IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 10, Issue 2 (Nov. - Dec. 2013), PP 01-05
www.iosrjournals.org

Optimized mould design of an Air cooler tank

M. Nagarjuna[1], E. Kavitha[2]
[1] *Department of Mechanical Engineering, P.V.P. Siddhartha Institute of Technology, Vijayawada, INDIA*
[2] *Department of Mechanical Engineering, P.V.P. Siddhartha Institute of Technology, Vijayawada, INDIA*

Abstract: *Proper modeling of mould for an air cooler tank is necessary to facilitate the ease for production line and weight reduction of the complete component assembly. The present research work aims at performing the structural analysis separately on 3 different models of moulds designed:*
Model-1: Mould extracted from the Pro-E software manufacturing module (say Thickness =‘t’).
Model-2: Thickness reduced to half of the previous model for weight reduction. ($t_1= ½(t)$)
Model-3: Thickness reduced to half of the previous model for weight reduction. ($t_2= ¼(t)$).

The aim of the present work is to study the variation in displacement and stress values between Model-1, Model-2 and Model-3. This analysis is performed using FEM in ANSYS Software. The study is intended for appropriate reduction of thickness there by reducing the weight of complete assembly, which in turn reduces the complete cost of production of mould for an air cooler tank.
Keywords; *ANSYS, Displacement, FEA, Stress*

I. INTRODUCTION

Air coolers also called evaporative coolers are used for cooling purposes. They are different from air conditioners in the sense air conditioners use refrigeration cycle principle whereas air coolers use the evaporation of water principle. Air coolers work on the principle of heat loss during the evaporation process, resulting in cooler air. Cooler tank is one of the most important parts in water coolers which store the water.
ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

II. PROBLEM MODELING

2.1 Geometric Modeling

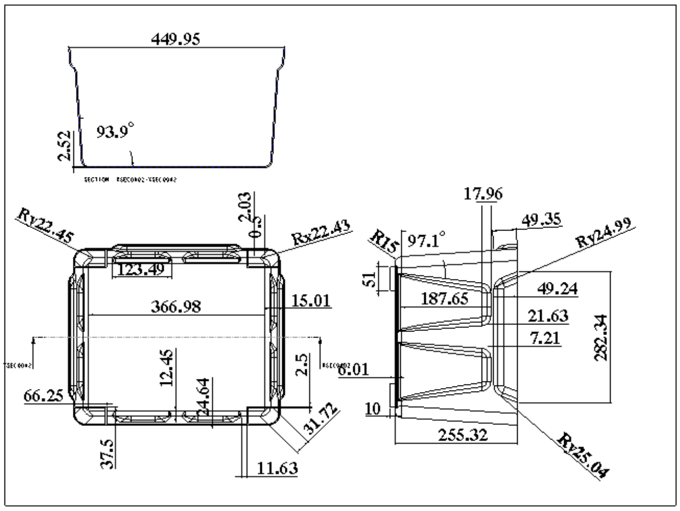

Figure 1: 2-D Specifications of the cooler tank

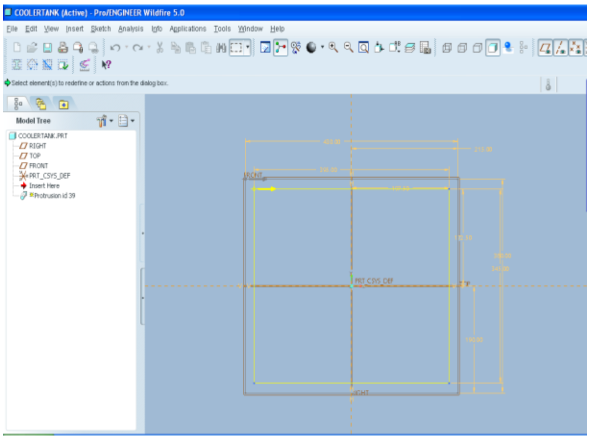

Figure 2: Dimensions of the cooler tank in Pro-E

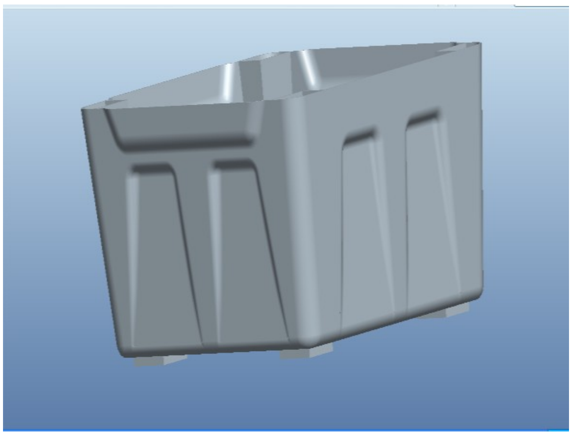

Figure 3: Model of cooler tank in Pro-E

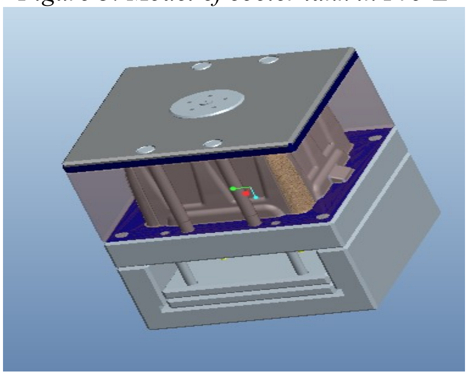

Figure 4: Die assembly in Pro-E Manufacturing module

2.2 Finite element Modeling

The problem is modeled in ANSYS software and the finite element mesh is generated using SOLID 20Node95 element. This is a 20node second order brick element having three degrees of freedom at each node and is suitable to incorporate solid isotropic material properties. Meshing is used to deconstruct complex problem into number of small problems based on finite element method.

2.3 Loads and Boundary conditions

At the bottom face of the mould, degrees of freedom in all directions are constrained. A pressure of 15.79 N/mm^2 is applied on the top surface of the model.

Pressure calculation:

Load (1000000*9.81) =9810000 N

Area = 621152 mm^2

Pressure = 15.79N/mm^2

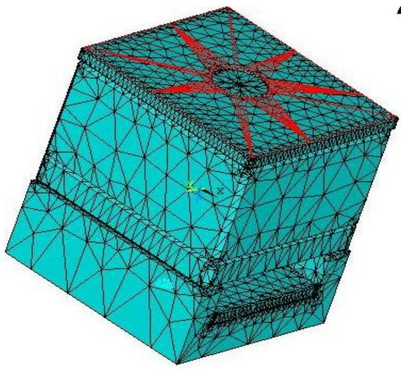

Figure 5: Mould assembly with Loads and boundary conditions

2.4 Material properties

Young's Modulus (EX) : 20900N/mm^2
Poisson's Ratio (PRXY) : 0.27
Density : 0.000007876 kg/mm^3

III. ANALYSIS OF RESULTS

Stress and displacement values are plotted for the 3 models.

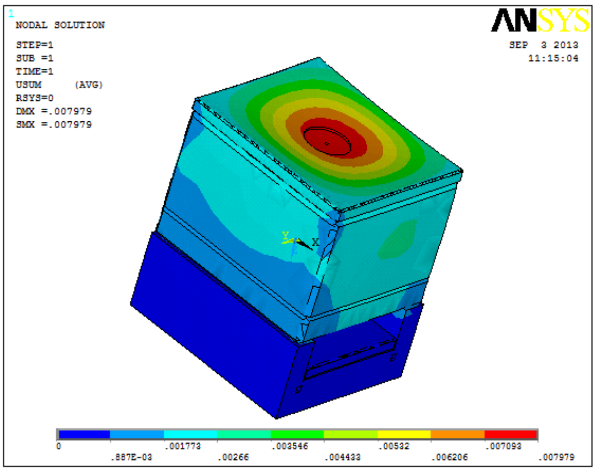

Fig 6: Contour plot for displacement in Model-1

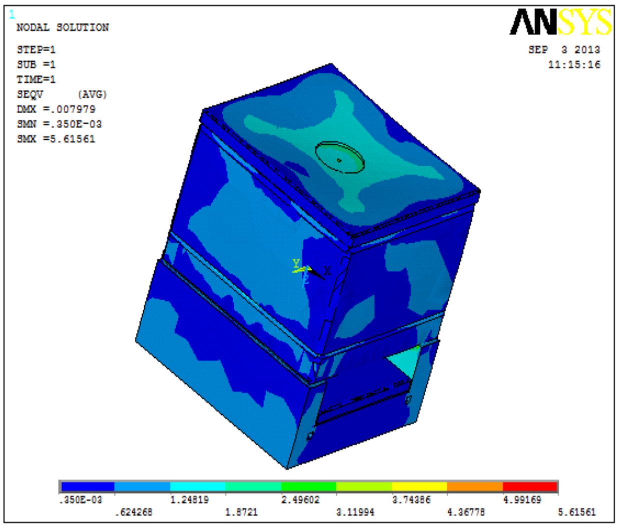

Fig 7: Contour plot for stress in Model-1

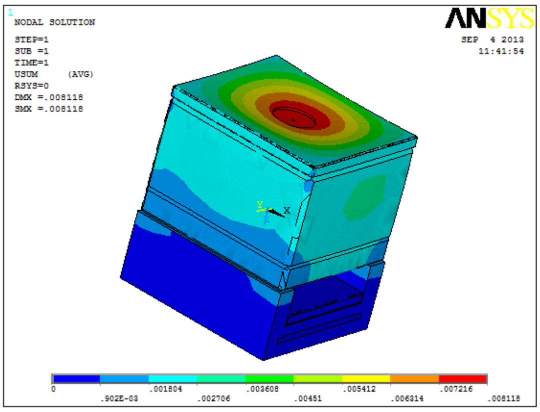

Fig 8: Contour plot for displacement in Model-2

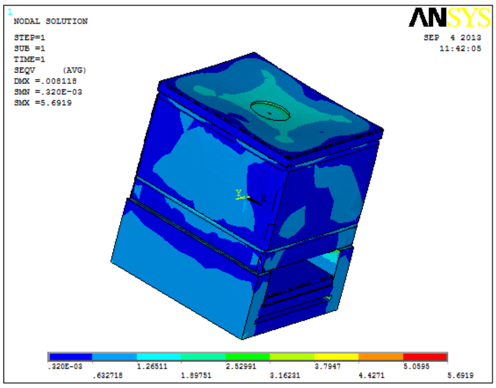

Fig 9: Contour plot for stress in Model-2

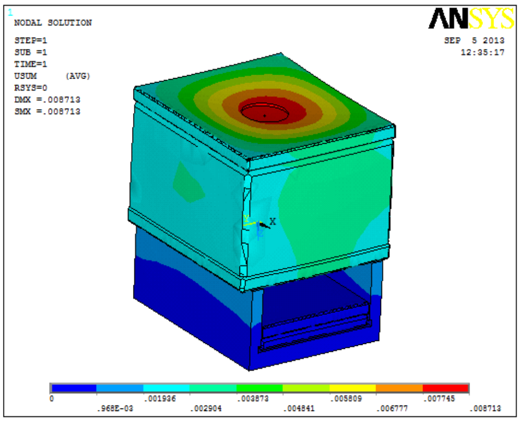

Fig 10: Contour plot for displacement in Model-3

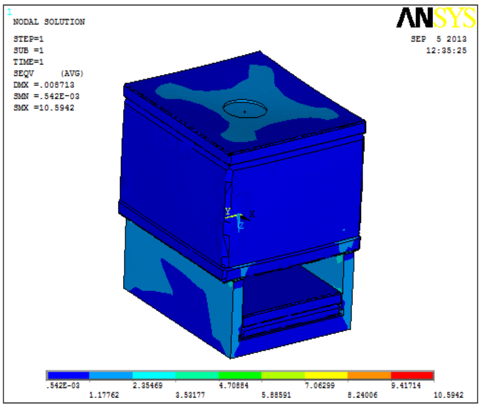

Fig 11: Contour plot for stress in Model-3

Structural analysis results:

	Displacement (in mm)	Stress (in N/mm^2)
Model-1	0.007979	5.61561
Model-2	0.008118	5.6919
Model-3	0.008713	10.5942

IV. CONCLUSION

Structural analysis of the three different models is performed by varying the thickness of the mould assembly components. Displacement and stress values are noted for all the models which are analyzed individually.

Hence, it is apparent from the above results that the most reduced model (Model-3) is the best model to be used for performing manufacturing to obtain a weight reduction of 75%.

REFERENCES

[1] "*Machine design"*, T.V.Sundararajamoorthy, 2nd Edition 2010.
[2] *Design data book*: P.S.G.College of Technology (Kalaikathirachchagam).
[3] ANSYS Reference Manuals, 2012.
[4] Daniel I.M. and Ishai.O, "*Engineering Mechanics of Materials*", Second Edition, Oxford press, 2006.