] Stuttgart Research Centre for Simulation Technology (SRC SimTech), Stuttgart University. (2008) "A new adaptive penalization scheme for topology optimization" A. Dadalau, A. Hafla, and A. Verl.
[14] Thomas R. Michael, (2010) "Shape and topology optimization of brackets using level set method", An Engineering project submitted to the graduate faculty of Rensselaer Polytechnic Institute in partial

fulfillment of the degree of Master of Engineering in Mechanical Engineering. Rensselaer Polytechnic Institute Hartford, Connecticut,

[15] M. Rouhi, R. R. Masood and T. N. Williams, (2010) "Element exchange method for topology optimization" *Struct. Multidisc. Optim.*

[16] Dheeraj Gunwant and Anadi Misra, (2012), "Topology Optimization of sheet metal brackets using ANSYS". *MIT Int. J. of Mech. Eng.,* Vol. 2. No. 2, Aug. 2012. pp 119-125.

AUTHORS

Dheeraj Gunwant obtained his bachelor's degree (B. Tech.) in Mechanical Engineering from Graphic Era Institute of Technology, Dehradun, Uttarakhand, in the year 2008 and M. Tech. in Design and Production Engineering from G. B. Pant University of Agriculture and Technology, Pantnagar, Uttarakhand in the year 2012. He is currently working as Assistant Professor in the Mechanical Engineering department of Apex Institute of Technology, Rampur, U. P. His areas of interest are optimization and finite element analysis.

Anadi Misra obtained his Bachelor's, Master's and doctoral degrees in Mechanical Engineering from G. B. Pant University of Agriculture and Technology, Pantnagar, Uttarakhand, with a specialization in Design and Production Engineering. He has a total research and teaching experience of 25 years. He is currently working as professor in the Mechanical Engineering department of College of Technology, G. B. Pant University of Agriculture and Technology, Pantnagar and has a vast experience of guiding M. Tech. and Ph. D. students.