Topological Optimization of a Refrigerator Bed

Chandana GN, Suparshwa Keerthi*, Acharya yaska, Prof. Sukesh N and Dr. Girish Kumar

SDM Institute of Ujire, India

*Corresponding author: Suparshwa Keerthi, SDM Institute of Ujire, India, Tel: +91-9480154333; E-mail: keerthisuparshwa@yahoo.com
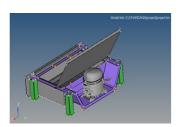
Abstract

This paper deals with optimization of a Refrigerator bed. It describes development of a Finite element model consisting of structural members which are designed using basic principles of structural design. Topological optimization is applied to the bed considering volume as the objective along with Displacement as the constraints. Manufacturing constraints are considered to provide the manufacturability and interpretable design proposal. From the interpretations of Topological optimization, two designs are proposed one for Sheet metal and another for Plastic. The refrigerator bed Structure is optimized to minimize the mass and the cost.

Keywords: Refrigerator; Topological optimization

Introduction

The refrigerator bed plays a vital role by supporting all the components majorly the compressor. The entire static and the dynamic load act on the compressor bed hence the design of the bed should withstand all the forces meanwhile. But most of our design will consist of materials which do not contribute to the strength or rigidity, instead become dead weight. To remove these extra materials Optimization is carried out.

Generally, there are different kinds of optimizations. But for this particular scenario where the material should be removed, reduction of the volume must be the objective with some other parameter as constraint, in this case Displacement. Which means the material is removed until the displacement remains the same. As the result of this a safe design is obtained but with unwanted materials removed.

Figure 1: CAD model of the existing refrigerator bed with compressor.

This paper deals with optimization of a refrigerator Bed (Figure 1). It describes development of a Finite element model and subsequent analysis and simulation of the Bed. Topology and Size optimization is applied to the bed considering volume as the objective along with Displacement as the constraint. Manufacturing constraints are considered to provide the manufacturability and interpretable design proposal. From the interpretations of Topology and Size optimization, two conservative designs are proposed, one for steel and another for plastic.

Problem Statement

To optimize the existing refrigerator (Figure 2) bed by applying different boundary conditions and to achieve a weight reduction and come up with two final designs, one for plastic and another for steel.

Objectives

1. To reduce the weight at least by 20% of the original weight.
2. To check the strength for different thickness. To come up with a prototype that is feasible from both Design point of view and the manufacturer's point of view.
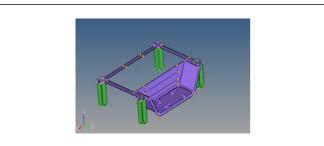3. To reduce the cost of the product.

Figure 2: The compressor bed to be optimized.

Methodology

The existing model of the refrigerator bed is optimized. First the CAD model is meshed using the HYPERMESH with maintaining the Industrial quality criteria. Then it is analyzed using RADIOSS for the displacement and stresses. Then keeping volume as the objective and displacement as the constraint it is optimized to get the material distribution. Several trials are performed and a final design is selected and CAD model of the same is made using UG-NX.

Bench Marking

To be successful, an organization must have a basis for competitive advantages. While an organization meets to do a reasonable job in

various competitive dimensions, it cannot be all things to all people. The enterprise must focus on one or two dimensions of competition to truly excel and be successful. The following are the competitive dimensions typically associated with product development:

Time-To market

Low development cost

Low cost producer/low cost, High value product

Innovation and product performance

Quality, reliability, ease of use, service ability etc.

These are the major things in Bench marking [1].

The aim here is to achieve the third, fourth and the fifth point by decreasing the cost of production, Better Performance Good quality, reliability compared to the existing model and come up with an innovative design.

Geometric model

The existing model comprises of four legs and frames connecting all the legs. The major part in this component is the compressor Bed which holds the compressor on it. The compressor is rigidly mounted on the bed with the help of bushings to isolate the vibration. The entire load both static and dynamic act on this component.

Finite element modelling

The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics, useful for problems with complicated geometries, loadings, and material properties where analytical solutions cannot be obtained. Model body is divided into an equivalent system of many smaller bodies or units (finite elements) one dimensional, two dimensional or three Dimensional (Figure 3) interconnected at points common to two or more elements (nodes or nodal points) and/or boundary lines and/or surfaces. Principles of FEA-The finite element method (FEM), or finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Boundary value problems are also called field problems. The field is the domain of interest and most often represents a physical structure. The field variables are the dependent variables of interest governed by the differential equation. The boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field.
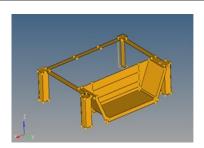

Figure 3: Model body is divided into an equivalent system.

A general procedure for finite element analysis

Preprocessing

Define the geometric domain of the problem.

Define the element type(s) to be used.

Define the material properties of the elements.

Define the geometric properties of the elements (length, area, and the like).

Define the element connectivity (mesh the model).

Define the physical constraints (boundary conditions). Define the loadings.

Solution

Computes the unknown values of the primary field variable(s)

Computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow.

Post processing

Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution [2].

Pre processing

First the model is imported into the software HYPERMESH and the mid-surface of the geometry is extracted (Figure 4).
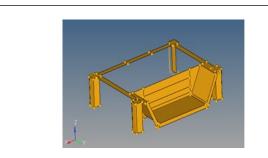

Figure 4: Mid-surface of the refrigerator bed.

This is done because in shell meshing, the 2D meshing is done on the mid-surface and then thickness is given to make it into a 3D element.

Washer split

The washer split is introduced in geometrically critical areas where there is most likely of stress concentration, for example circular holes etc. washer split is introduced and meshed at the beginning so that there will be no compromise in that region and all the other meshes are adjusted.

Meshing

The surface is divided according to our convenience and meshed. Meshing has to be done first in the washer split. Maximum care has to be taken to avoid trias (Figure 5). The total number of trias should not increase more than 15% of the total number of elements.

Figure 5: Meshed component.

Quality check

Warpage (15'): Warpage is distortion of elements when there is curvature the elements are not perfectly along the curves. So it measures the deviation the deviation of an element from its ideal or perfect shape.

Skew (60'): Skew of a triangular element is calculated by finding min angle between the vector from each node to the opposing mid side and the vector between the two adjacent mid sides.

Aspect ratio (5) it is the proportion between the width and the height of picture. It is often expressed in the W: H format (width: height)

Jacobian (.65): This measures the deviation of an element from its ideal shape for e.g. –for a triangle from an equilateral triangle. The Jacobian values ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. The determinant of the Jacobian relates the local stretching of the parametric space which is required to fit Inyo the global coordinate space.

Figure 6: Quality check.

Element size: Very fine meshing results in more number of elements whereas too large elements results in raising from the surface near the fillets, around the hole etc. Therefore normal element size gives a better solution especially near the fillets.

Minimum length of the element (2) - limits how small each mesh element can be

Maximum length of the element (10)-limits how big each mesh element can be

Average length of the element (5)

Angle: An angle measures the amount of turn i.e. the deviation/rotation. Minimum and maximum Angle of the trias (20'-120') - the inside angle Minimum and maximum Angle of the quad (45'-135')

The Meshing done must not violate any of the criteria (Figure 6).

Loads and boundary conditions

The legs of the component is fixed in all directions (i.e. constrained in three translational and three rotational motions) to avoid any imbalance in the component. The total weight of the refrigerator is found to 1500 N which is equally distributed among four legs which takes about 375N each. At the top of each leg, rigids are created so that frame does not deform at that place.
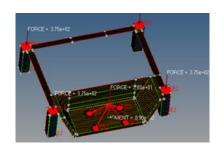

Figure 7: Loads and boundary conditions applied.

Compressor is meshed by using auto mesh to find out the Center of Gravity of the Compressor. After the Center of gravity is found one more rigid is created joining the Center of Gravity and four bushes at the base. The moment and weight of the particular compressor is found out to be 890N/m and 20N respectively. This loads are applied at the Center of Gravity of compressor where rigid is created (Figure 7).

Analysis

Once the loads and the boundary conditions are set, the material properties and the thickness is assigned then the ANALYSIS is carried out using the Radioss solver. The Displacement is noted down as the whole optimization is done on the basis of constraining the displacement.
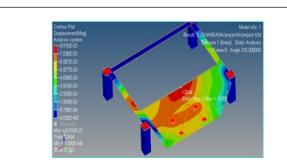

Figure 8: Analysis report showing displacement.

The analysis result showed that plastic had more displacement compared to the steel (Figure 8).

Objective

Based on the result of analysis there are two materials that can be used for the refrigerator bed, Steel and plastic. So we came up with the following modules and select the best among those.

Steel

Entire sheet Metal of body Thickness 1.5 mm and leg 5 mm.

Entire Sheet metal of body Thickness 2 mm and leg 5 mm.

Plastic

Plastic body Thickness 2 mm and steel legs 5 mm.

Plastic body thickness 3 mm and steel legs 5 mm.

Optimization

Optimization is the selection of a best element from some set of available alternatives. A part can be modified to meet requirements on stress, weight, reliability etc. Through optimization where optimum designs are developed in shorter design cycles. Topological optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets. It is implemented through the use of FEM FOR THE ANALYSIS, and the optimization techniques. The best use of material in this case represents the "maximum-stiffness" design Uses Solves the problem of distributing a given amount of material in a design domain subject to load and support conditions, such that the stiffness of the structure is maximized. Improve design quality which is important in the present world Goal-the objective function-is to minimize the energy of structural compliance while satisfying a constraint on the volume (V) of the structure [3,4].

Topology

The first step in topological optimization is to set the design variables. The region to be optimized is selected. The minimum member size of the element is selected. The pattern of optimization is also selected which will help in the design process later. Basically in this we are setting the expected design on the optimized model.

Responses

The responses have to be created for the parameters that have to vary in the optimization like mass or volume and also to the parameters that has to remain constant or be the constraints. We have selected Volume as the parameter to change and displacement as the constraint.

Constraints

As said earlier one of the parameter must remain as the constraint which acts as the guiding parameter. Here displacement is the constraint and the upper and lower boundary plus or minus .5 the value of the displacement was selected.

Objective

The objective of the optimization is to minimize the volume keeping displacement constraint.

Case study

Case I:

The optimization was carried out with considering both static and dynamic forces. The minimum member size was 15mm and no pattern

grouping was selected. Figure 9 shows the optimization result for case I.
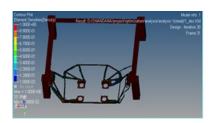

Figure 9: Optimized bed with dynamic force.

Result for case I

Material removal was asymmetric due to no pattern grouping

Many trusses were formed due to min size of the element given was 12

Hence this method is not feasible from the manufacturer's point of view.

Case II:

The analysis was carried out for plastic of thickness 2 mm.The analysis showed a maximum displacement of 2.42 mm at the back panel of the component. This model was not optimized as displacement was maximum hence this was discarded. The analysis results are shown in Figure 10.
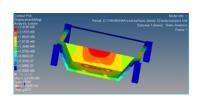

Figure 10: Analysis result for plastic body with thickness 3mm.

Case III:

When sheet metal of thickness 1.5 mm was considered for analysis the displacement was found to be3.93E-2 mm. As this displacement is very small this model was optimized and result showed lot of trusses at some places which cannot be manufactured in case of steel but there was also scope for bulk material removal at some locations .This result is shown in Figure 11.
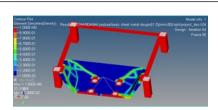

Figure 11: Optimization result for sheet metal thickness 1.5 mm.

Case IV:

When sheet metal of thickness 2mm was analyzed it showed a displacement of 1.75E-02 mm. As the displacement can be neglected for steel model was optimized and there was a good pattern of material which was symmetric also. The results are shown in Figure 12.
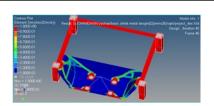

Figure 12: Optimization result sheet metal thickness 2 mm.

Case V:

The analysis was carried out for plastic of thickness 3mm. The analysis showed a maximum displacement of 1.1 mm at the back panel of the component. This model was optimized as displacement was acceptable. The material removal pattern showed trusses but it can be molded in case of plastic the analysis results are shown in Figure 13.
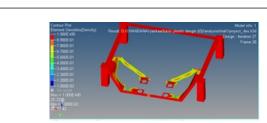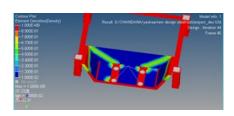

Figure 13: Optimization result for plastic thickness 3mm.

Case VI:

By looking at all the results in sheet metal 2 mm design is better. So some changes was made in the non-design area and optimized again for the support in the structural member. This optimization result is shown in Figure 14.
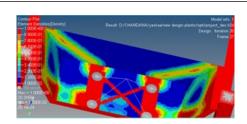

Figure 14: Optimization result for 2 mm sheet metal with change in design.

Case VII:

By looking at all the results in plastic, 3mm design is better. So some changes was made in the non-design area and optimized again for the

support in the structural member. This optimization result is shown in Figure 15.

Figure 15: Optimization result for 3 mm plastic with change in design.

Conclusion

When optimization was done for two different materials i.e. steel and plastics different patterns of material removal was found. The maximum displacement was also different for two materials. This is because that both materials differ in their material density and their load bearing capacity.
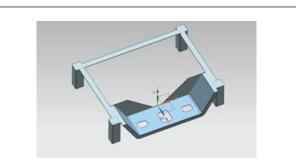

Figure 16: Sheet metal model.

When optimization was carried out for steel the least displacement was found to be 1.75E-02 for material thickness 2 mm. The material removal was found to be almost uniform and material removal was symmetric (i.e. case 4). But due to manufactures point of view other models (case 1, 2, and 3) were discarded as it consisted of many trusses. Later on CAED model was developed for the same which is shown in Figure 16.
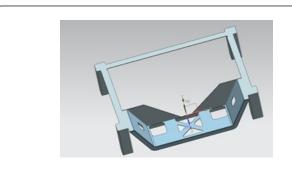

Figure 17: Plastic model.

When optimization was carried out for plastic the least displacement was found to be 1.1 mm for material thickness 3mm. The

material removal was found to be almost uniform and material removal was symmetric (i.e. Case V). Even though there was lot of trusses it could be manufactures as in case of plastic. Later on CAED model was developed for the same which is shown in Figure 17.

References

1. Crow K (1999) Bench marking best practices to improve product development.

2. Hutton DV (2015) Introduction to Finite Element Analysis (FEA) or Finite Element Method (FEM)

3. Goelke M (2016) The academic training center (ALTAIR) training(14th edn)

4. Ansari NS, Ansari N, Kulkarni V (2013) Structural Optimization of Drive Unit Bracket Using OptiStruct. Altair Technology Conference, India.