International Journal of Engineering Research and Reviews ISSN 2348-697X (Online)
Vol. 3, Issue 1, pp: (136-144), Month: January - March 2015, Available at: www.researchpublish.com

Finite Element Analysis and Optimization of Refrigerator Structure

Kesani Dinesh

PG Student, Department of mechanical Engineering, Bharath University, Chennai-India

***Abstract*: In modern industries, commercial refrigerators play a major role in preserving food groceries, beverages, etc. Hence designing and analysis of these refrigerators to withstand given load has become a major concern in recent years. The objective of this dissertation was to analse and optimize the refrigerator shelf and to find maximum stress and deflections of entire refrigerator assembly for given loading and boundary conditions using Finite Element Method. First stage consists of Analysis and optimization of refrigerator shelf. Second stage consists of the Analysis of entire refrigerator assembly. While lifting the structure from one place to another place, structure has to be tilted to a small angle so that the structure sits firmly on the automated guided vehicle. Hence it is very important to find the critical angle of tilt for safe transportation and thereby preventing from a huge loss. Finally, Toppling analysis of entire refrigerator assembly was carried out to find the critical angle of tilt. In this critique, Pro/Engineer 3D CAD models of refrigerator shelf and entire Refrigerator assembly were imported to Hypermesh as an IGES format. The meshed finite element model with given loading and boundary conditions were exported to Finite Element Solvers (NASTRAN, ANSYS) to determine the maximum displacement, maximum stress and deflections at specified locations. Finally the model of refrigerator structure was exported to motion view in H3D format to carry out toppling analysis.**

***Keywords:* FEA, CAD, MPA, DOf.**

1. INTRODUCTION

OBJECETIVE:
Main objectives of this project are Static analysis of refrigerator shelf assembly to find maximum displacement. Design optimization of refrigerator shelf components for minimum mass of shelf within the allowable maximum deflection limits. Static analysis of entire refrigerator assembly to find the maximum displacement, maximum stress and deflections at specified locations. Toppling Analysis of entire refrigerator assembly to find the critical angle of tilt.

2. LITERATURE SURVEY

STUDIES RELATED TO PREVIOUS WORK:
Refrigerators are commonly used devices in preserving beverages, medicines, food items, etc. Until recently, the primary analysis had been hand calculations and empirical curves. New computer advances have made Finite Element Analysis (FEA) a practical tool in the study of static analysis, especially in determining stresses in local areas which are difficult to analyze by hand. Finite Element Analysis is an extremely powerful tool when employed correctly. Depending on the desired solutions, there are different methods that offer faster run times and less error.

While Finite Element Analysis offers another way to analyze structures, it requires an understanding of the program and the subject being modeled. If the operator does not use the correct model, time is wasted and more importantly, data is useless. Initially, finite element analysis was used in aerospace structural engineering. The technique has since been applied to nearly every engineering discipline, from fluid dynamics to electro-magnetic. The difficulty in analysis of stress and strain in structural engineering depends on the structure involved. As the structure grows in complexity, so does the analysis. Many of the more commonly used structures in engineering have simplified calculations to approximate stress and strain. However, these calculations often provide solutions only for the maximum stress and strain at certain points in the structure.

(i) Giridhar Kumar, Uma Maheswari, Abhijit Kulkarni & Shyam T. Gopal, [Global Technology and Engineering Centre, Whirlpool of India, Pune] in their paper "FEA SIMULATION OF REFRIGERATOR CABINET DEFLECTION" presented at Ansys India User Conference, Nov-2003 defines that, a refrigerator cabinet is constructed primarily of sheet metal parts with a plastic liner and a foam core. The outer steel is comprised of a wrapper with roll-formed front and rear flanges, corner bracket reinforcements, a back panel, and a deck. The inner plastic liners form the refrigerator and freezer compartments. The inner and outer layers adhere to the polyurethane foam core. The cabinet's support structure is made from formed sheet metal parts: front rail, back rail, channels and glider rails. The design of the cabinet is constrained by the need to be assembled on a high-speed production line as well as by aesthetic requirements.

The first step in the analysis was to build a detailed finite element model of the entire cabinet assembly. The geometry of the parts was created in Pro/ENGINEER software and imported in ANSYS.

Regarding to the elements, authors of the technical paper states that, the refrigerator structure is a combination of shell and solid component. Solid 92 was used for solid (thick section) component e.g. PU foam in cabinet & door. Shell 63 was used for shell type (thin section) component e.g. back panel, liner, door hinges, compressor mounting plate, shelves, etc. The cabinet & door structures consist of PU foam in between plastic and steel layers. Foam gets bonded strongly between liner and wrapper. Foam was meshed by solid 92 elements and one of the surfaces sheet metal was meshed by shell63 elements to simulate wrapper and another surface plastic liner was meshed by shell 63 elements to simulate liners.

In experimental setup, loads were placed on shelves which are subsequently resting on ribs, protruding inside the cabinets. Hence shelve loads were applied as nodal force on the corresponding locations of ribs. Similarly loads of door shelves were applied on door protrusions as nodal forces. Compressor mass was converted to weight and equivalent force was applied as nodal force on the respective nodes of compressor location. Boundary conditions were applied on the base of cabinet at all four leg locations restricting in all Dof's.

(ii) Owen, Steven J., Scott A. Canann and Sunil Saigal proposed a method whereby an existing non-conforming, mixed hexahedra-tetrahedra element mesh, is altered to conform by the insertion and formation of five-node or thirteen-node pyramids. Local tetrahedral transformations are performed to provide the topology enabling the merging of two adjacent tetrahedra into one pyramid. Local smoothing and cleanup operations improve the quality of the transition region. Other methods for the creation of transition pyramid elements are also discussed. Results show superior performance of the resulting elements in a commercial finite element code over non- conforming interface conditions.

(iii) A typical geometry model usually consists of both solids sections and thin walled sections. Senthil Kumar A and K.H.Lee used a suitable dimensional reduction algorithm to reduce a model to a non-manifold model consisting of solid portions and two-dimensional portions which represent the mid-surfaces of thin-walled sections. He devised an algorithm to mesh the solid entities using three-dimensional elements and the surface elements using two-dimensional elements. The algorithm automatically generates a mesh of mixed two-dimensional and three-dimensional elements. It also ensures that the mesh is conforming at the interface of the non-manifold geometries.

PROPERTIES OF MATERIALS USED:

Table 1: Material properties

	Steel	**Aluminium**	**Foam**	**PVC**
Young's modulus (N/mm^2)	1.9e5	81358.14	5.060752	1447.899
Density (Kg/ mm^3)	7.8e-9	2.4e-9	4.98e-11	1.037e-9
Poisson Ratio	0.29	0.330	0.4	0.380

SPECIFIED ELEMENT QUALITY:

Table 2: Element quality

Specifications	2-D	3-D
Warpage	1	30
Aspect ratio	5	10
Skew	60	60
Length (mm)	1	500
Jacobian	0.5	0.7
Min. angle of quad (deg)	45	45
Max. angle of quad (deg)	135	135
Min. angle of tria (deg)	20	20
Max. angle of tria (deg)	120	120

CONSTRAINTS, LOADING AND BOUNDARY CONDITIONS:

•Since bracket is fixed to the upright of refrigerator assembly, bracket design should not be altered.

•Shelf bolt diameter and locations should not be altered.

•Loading of 1.9e-3 Mpa (0.278 PSI) pressures are applied uniformly on face of elements in top plate of the shelf assembly.

•Three protrusions are provided in bracket, which in turn are fixed to upright. All three protrusions are constrained in three translational directions (x, y, and z).

SPECIFIED REQUIREMENTS:

•Maximum Shelf Mass should be 6.5 Kg (14.5 lb).

•Maximum allowable Vertical Deflection of Shelf is 6 mm (0.235 inches).

3. FINITE ELEMENT MODEL OF REFRIGERATOR ASSEMBLY

DESIGN DETAILS:

Brief description: All sheet metal parts (Upright, bottom-tray, cross-brace, rear support, etc.) are meshed with shell63. PVC I-Section and Shim was meshed with Solid45. Foam with foam panel was meshed with shell181 (Layered Shell with different layer thickness & material properties). Doorframe was meshed with Solid45 and Beam188. The horizontal component (Top and bottom) of doorframe was meshed with Solid45 and vertical component (End and center mullions) of doorframe was meshed with Beam188. Glass doors are meshed with mass element at its centroid.

Assumptions and restrictions: 2-D elements are assumed to be in a global Cartesian Z = constant plane. The mass element has no effect on the static analysis solution unless acceleration or rotation is present, or inertial relief is selected. Glass door weight 34 Kg (75 lbs) each was applied as a downward force on the mass element at the centroid location. The Vertical components of the door frame (end and center mullions) were modelled with beam elements of circular cross section of radius 25.4 mm (1").Aluminium material properties were assigned to the beam element.

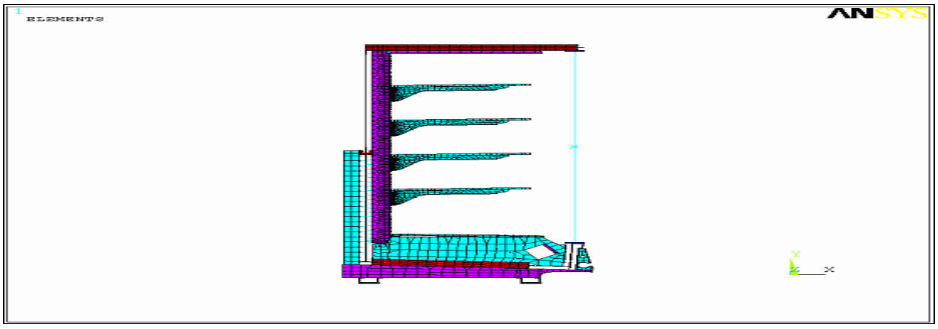


Fig 1: Finite Element Model of Refrigerator assembly

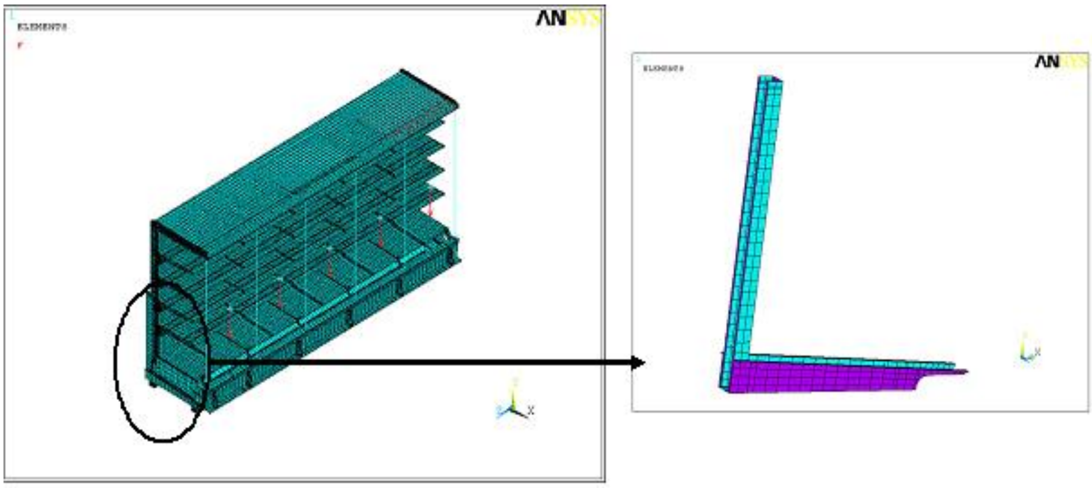


Fig 2: External Rear Support and Cross Brace

Boundary Conditions: All the parts were connected using Rigids

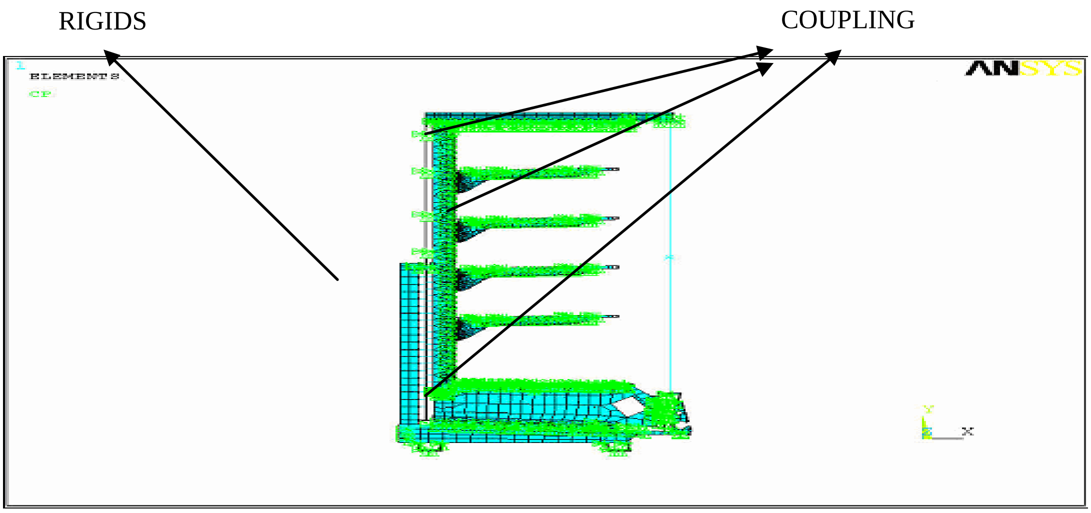


Fig 3: FE Model shows the connection of components

The mass elements were connected to the Door frame (Beam188) at top and bottom through the Rigids.

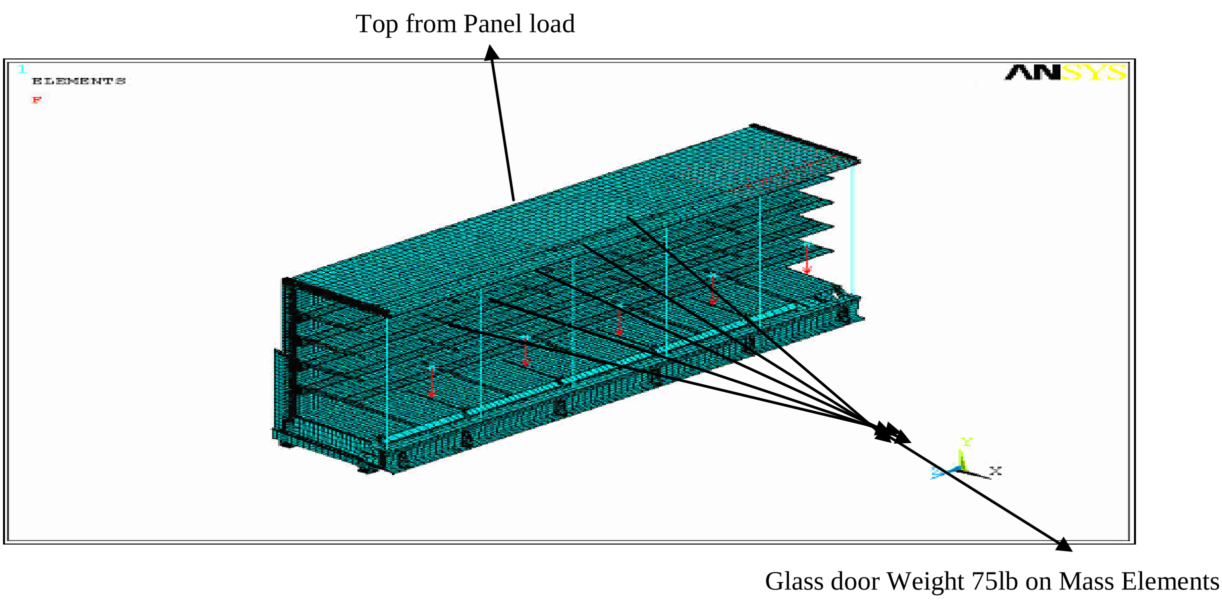


Fig 4: FE Model shows the connection of mass element to the beam element

The bottom surface of the shim was constrained in all direction

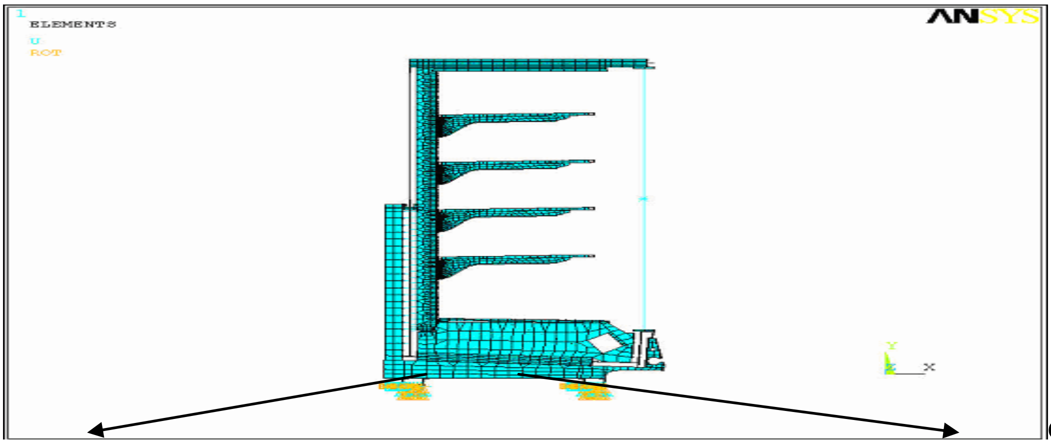


Fig 5: FE Model with constrained shim

Loads are applied on shelves

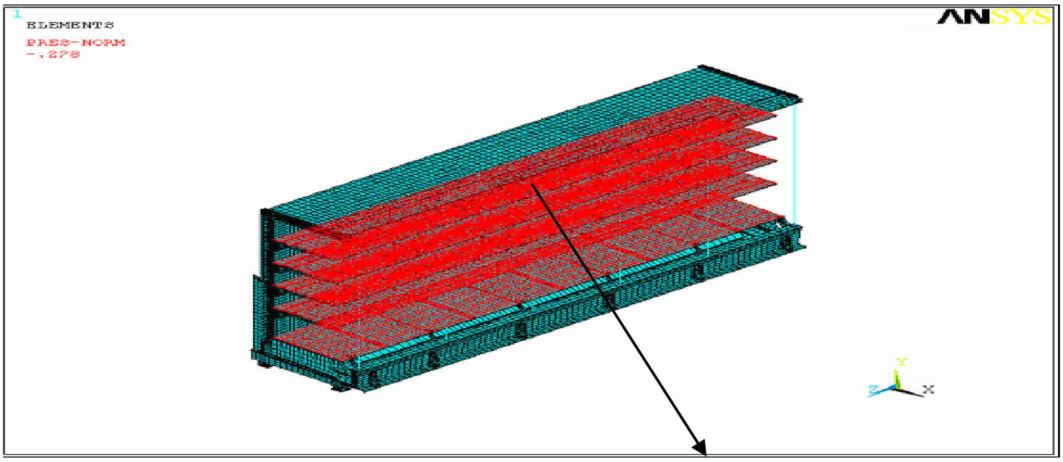


Fig 6: FE Model with Loads applied on shelves

4. RESULT PLOTS AND OBSERVATIONS

STRESS PLOT:

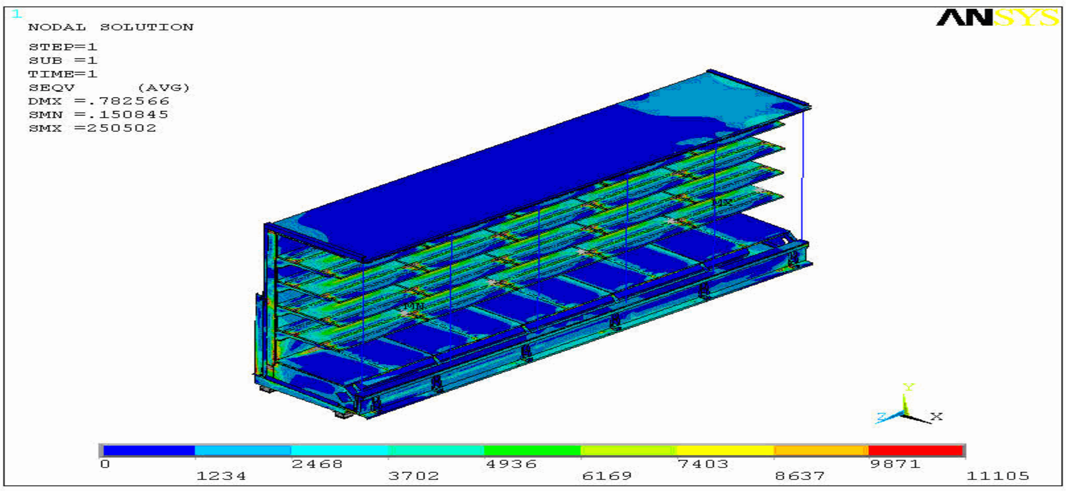


Fig 7: Von-Misses stress plot – Full structure

Observations: The high stress regions were observed at the Brackets area where it changes from smaller to higher width. The high stress regions were observed at the connecting area of Upright and Brackets. The maximum value of the stress is 76.56628 Mpa (11105.0 Psi).

DISPLACEMENT PLOT:

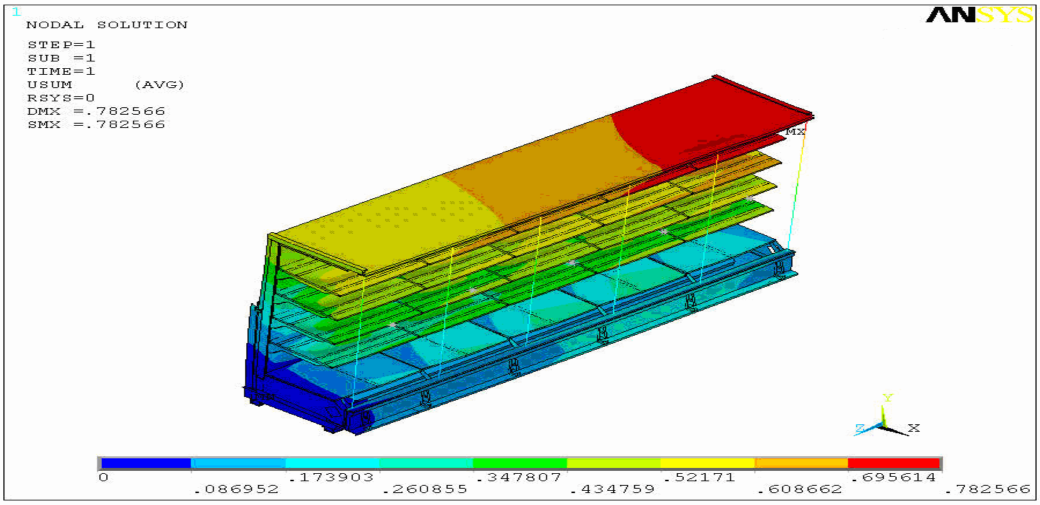


Fig 8: Displacement plot – U sum

Observations: The high stress regions were observed at the Brackets area where it changes from smaller to higher width. The high stress regions were observed at the connecting area of Upright and Brackets. The maximum value of the stress is 76.56628 Mpa (11105.0 Psi).

Deflection Results:

Deflections in the x-axis at the rear top foam panel at each rear support location are shown below table

Table 3: X-Deflection at 1 to 6 locations

	Def @ loc1	Def @ loc2	Def @ loc3	Def @ loc4	Def @ loc5	Def @ loc6
FE value (mm)	12.993	13.469	14.556	15.666	16.753	17.873
FE value (inch)	0.5092	0.5303	0.5731	0.6168	0.6596	0.7037

Deflections in the x-axis at the bottom of the upper foam panel at each rear support location are shown below table

Table 4: X-Deflection at 7 to 12 location

	Def @ loc7	Def @ loc8	Def @ loc9	Def @ loc10	Def @ loc11	Def @ loc12
FE value (mm)	5.1333	5.750	6.517	7.066	7.5438	7.693
FE value (inch)	0.2021	0.2264	0.2566	0.2782	0.2970	0.3029

TOPPLING ANALYSIS:

Toppling analysis is carried out to find the critical angle of tilt due to unbalance of structure while transportation. While lifting the structure from one place to another place, structure has to be tilted to a small angle so that the structure sits firmly on the automated guided vehicle. Hence it is very important to find the critical angle of tilt for safe transportation and thereby preventing from a huge loss. Finally Toppling analysis of entire refrigerator assembly was carried out to find the critical angle of tilt. One of the main applications of toppling is in rock sliding. Toppling occurs when, center of gravity of structure fall outside the dimension of its base. During toppling, the structure rotates about a pivot point. For toppling to occur, forces and weight of the structure must generate a negative moment on the structure with respect to pivot point.

RESULT PLOTS:

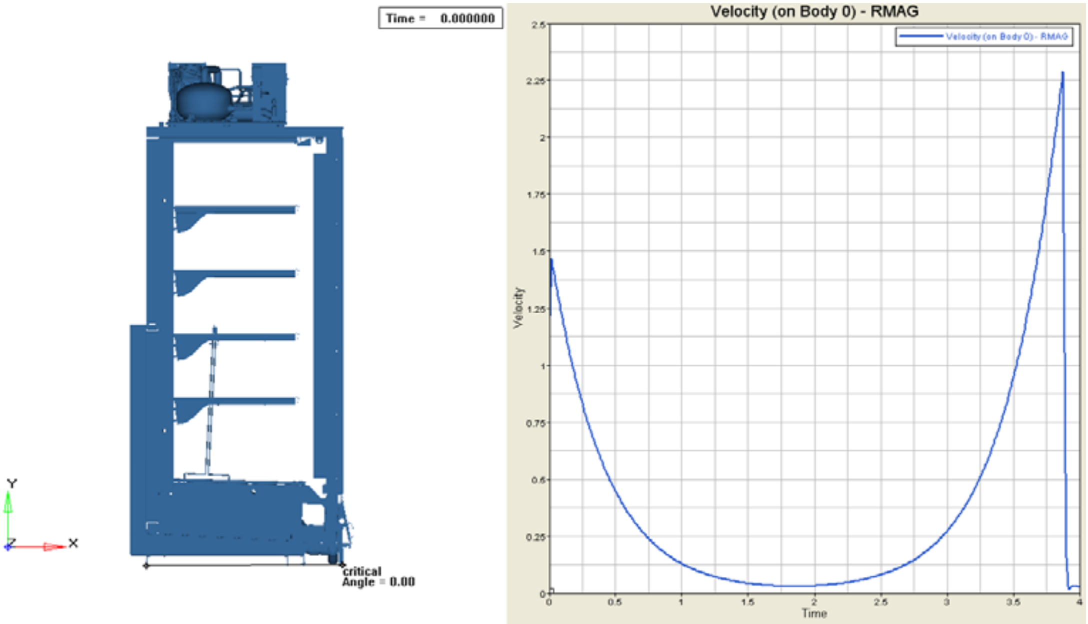
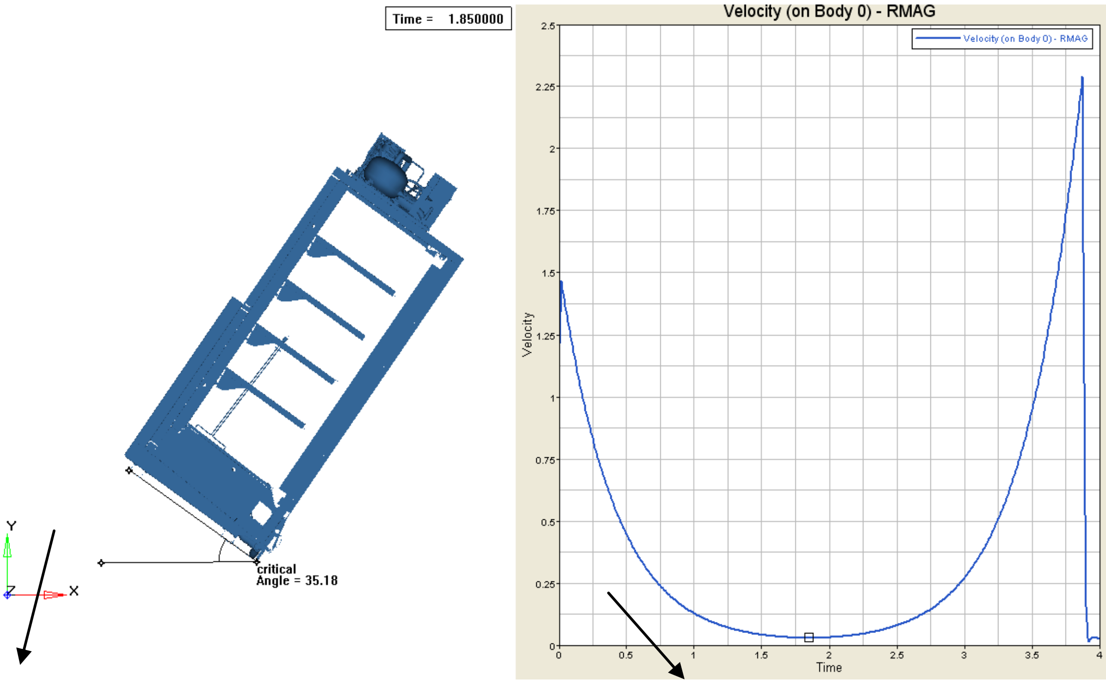


Fig 9: Refrigerator at rest



Critical angle =35.18 degrees

Velocity is approximately zero

Fig 10: Refrigerator at critical angle position

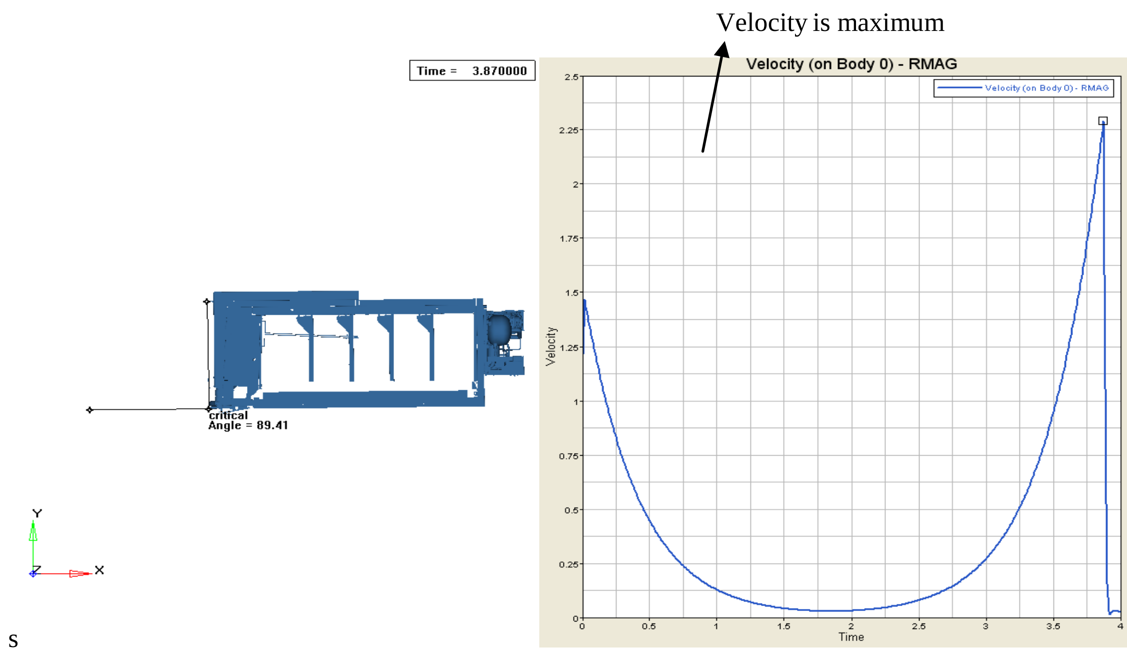


Fig 11: Refrigerator at ground impact position

5. CONCLUSION

The application of FEA in the design optimization of engineering structures has been emphasized through this work. Objective of first stage is achieved in third iteration conducted on the refrigerator shelf components. A summary of design iterations conducted on the refrigerator shelf components and comparison of optimization model masses with respect to base model mass in each of the iterations, are listed in the following table.

Reduction in mass of optimization models with reference to base model

Table 5: Percentage reduction in mass

	Displacement(mm)	**Mass(Kg)**	**Mass Reduction(%)**
Base Model	5.814	7.155	0
Optimized Model (Iteration1)	6.1	6.5	9.154
Optimized Model (Iteration2)	6.07	6.54	8.5953
Optimized Model (Iteration3)	5.86	6.343	11.34

The objective of second stage is achieved by analysing the entire refrigerator structure to find the maximum stress and displacements. The Maximum stress is observed at the connecting regions of upright and shelf brackets. The maximum value of the stress observed is 76.56628 Mpa (11105.0 Psi). The maximum deflection was observed at the right corner of the top shelves. The maximum deflection is 19.8755 mm (0.7825 inch). The deflections at 12 different locations above rear support are tabulated in Table. Toppling analysis was carried out to find the critical angle of tilt while transportation. It was very important to know the critical tilt angle because the structure may fall during transportation, when it is tilted while lifting. From the analysis results, it can was known that the critical angle of tilt for the refrigerator structure is 35.18 degrees. If the structure is tilted more than the critical tilt angle, it will the ground and may result in a huge damage to the structure. The finite element analysis capability is to quickly and accurately solve complex problems such as the one presented in this project, makes this method an ideal approach for such situations. Combined with competent error analysis techniques and in the hands of a knowledgeable finite element modeler, enabled by advanced software (such as Ansys, Nastran, Abaqus), finite element analysis should prove itself to be an indispensable part sof future minimally invasive product design and development.

NOMENCLATURE:

FEA Finite Element Analysis

E Young's Modulus

µ Poisson Ratio

ρ Density

t Thickness

Dof Degrees of freedom

.mdl Model Data Language

IGES Initial Graphics Exchange Specification

Mrf Animation file format in MotionView

.abf Plot file format in MotionView

REFERENCES

[1] Giridhar Kumar, UmaMaheswari, Abhijit Kulkarni & Shyam T Gopal, 2003 “Fea Simulation Of Refrigerator Cabinet Deflection”, Ansys India User Conference, Nov 2003.

[2] Owen, Steven J., Scott A. Canann and Sunil Saigal, 1997 "Pyramid Elements for Maintaining Tetrahedra to Conformability", Ansys India User Conference, Nov 2003.

[3] Robert Cook. D. 1989, “Concepts and Applications of Finite Element Analysis”, John Wiley and Sons., Canada.

[4] Tirupathi R Chandrapatla and Ashok D. Belagundu, "Introduction to Finite Elements in Engineering", Prentice Hall of India. Ltd.

[5] Klaus Jurgan Bathe, "Finite Element Procedures", Prentice Hall of India, New Delhi.

[6] Reddy, J. N., 1984, 1993, "An Introduction to the Finite Element Method".

[7] Desai, C. S. and Abel, J. F., 1972, "Introduction to the Finite Element Method".

[8] "Finite Elements and Solution Procedures for Structural Analysis", Vol.1: Linear Analysis, Crisfield, M. A., 1986.

[9] S.S. Bhavikatti, 2002, "Strength of materials".

[10] L.S. Srinath, 2003 "Advanced Solid Mechanics".

[11] Timoshinko, (Vol.1 & 2) "Strength of Materials".

[12] Phillip.L.Gould, 1980, “Static analysis of shells”, Washington university.

[13] HyperMesh, HyperView, MotionView user manuals.

[14] Quick reference guide – MSC .Nastran 2005.

[15] Ansys user manual – Structural Analysis Guide.

[16] Sheet Metal Thickness Gauges- American standards, Russ Rowlett and the University of North Carolina.

[17] Old reports of HCL technologies limited, Bangalore.