International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 08 | Aug - 2024 SJIF Rating: 8.448 ISSN: 2582-3930

Topology Optimization of Different Shapes

Mr.Patil G.B.,Mr. Dandge A.G., Mr. Jadhavar K.S.

Mechanical & RIT Polytechnic

---***---

Abstract - Topology optimization is a relatively new but rapidly expanding field of structural optimization. Topology optimization is used in an increasing rate by automobile and aerospace industries. The reason for this is that it often achieves greater savings in material and design improvements. It solves the basic engineering problem of distributing a limited amount of material in a design space, where a certain objective function has to be optimized. A common objective in static problems is to minimize the compliance of a structure by a change in the distribution of mass. This paper includes detailed procedure of topology optimization of mechanical system or component. Various software's are available for topology optimization, Optistruct is one of the useful software which is widely used for topology optimization is explained in brief. Also, case study of topology optimization of control arm is included.

***Key Words*:**optimization,topology,optistruct,distributing,

1.INTRODUCTION

1.1: Background:

In the past decades, there have been dramatic improvements in the engineering design process due to the widespread use of finite element analysis (FEA) as the computational tool and the increases in computer speed. Structural optimization is an important field of research due to its contribution to cost, material and time saving in engineering design. Among various structural optimization problems, topology optimization has deserved great attention. The increasing demand from the industry for lightweight, high-performance and low-cost structures drives the considerable current research going on in the field of structural optimization. Topology optimization is a relatively new but rapidly expanding field of structural optimization. Topology optimization is used in an increasing rate by automobile and aerospace industries. The reason for this is that it often achieves greater savings in material and design improvements.

1.2: Basic Terminology:

a) Optimization:

Optimization is a process of converging onto a final solution amongst a number of possible options, such that certain requirements are best satisfied.

The objective of the optimization problem is often some sort of maximization or minimization, for example minimization of required time or maximization of stiffness. To be able to find the optimum solution the goodness of a solution depending on a particular set of design variables needs to be expressed with a numerical value. This is typically done with a function of the design variables known as the cost function. Mathematically the general optimization problem is most often formulated as minimization of the cost function (which can easily be transformed to maximization by minimizing the negative function) subject to constraints.

Mathematically one can pose a generic problem as follows:

$$\min_{\rho} \int_{\Omega} \phi(\rho)\, \mathrm{d}\Omega$$

Subject to: Design Constraints

Where,

Ω - Design space

ρ - Design variable

Ø -Objective function

b) Objective function:

This is the goal of the optimization study which is to be minimized over the selection field.

c) Design Variables: System parameters that are varied to optimize system performance.

d) Design Space:

Design space is the allowable volume within which the design can exist.

e) Response:

Results that optimizer need to track for minimizing objective function.
Ex. Mass, volume, frequency, displacement, stress, strain, force, etc.

f) Feasible Design: Design that satisfies all the constraints.

g) Infeasible Design: Design that violates one or more constraints.

h) Optimum Design:

Set of design variables along with the minimized (or maximized) objective function and satisfy all the constraints.

i) Design constraints:

These are design criteria that need to satisfy. These could include displacement constraints, stress constraints etc.

j) Manufacturing constraints:

Extrusion constraint: Forces the design to have the same cross-section in a specified direction, or along a curve.

Draw direction constraint: Makes the design not have any cavities in one direction to make it possible to manufacture the component by casting.

Symmetry constraint:

Symmetry in 1, 2 or 3 planes or cyclic symmetry (rotational)

Maximum member size:

It is the largest dimension allowed in the topology design.

Chapter 2

Topology Optimization

2.1: Introduction:

Definition: It is a mathematical approach that optimizes material layout within a given design space for a given set of loads and boundary conditions.

The topology optimization problem solves the basic engineering problem of distributing a limited amount of material in a design space, where a certain objective function has to be optimized. In the case where the design domain is subjected to any static loads, we speak of topology optimization of static problems. A common objective in static problems is to minimize the compliance (maximum global stiffness).Topology optimization has been shown to be an extremely powerful tool in generating efficient design concepts in the early stage of a design process. Topology optimization is used at the concept level of the design process to arrive at a conceptual design proposal that is then fine tuned for performance and manufacturability. This replaces time consuming and costly design iterations and hence reduces design development time and overall cost while improving design performance.

In some cases, proposals from a topology optimization, although optimal, may be expensive or infeasible to manufacture. These challenges can be overcome through the use of manufacturing constraints in the topology optimization problem formulation. Using manufacturing constraints, the optimization yields engineering designs that would satisfy practical manufacturing requirements. In some cases, Additive manufacturing technologies are used to manufacture complex optimized shapes that would otherwise need manufacturing constraints. In the last decade the use of commercial optimization software for topology optimization in the industry has increased rapidly and has shown to be applicable to many different types of problems. The idea is that the designer and the structural engineer both are involved in the initial stage when the first design proposal is conceived. Topology optimization is used to generate a good design concept.

From a given design domain, the purpose is to find the optimum distribution of material and voids. To solve this problem, it is discretized by using the finite element method (FEM) and dividing the design domain into discrete elements (mesh). The resulting problem is then solved using optimization methods to find which elements that are material and which are not. The main solution strategy for solving the Topology optimization problem is the density method. Known quantities in the Topology Optimization problems are applied loads, possible support conditions, volume of the structure, design restrictions such as the location and size of prescribed holes or solid areas.

2.2: Problem statement:

The objective of most common topology optimization problems is to find the minimum compliance (C) of a structure by a change in the distribution of mass or, in a fixed geometry (volume), the distribution of densities. The objective function can therefore be defined as

$$C(x) = f^T u$$

This compliance is the scalar product of the two vectors and resembles the work done by the force vector along the calculated displacements. Thus the given expression is actually a potential similar to common formulation for energy equilibrium.

Thereby the force vector f is equal to the resulting displacements of the finite elements analysis multiplied by the stiffness matrix with the current density distribution
f = K(x

With some further transformation the objective function can be written as

$$\min : c(\mathbf{x}) = \sum_{e=1}^{N} \mathbf{u}_e^{\ T} \mathbf{k}_e(x_e) \mathbf{u}_e$$

The compliance here is a linear combination of the compliances of each element. Since it is a normalized value, the design variable can only range between the values 0 (void) and 1(solid) and therefore has to be restricted. For prevention of possible singularities in the system's matrices the densities are not restricted by zero but by a lower bound. Also, since this optimization method is basically a redistribution of material, the mass has to be constrained.

2.3: Flowchart:

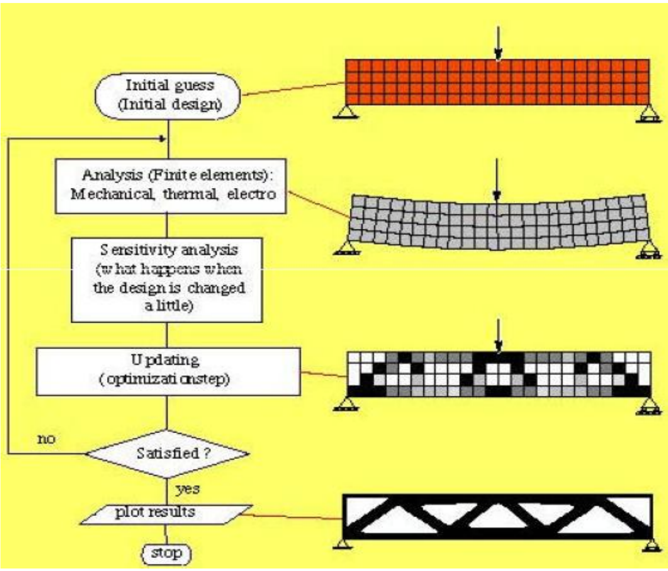

Fig.2.1 Flow chart of topology optimization process

Make initial design, e.g., homogeneous distribution of material. For this distribution of density, compute by the finite element method the resulting displacements and strains. Compute the compliance of this design. If only marginal improvement (in compliance) over last design, stop the iterations else continue. For detailed studies stop when necessary conditions of optimality are satisfied. Compute and update of the density variable. Repeat the iteration loop. Post processing of results.

2.4: Advantages:

- It gives the best concept design that meets the design requirement.
- It reduces overall cost while improving design performance.
- Reduces weight of component or structure
- Improve mechanical performance of component or system.
- Material is distributed throughout the structure in an efficient manner.
- Maximize stiffness.

2.5: Topography Optimization:

Topography optimization is an advanced form of shape optimization in which a design region for a given part is defined and a pattern of shape variable-based reinforcements within that region is generated using optistruct.

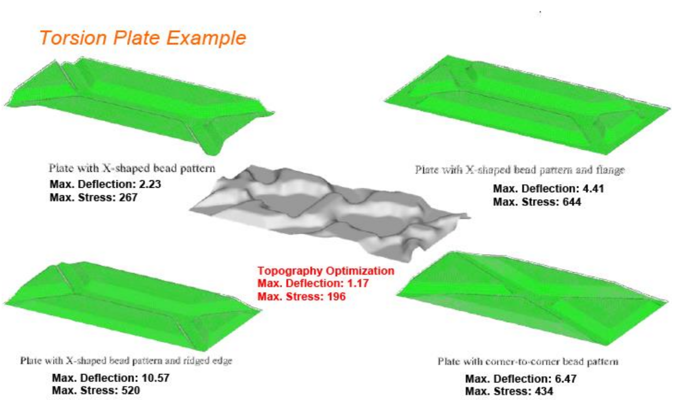

Fig.2.2 Torsion plate example

Moulded Pressure Tank:

Topography optimization of molded pressure tank is shown in fig. Maximum deflection varies as we change surface pattern. We should notice that more rib does not necessarily mean more stiffness.

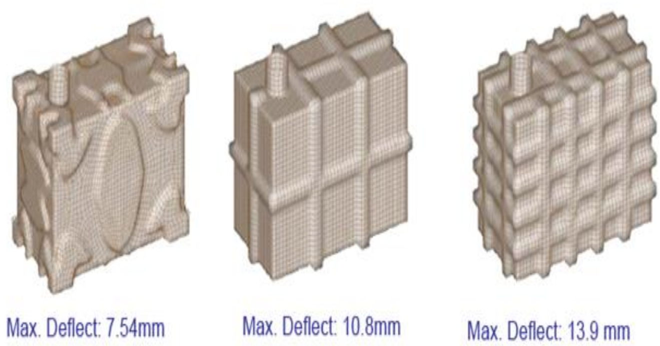

Fig.2.3 Moulded Pressure Tank

Chapter 3

Methodology of optimization

3.1: Overview of the design process:

Here is a simplified overview of the traditional design process and where the different types of structural optimizations can be applied. There are basically two different areas where structural optimization should be performed; in the early design phase where topology optimization is used to generate a good concept and in the detailed design phase where size- and shape-optimization is used to further improve the structure. The optimization methodology can be applied to a variety of mechanical problems including solid structures, shell structures and truss structures. Problems were it should be used with caution due to small outlooks to improvement and good results are:

- For substructures that in means of mass and stiffness have small impact on the whole structure. The changes in properties due to chances of the substructure must be measurable in order to get a working optimization.

- For structures with obscure load cases and requirements. Optimization designs the structure for the specified problem only which means that specifications must be known to some extent.

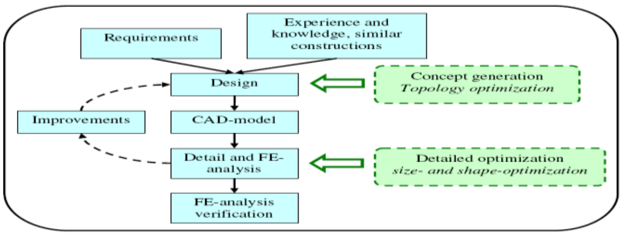

Fig. 3.1 Overview of the design process

One important principle that applies to both concept generation and detailed optimization is that many different optimizations should be performed. It is not possible give guidelines that will yield a good result in just one optimization. Instead multiple optimizations must be performed and then evaluated to possibly refine the problem in an iterative way to get the best result. In the beginning the requirements and specifications should be as relaxed as possible to not constrain the optimization too much and risk missing possible easy design improvements.

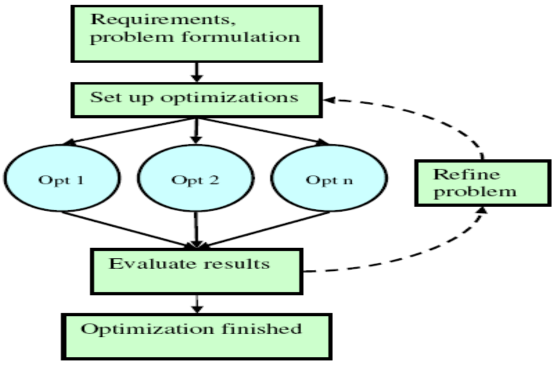

Fig.3.2 Optimization cycle

3.2: Concept generation:

A flow chart for the concept generation is presented followed by a deeper description of each step. Chose of parameters and other details are discussed in the respective sections. The process of concept generation should be seen as an iterative process where the problem formulation and design domain is incrementally improved until the best possible solution is found.

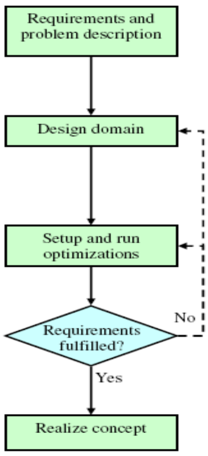

Fig.3.3 Concept generation process

Requirements on geometrical interface, loads, mechanical responses, manufacturability etc. are specified. A design domain is generated from geometrical requirements. This step may have to be redone. The design is discretized. Loads and constraints are specified. Design domain, optimization parameters, objective function and optimization constraints are specified. Topology and/ or free size optimization is run. The topologies are analyzed to see if it fulfills the requirements and if it is possible to realize in a concept. A topology is chosen and a concept is interpreted from it.

3.3: Requirements and problem description:

In this introducing step all requirements for the final product are presented and translated to numerical requirements that can be introduced in the optimization work. Load cases, constraints and modeling aspects are also defined. The objective of the work is presented, e.g. minimization of mass or maximization of Eigen frequencies.

3.3.1: Geometrical requirements:

The geometrical interface is specified in terms of

Largest allowed design Volume	Define the largest allowed domain (volume or area) in which the final structure must be formed.
Predefined parts	Define position and shape of parts that must not be changed by the optimization. E.g. contact surfaces connections to other structures etc.
Parts that must be present	Define what parts that must be present in the final structure but can be placed according to the optimization. E.g. Screw holes, holes for cables, connections to other structures etc.
Symmetry	Is there any type of symmetry or pattern repetition that is desired?
Accessibility for mounting	What points of the structure must be accessible for mounting or other requirements. They have to be defined so that voids can be placed at those positions in the design domain.

3.3.2: Load cases:

Define loads for the structure.

Static loads	Define magnitude, direction and position for forces as well as the corresponding constraints. The right relation between the loads is more important than the absolute magnitudes. The structure may be arbitrarily bad for load cases that are not introduced.

3.3.3: Mechanical requirements:

Define requirements on masses and mechanical responses.

Mass	Define the largest/smallest allowed mass of the final structure
Stress	Define the largest allowed stresses of the final structure, any safety limit?
Displacement	Define the largest/smallest allowed displacement at different parts of the structure if necessary.

3.3.4: Manufacturability requirements:

What method will be used for manufacturing? What are the thinnest/thickest allowed material thicknesses?

Extrusion	What direction?
Casting	Single or split? Directions?
Material thickness	Minimum and maximum member size?

3.4: Objective:

Define the objective of the conceptual design and what type of result that shall be considered as a success.

3.5: Design domain:

The design domain is created, preferably by a designer and an analysis engineer in cooperation. The idea is to start with a simple domain leaving much for the optimization to do and successively refine it after optimizations. In this way the chances to reach a global optimum are increased and the concept will be more optimal. The design domain is preferably made as a CAD-model.

Set up the design domain using the following tips

First design domain	Construct a design domain using the geometrical requirements. Design the predefined parts, fill up the largest allowed design volume and introduce voids for mounting etc at predefined positions if any.
Refined domain	After initial optimizations parts that must be present in the final structure but don't have predefined positions are introduced. Use conclusions from the optimizations to choose the best positions. Construct additional voids for mounting. If some part of the design domain obviously is unnecessary it could be removed to make computations faster.
Partition the design Domain	The design domain may be partitioned and parts can be left out of the calculations. E.g. if a structure is to be placed on a plate which is to be fastened in something; the plate can be excluded and the structure supposed to be fixed in the former contact surface to the plate. The plate is then introduced later when the other parts of the structure are ready. This method is used when it is hard to fulfill manufacturability constraints.

3.6: Discretize the model and define load cases:

Mesh th e model with an element size of a third of the desired minimum member size. This will ensure a mesh independent solution later on. Complete the model and connect it to possible adjacent structures. Define the loads, constraints and load cases the structure shall be subjected to. When the design domain has been refined and shall be optimized again the mesh size can be slightly shifted to study if the analysis is mesh independent. When the model is finished a test run is performed to study if

- There is a possibility to reach a successful result with the optimization; that the requirements are not unreasonable.
- The model is built up properly; that elements are connected correctly etc., a modal analysis is good to spot errors.

3.7: Parameters:

A number of parameters that influence the optimization result can be activated and varied to achieve better results. The parameters and their effects are described below.

MATINIT	The initial material distribution.
OBJTOL	The convergence criteria. Default is 0.005, lower leads to more iterations. Is most often OK with the default value.
DESMAX	Maximum number of iterations. Default is 30 (80 when MINDIM is used). Default is most often sufficient; if max number of iterations is reached and the solution appears to converge, DESMAX should be increased.
SCREEN	A control card that should be used to output information about the iterations to the command window.
DISCRETE	Controls the penalization of intermediate densities, a value of 2 is recommended for 2D-elements and 3 for 3D-elements.

3.8: Responses

A number of different responses that can be used both as objective function and constraint are available and can be seen below

Compliance	Typical as objective function, inverse of the stiffness of the structure
Mass	

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 08 | Aug - 2024 SJIF Rating: 8.448 ISSN: 2582-3930

Volume	
Static displacement	Displacement in one node, in one direction or the total distance. If the acting force is applied in the measured node, minimization of displacement is very similar to minimization of compliance
Von Mises stress	

Objective:

The objective function is defined by choosing a response to either maximize or minimize. Exactly one objective function must be created.

3.9: Constraints:

Constrains are defined by setting upper and/or lower bounds on a response. Multiple constraints can be created. Instead of using the available stress constraint it is recommended to perform multiple optimizations with the objective of minimizing the compliance together with different upper bounds on the volume fraction. From this the stresses for each volume fraction is studied manually and a volume fraction that gives reasonable stresses is chosen.

3.10: Criterion of success:

The properties of the resulting topologies are hard to measure in absolute numbers. The mass and other properties of the final structure will most probably differ from those of the structure at this stage. Therefore a limit of safety should be used on Eigen frequencies, displacements, etc. Topology optimization will most likely give a structure with high stresses; however, such matters are designed for in the detailed design phase. A structure that is considered as a candidate to continue with should fulfill the following requirements:

- Any demands on Eigen frequencies or displacement etc. are fulfilled with a limit of safety
- The topology should be able to realize in a concept without any drastic changes in topology
- The topology has been proven to be a global optimum
- The topology has shown to be robust by using variation of load cases
- The overall behavior and shape of the topology seems reasonable

3.11: Interpret the results:

One of the most difficult tasks when performing the topology optimization is to interpret the results. By setting up the optimization problem in a good way this step can be made easier but it is still far from unambiguous. The designer and the analysis engineer should at this stage work together so that aspects such as manufacturing are taken into account and sound simplifications are made. The results are post processed in Hyper View.

Chapter 4

Overview of Used Tools

4.1: Work flow in Hyper works:

Here the most important software used is briefly described. The main program used for performing finite element analyses and optimizations is the solver Optistruct 10.0 from Altair Engineering. There is also other available software for design optimization such as Top opt, Tosca, etc. To be able to set up the problem and review the results Hyper Mesh and Hyper View are also used. Hyper Mesh is the preprocessor which is used to discretize (mesh) a CAD model, set boundary conditions, properties and options and to set up the problem to be solved (optimization, static analysis, modal analysis etc.). From Hyper Mesh a file which completely describes the problem is exported and then processed using Optistruct. The results from Optistruct can then be evaluated using the postprocessor Hyper View. A schematic overview of the workflow can be seen in Fig. Hyper Mesh, Optistruct and Hyper View are all part of the software suite Hyper works 10.0, and as such they are designed to easily integrate with each other.

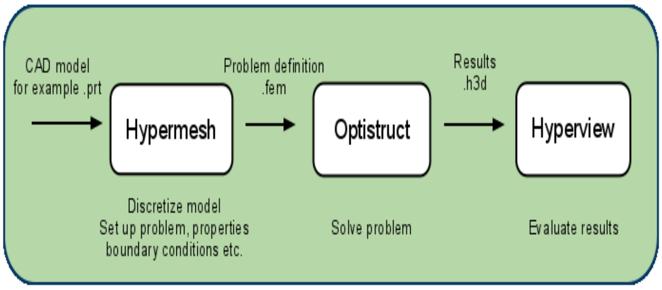

Fig.4.1 An overview of the workflow in Hyper works

4.2: Optistruct:

Optistruct is the solver used for performing Topology optimization. The only problem solved was the minimization of weighted compliance and/or Eigen frequencies using the homogenization method. In 1993 the first commercial version was marketed as Altair Optistruct 1.0. The current version of Optistruct is 11.0

Optistruct is capable of performing a range of different finite element analyses including static, modal, buckling and thermal analyses. Different types of loads such as point forces, pressure, gravitational loads, thermal loads, etc., can be applied. When setting up the FE model many different types of elements are supported including different types of three-dimensional solid elements, two-dimensional shell elements and other types of elements such as beams, bars, springs and point masses.

4.3: Steps for Topology Optimization in Optistruct:

- Create a Finite Element model
- Define design variables and constraints on the design variable
- Define responses that will use as objective or constraints
- Formulate optimization objective
- If desired, set constraints on responses
- Run optimization in optistruct
- Interpret the results

4.4: How Optistruct work:

First discretize the domain into a finite element mesh. Optistruct calculates material properties for each element Optistruct solves topological optimization problems using the density method under topology optimization, the material density of each element should take a value of either 0 or 1.Defining the element as being either void or solid and interpret the result

4.5: Advantages of Optistruct:

- It solves the most complex optimization problem in significantly less time.
- It capable of performing range of finite element analysis.
- It generates reliable and highly accurate result.

Chapter 5
Case Study
Topology Optimization of Control arm:
5.1: Introduction:

In this case study topology optimization using draw direction constraints on a control arm is explained. The finite element mesh containing designable (brown) and non-designable regions (blue) is shown in Fig.

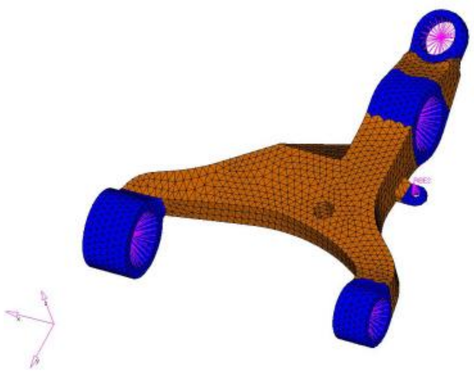

Fig.5.1 Control arm schematic.

5.2: Steps of Topology Optimization of Control arm:

Step 1: Launch Hyper Mesh and Load the User Profile.

Step 2: Retrieve the controlarm.hm file

Step 3: Create Design Variables for Topology Optimization with Draw Direction constraints

The draw direction constraints allow the casting feasibility of the design so that the topology determined will allow the die to slide in a given direction. These constraints are defined using the DTPL card. Two DRAW options are available. The option 'SINGLE' assumes that a single die will be used. The option 'SPLIT' assumes that two dies splitting apart in the given draw direction will be used to cast the part.

1. From the Analysis page, select the optimization panel.
2. Select the topology panel.
3. Make sure the create radio button is selected on the left hand side of the panel.
4. Enter a name in desvar = dv1.
5. Click props and select Design.
6. Click select.
7. Set the type: toggle to PSOLID.
8. Click create.
9. Click the draw radio button, and toggle the draw type: to single.

The option 'SINGLE' assumes that a single die will be used and it slides in the given drawing direction. The anchor node and the first node define the drawing direction.

10. Click anchor node.
11. Enter the value 3029 and press ENTER.

This selects node ID 3029.

12. Click first node.
13. Enter the value 4716 and press ENTER.

This selects node ID 4716.

14. Under obstacle, double-click props, select Non-design and click select.

This selects the non-designable parts as obstacles for the casting process on the same DTPL card. This preserves the casting feasibility of the final structure.

15. Click update.
16. Click return to go back to the optimization panel.

Step 4: Create the Optimization Responses

1. Select the responses panel.
2. Enter Volfrac in the response = field.
3. Select the switch below response type: and set it to volumefrac.
4. Click create.
5. Enter Compl in the response = field.
6. Set the response type: to weighted comp (located on the second page, accessed through arrows).
7. Click loadsteps, check the boxes next to both loadcases, and click return.
8. Click create.
9. Click return to go back to the optimization panel.

Step 5: Apply Design Constraints on Volume Fraction Response

1. Click dconstraints to define the constraints.
2. Enter Constr in the constraint = field.
3. Select upper bound = by clicking in the box preceding it.
4. Click in the text box and enter the value 0.3.
5. Click response = and select Volfrac.
6. Click create.

The volume fraction constraint is now created.

7. Click return to go back to the optimization panel.

Step 6: Set Minimize Compliance as Objective

1. Click objective to define the objective function.
2. Click the min/max switch in the upper left corner and select min.
3. Click response and select Compl.
4. Click create.
5. Click return twice.

This completes your optimization problem setup.

Step 7: Save the Database

1. From the File drop-down menu on the toolbar, select Save as.
2. Change the file name to controlarm_opt.hm and click Save.

Step 8: Run OptiStruct

1. From the Analysis page, select the OptiStruct panel.
2. Set the export options: toggle to all.
3. Click the run options: switch and select optimization.
4. Set the memory options: toggle to memory default.
5. Click OptiStruct.

This launches the Opti Struct job. If the job is successful, new result files will be seen in the directory where Hyper Mesh was invoked. The control arm_opt.out file is a good place to look for error messages that will help to debug the input deck if any errors are present.

Post-processing the Results using Hyper View:

Opti Struct provides the Element density information for all of the iterations. OptiStruct will also show Displacement and Von Mises stress results of a linear static analysis for iteration 0 and iteration 38. This section describes how to view those results in Hyper View. First, the Hyper Mesh binary results file needs to be loaded.

Step 9: Contour Plot of Element Densities

1. From the Opti Struct panel, click the Hyper View button.

2. Click close to close the message GUI.
3. Select the last iteration in the Results Browser.
4. Click the Contour panel toolbar button.
5. Under Result type: select Element densities (s) from the drop-down menu.
6. Set the Averaging method: to Simple.
7. Click Apply.

The resulting contours represent the displacement field resulting from the applied loads and boundary conditions. In this model, refining the mesh should provide a more discrete solution; however, for the sake of this tutorial, the current mesh and results are sufficient.

Step 10: Iso Plot of the Densities:

The iso surface feature can be a very useful tool for post-processing density results from Opti Struct. For models with solid design regions, this feature becomes a vital tool for analyzing density results.

1. Click the Iso Value panel toolbar button .
2. Set the Result type: to Element densities (s).
3. Make sure the last iteration is still selected in the Results Browser.
5. Click Apply.
6. Enter a value of 0.3 in the Current value: field.
7. Make sure that Show values: is set to Above.

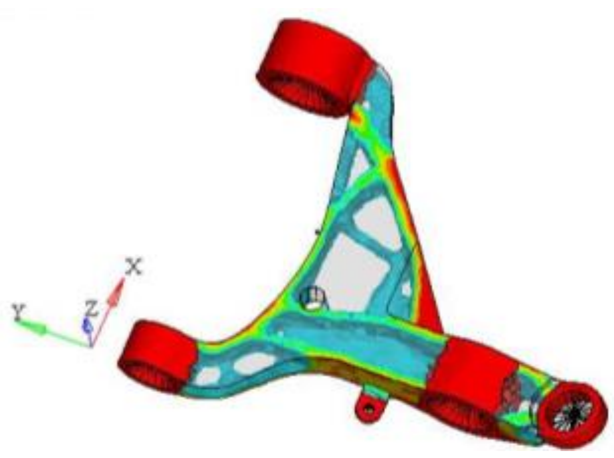

Fig.5.2 Iso surface plot of element densities.

Step 11: Plot a Contour of Displacements and Stresses

1. Click the Next Page arrow to proceed to the results of Load Case1 on page 2.
2. Set the animation mode to Linear Static.
3. Click the Contour panel toolbar button.
4. Set the Result type: to Displacements (v).
5. Click Apply.

This will give a displacement plot for Iteration 0.

6. Click to review the results for the next iteration.

A displacement plot for the last Iteration should be visible.

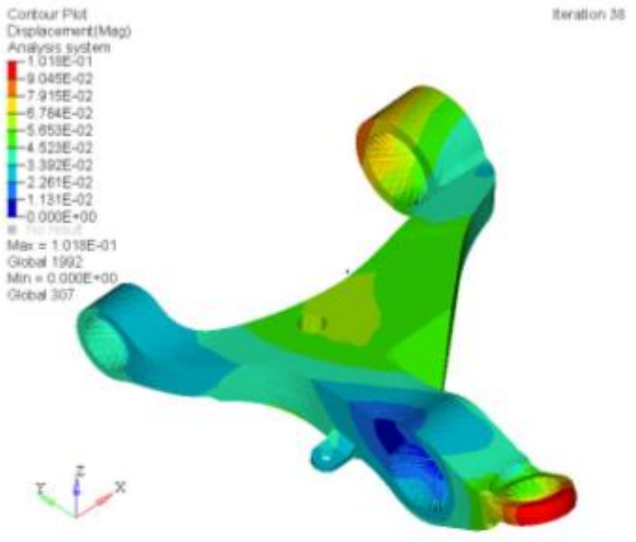

Fig.5.3 Displacement contour for the first load step at the last iteration.

The stress results are also available for the respective iterations.

7. Similarly, results for Load Case 2 can be viewed on page 3 using the Next Page button.

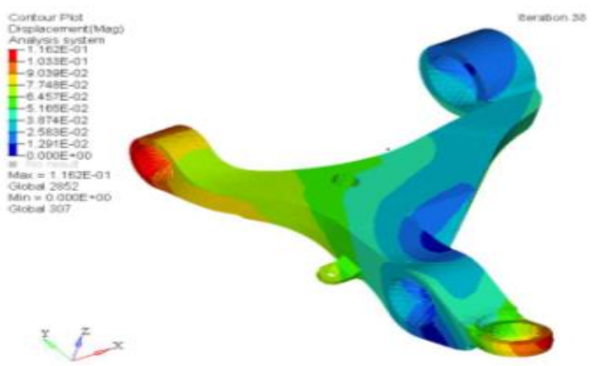

Fig.5.4 Displacement contour for the second load step at the last iteration.

Chapter 6
Conclusion

Topology optimization has been shown to be an extremely powerful tool in generating efficient design concepts in the early stage of a design process**.** It can be applied to ensure that Material is distributed throughout the structure in an efficient manner and maximize stiffness. Topology optimization of mechanical components improves mechanical properties of component or structure and reduces some properties like weight, stress concentration factor. The objective of topology optimization problems is to find the minimum compliance of a structure by a change in the distribution of mass. Topology optimization is used in an increasing rate by automobile and aerospace industries. The reason for this is that it often achieves greater savings in material and design improvements. Opti struct which is used for topology optimization solves the most complex optimization problem in significantly less time. Result obtained from Topology optimization has good quality in terms of accuracy and efficiency.

- Topology optimization ensure that material is distributed throughout the structure in an efficient manner.
- It maximizes stiffness of structure.
- Reduces some properties like weight, stress concentration factor.
- Opti struct which is used for topology optimization solves the most complex optimization problem in significantly less time.
- Result obtained from Topology optimization has good quality in terms of accuracy and efficiency.

References

1] Bendsøe, M.P. and Sigmund, O. (2004).Topology Optimization, Theory, Methods and Applications.2nd edition. Springer-Verlag, Berlin.

2] Achtziger, W (1998). Multiple-load truss topology and sizing optimization: Some properties of minimum compliance, Journal of Optimization Theory and Applications 98(2): 255-280.

3] Buhl, T. (2002). Simultaneous topology optimization of structure and supports, Structural and Multidisciplinary Optimization 23(5): 336-346.

4] Maute, K. & Ramm, E. (1998). Adaptive topology optimization, Structural Optimization 10(2): 100-112.

5]Anton olason and Daniel Tidman (2008) Methodology for Topology and Shape Optimization in the Design Process

6] Altair optistruct–concept design with topology optimization helps manual and tutorials.