Proceedings of the World Congress on Engineering 2010 Vol II
WCE 2010, June 30 - July 2, 2010, London, U.K.

Design, Finite Element Analysis and Optimization of HRC Trays used in Heat Treatment Process

Dinkar Nandwana, Bhupendra, Nikhil Khandelwal, Tushar Bhargava, Kapil Nandwana and Ganesh Jawale

Abstract— **This paper presents a methodology to increase the life cycle of Heat Resistant Casting (HRC) trays whilst reducing their total weight. Structural analysis is performed using displacement based Finite Element Method. The geometric modeling is performed via CATIA V5 R11 while the meshing and analysis is performed in ANSYS WORKBENCH 10.0. In depth study of stresses and deformations in the existing tray is performed to optimize the structure.**

Index Terms: **Heat Resistant Casting Trays, life, finite element analysis, design, quenching.**

I. Introduction

The existing HRC trays used for mounting crankshafts, knuckles and various other products for heat treatment in the Heat Treatment Department develop cracks and become unusable after 3 months of continuous use. Therefore, new trays have to be used every 3 months. Manufacturing them is both time and money consuming process. The objective of the current study is to perform a detailed structural analysis of the existing trays and modify the existing design to increase its life and thereby increasing profit.

II. Tray and its function

Forged components like crankshaft, knuckles are heat treated to increase their hardness and stiffness. But due to typical design, these parts cannot be directly loaded into to furnace. The trays serve as a base for these parts. The parts which are to be heat treated are loaded on it and then introduced into the furnace .The tray is designed in such a way that the treated parts are uniformly hardened, quenched, tempered and cooled. The tray, made up of structural steel, undergoes changes in material properties due to temperature variations in the heat treatment process. The following section deals with temperature dependent variation in material properties of steel.

Manuscript received on March 26, 2010. This work was supported by Bharat Forge Limited, Pune, India.

(a)Dinkar Nandwana is an undergraduate student pursing B.E. Mechanical Engineering from Birla Institute of Technology and Science, Pilani, Rajasthan-333031, India. (Corresponding Author. Phone: 091-294-2417445, e-mail f2007721@bits-pilani.ac.in)
(b)Bhupendra is an undergraduate student pursing B.E Mechanical Engineering from Birla Institute of Technology and Science, Pilani, Rajasthan-333031, India.(e-mail: f2007361@bits-pilani.ac.in)
(c)Nikhil Khandelwal is an undergraduate student pursing B.E Mechanical Engineering from Birla Institute of Technology and Science, Pilani, Rajasthan-333031, India. (e-mail: f2007370@bits-pilani.ac.in)
(d)Tushar Bhargava is an undergraduate student pursing B.E Mechanical Engineering from Birla Institute of Technology and Science, Pilani, Rajasthan-333031, India.(e-mail: f2007296@bits-pilani.ac.in)
(e)Kapil Nandwana is a graduate student from The Ohio State University, Columbus, Ohio, USA. (e-mail: nandwana.1@osu.edu)
(f)Ganesh Jawale is presently the Associate Vice President of Heat Treatment Department, Bharat Forge Limited, Pune, India. (e-mail: ganeshjawale@bharatforge.com)

III. Mechanical properties of steel

In one cycle the tray is subjected to different temperatures at different time. During quenching the properties of the steel changes instantaneously due to the instantaneous change in temperature from 800 °C to 50 °C . Figure 1 to 4 shows the variation of mechanical properties with temperature.

IV. Specification of existing tray

a) The length, width and height are 1140mm, 760mm and 45 mm respectively. (Figure 5)
b) Weight of the existing tray is 85 kg.
c) The load applied on the tray is 650 kg.
d) Existing life of this tray is 3 months, when used continuously.
e) Each tray costs around USD 770.
f) Each tray takes approximately 6 hours for one cycle before it can be reused.
g) Each tray is used 3 times a day for heat treatment.

V. Finite Element Analysis of existing tray

The finite element analysis of the tray was completed in 3 phases. First, the exact geometry of tray was modeled in CATIA V5 R11.The model was then imported in ANSYS WORKBENCH 10.0 for meshing. Boundary and loading conditions were applied to the FE model to simulate the exact environmental conditions.

A. *Modeling*

Figure 6 shows the 3D model designed in CATIA V5 R11.

B. *Meshing*

The imported model was then meshed in ANSYS WORKBENCH 10.0 (Figure 7). The mesh was then redefined continuously until convergence (5% error norm on maximum deformation) was reached.
The features of converged mesh are:-
a) Element Type: 3D-10 Node Tetrahedron (SOLID 92)
b) Number of nodes: 124829.
c) Number of elements: 63170.

C. *Loading and Boundary conditions*

The loads and boundary conditions applied on the tray (Figure 8) are-
a)Standard Gravitational Force acting vertically downwards ($g=9.8\ m/s^2$).
b) The weight of the components to be forged is applied as pressure acting uniformly over the top surface of the tray. The pressure applied is 7450 Pa.
c) The fixed support of conveyor present on the bottom of the tray.
d) A time dependent temperature condition of tray as shown in Figure 9.

VI. Results of analysis

A. Stress Plots

Figure 10 shows the Von Misses stress plots of the tray. It can be inferred that-
a) The maximum Von Misses stress on the tray is 9.756 GPa and it acts on the corner rib.
b) The most critical regions are the corners of the ribs in the tray.

This theoretical prediction correlates perfectly with actual regions of crack on the existing tray. (Figure 11).Hence, the Finite Element Analysis results are correct and can be used for future practical predictions.

B. Deformation Plots

Figure 12 shows the deformation plot of the tray. It can be inferred that-
a) The maximum deformation is 11.75 mm and it occurs at the corners of the tray.
b) The collars of the tray are more prone to deformation.

VII. Changes for Modified Tray model

Study of Stresses and Deformations in the existing tray suggests the following changes can be made to the design so as to increase the life of the tray. (Figure 13)
a) By looking at the maximum stress and deformation region it can be inferred that there should be a margin for contraction and relaxation of the boundaries. 'U' cuts can be introduced along the periphery of the tray for contraction and relaxation.
b) The ribs are subjected to the maximum stresses. Introduction of circular holes allows contraction and relaxation thus reducing stress and decreasing the weight as well.
c) The corners of collar face more deformation. Changing the shape to circular can reduce deformation.
d) In order to increase the strength of the ribs, the thinner ribs present alternatively can be converted into the same shape as the thicker one.
e) The cylindrical holes present on the lengthwise boundary can be made completely hollow since the region faces less stress. This will reduce the weight maintaining the same strength.
f) Sharp corners can be removed to avoid notch formation.

VIII. Finite element analysis of new tray

Same procedure as described earlier was followed for the modified tray.

A. Modeling

The modified tray was modeled.

B. Meshing

The meshed model of the modified tray is shown in figure 14. The mesh was then redefined continuously until convergence (5% error norm on maximum deformation) was reached.
The features of converged mesh are:-
a) Element Type: 3D-10 Node Tetrahedron (SOLID 92)
b) Number of nodes: 200252.
c) Number of elements: 107740.

C. Boundary conditions

Similar loads and boundary conditions, as for existing tray, were applied on the modified tray.

IX. Results of the analysis

A. Stress Plots

Figure 16 shows the stress analysis plot of the modified tray. It can be inferred that-
a) The maximum stress on the tray is 4.77 GPa and it acts on the corner rib.
b) The most critical regions are the corners of the ribs of the tray.

B. Deformation Plots

Figure 17 shows the deformation plot of the tray. It can be inferred that the maximum deformation is 11.48mm and it occurs at the corners of the tray.

X. Conclusions

By comparing the results of the existing tray to the modified tray it can be concluded that-
a) The maximum stress on the modified tray is 4.77Gpa compared to that of existing one which is 9.756Gpa. Since the stress value has decreased by half, it can be predicted (without performing detailed failure analysis) that the life of the tray will increase by at least two times. Therefore predicted life of this tray is 6 months as compared to 3 months of the existing one.
b) Modified design undergoes less deformation compared to the existing design.
c) The weight of the new tray is 80 kg as compared to the existing one which is 85 kg. Thus, 5 kg weight has decreased.
e) The cost of the new tray will be approximately USD 720.Thus the cost has reduced by USD 50 and life has increased by 3 months.

XI. Further recommendations

Apart from these changes, some further changes can be made in the tray design-
A) In order to decrease stresses at the corners, introduce a small strip of steel connecting the corner to the diagonally opposite end.
B) To decrease the weight further, introduce circular or cylindrical holes on the hollow ribs because these regions are low stress and deformation zone.
C) Weight of the tray can be further reduced by the introduction of 'U' cuts on the corners of the periphery of the tray.

References

[1] Shigley J.E., C.K. Mischke, "Mechanical Engineering Design", 6th edition.
[2] Bhandari, V.B., "Design of Machine elements", 2nd edition, Tata McGraw-Hill Publishing Company Limited, New Delhi.
[3] Pawar B.B. (2005), "A Dissertation on Design, Static analysis, stress correlation and weight optimization of Front Axle Beam", BFL, Pune.
[4] Gokhale, N.S., et al., "Practical Finite Element Analysis".(ISBN : 978-81-906195-0-9)
[5] Smoljan, B. (2002). "Numerical simulation of steel quenching" Journal of Materials Engineering and Performance, Volume 11, Issue 1, pp.75-79.
[6] Yeoh, Chuan Shuan. (2002). "Thermal Behavior of Steel Bar during Quenching." University Malaysia Sabah: Thesis Degree of Master of Science (Mechanical Engineering).

FIGURES

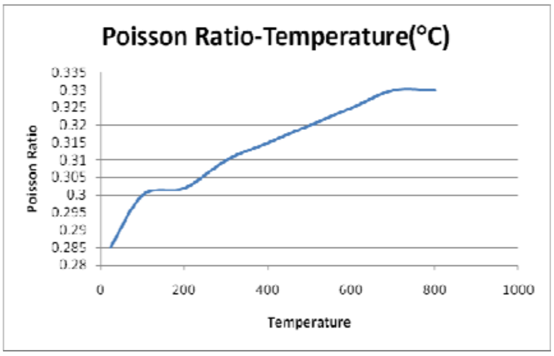

Figure 1: Poisson Ratio -Temperature Curve

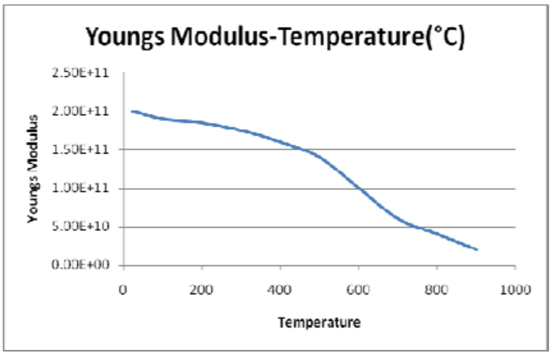

Figure 2: Young's Modulus- Temperature Curve

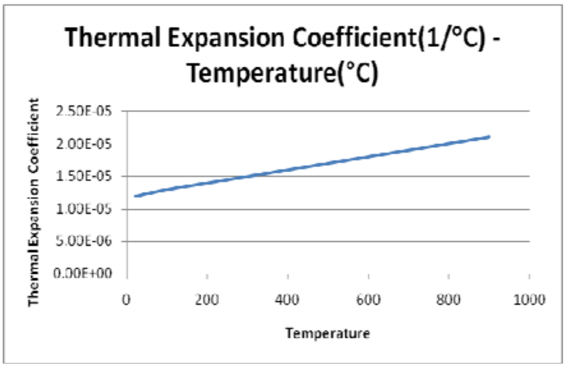

Figure 3: Thermal Expansion Coefficient-Temperature Curve

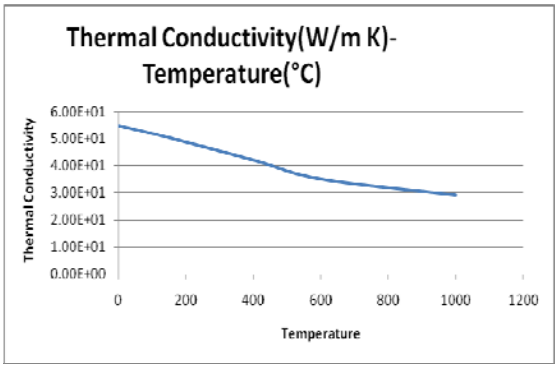

Figure 4: Thermal Conductivity-Temperature Curve

Figure 5: Photograph of Existing Tray Courtesy: Bharat Forge Limited,Pune

ISBN: 978-988-18210-7-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

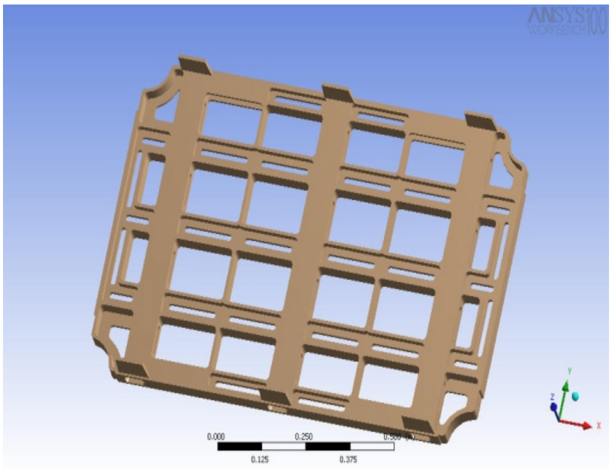

Figure 6: 3D model of existing tray

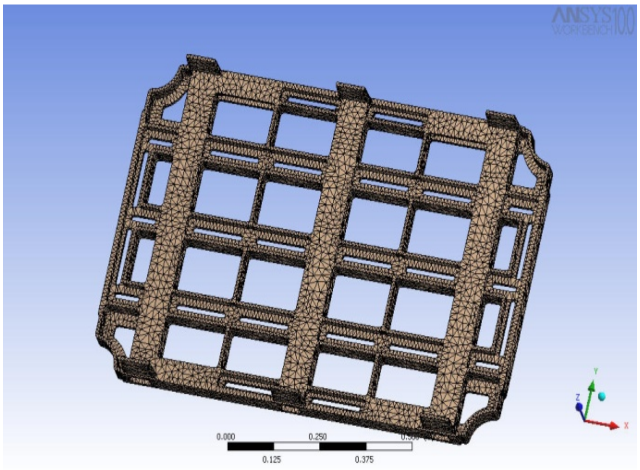

Figure 7:Meshed Model of existing tray

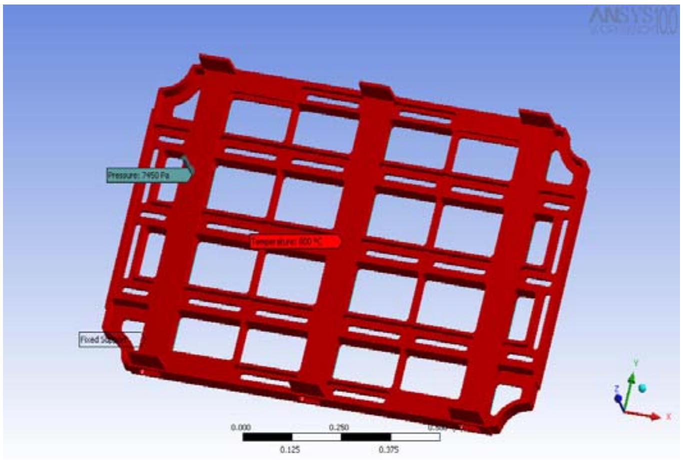

Figure 8: Loading and Boundary Conditions on existing tray

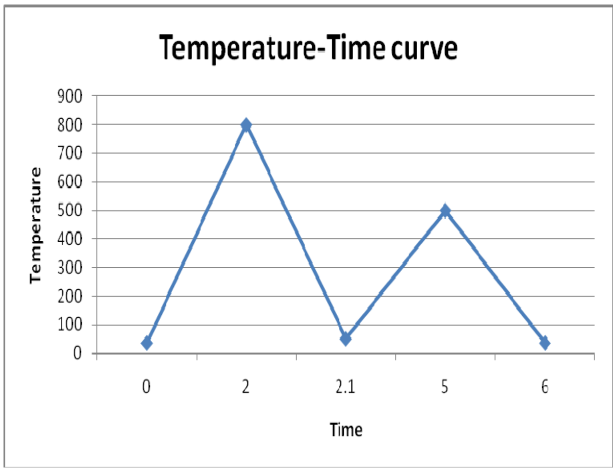

Figure 9: Temperature Time Curve

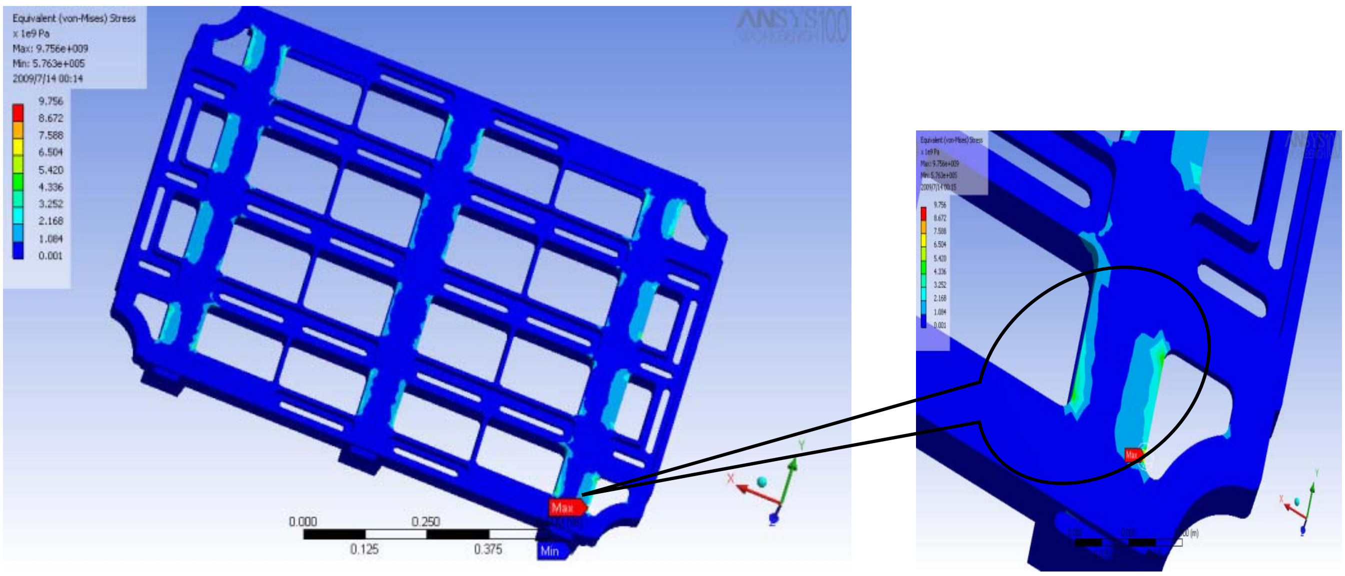

Figure 10: Stress plot of existing tray(right side shows the magnified view)

ISBN: 978-988-18210-7-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

Figure 11: Photograph showing crack formation in high stress regions in existing tray

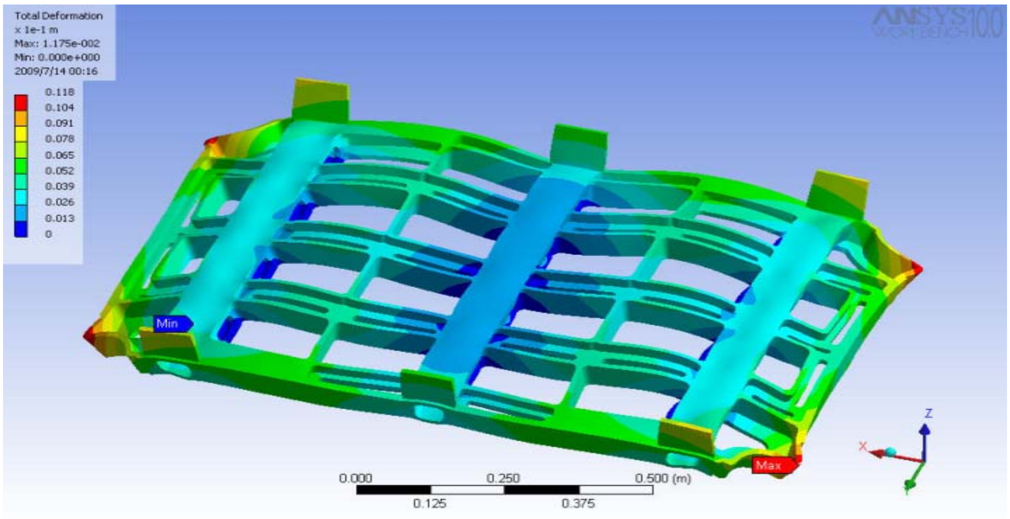

Figure 12:Deformation plot of existing tray(magnified X1000)

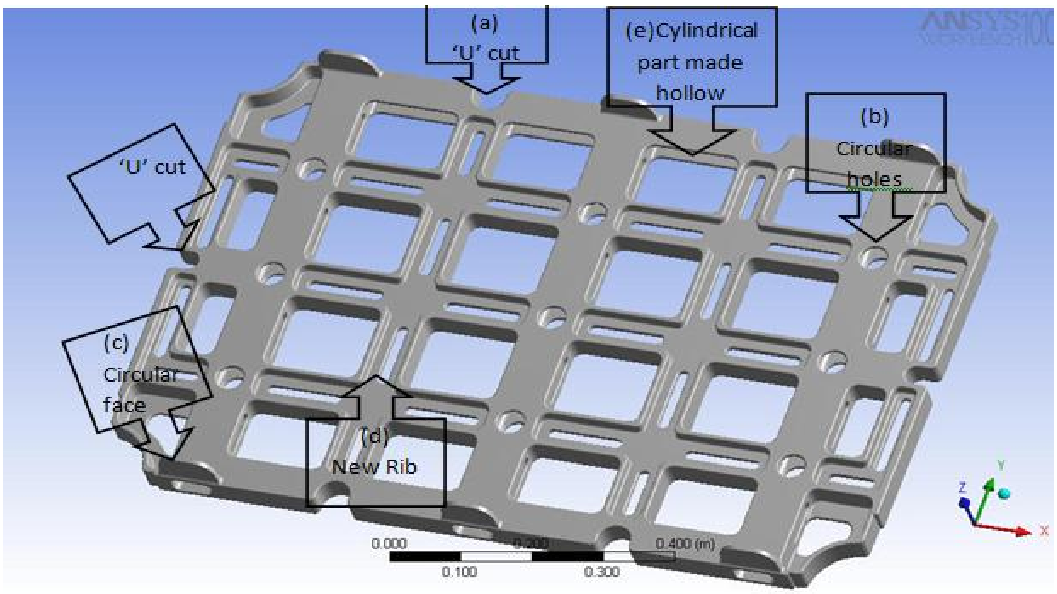

Figure 13: 3D Model of Modified Tray

ISBN: 978-988-18210-7-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

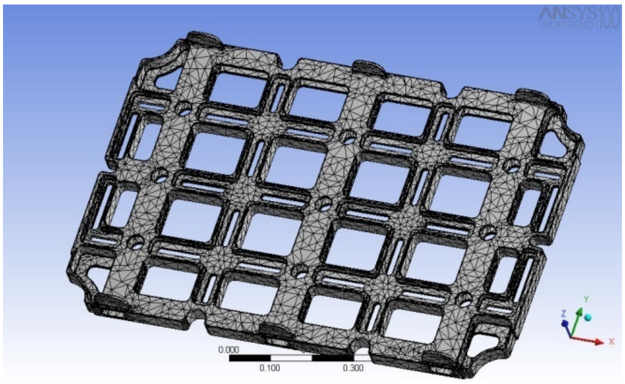

Figure 14: Meshed model of modified tray

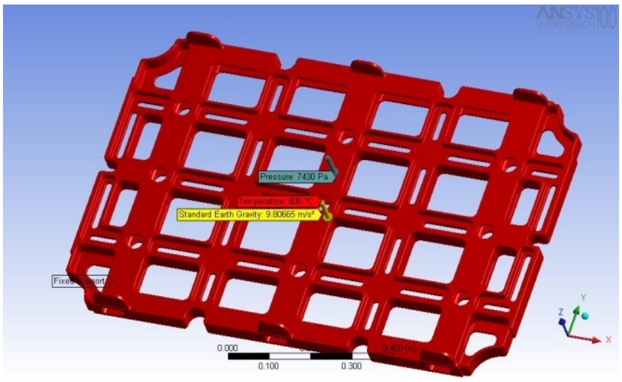

Figure 15: Loading and Boundary conditions on modified tray

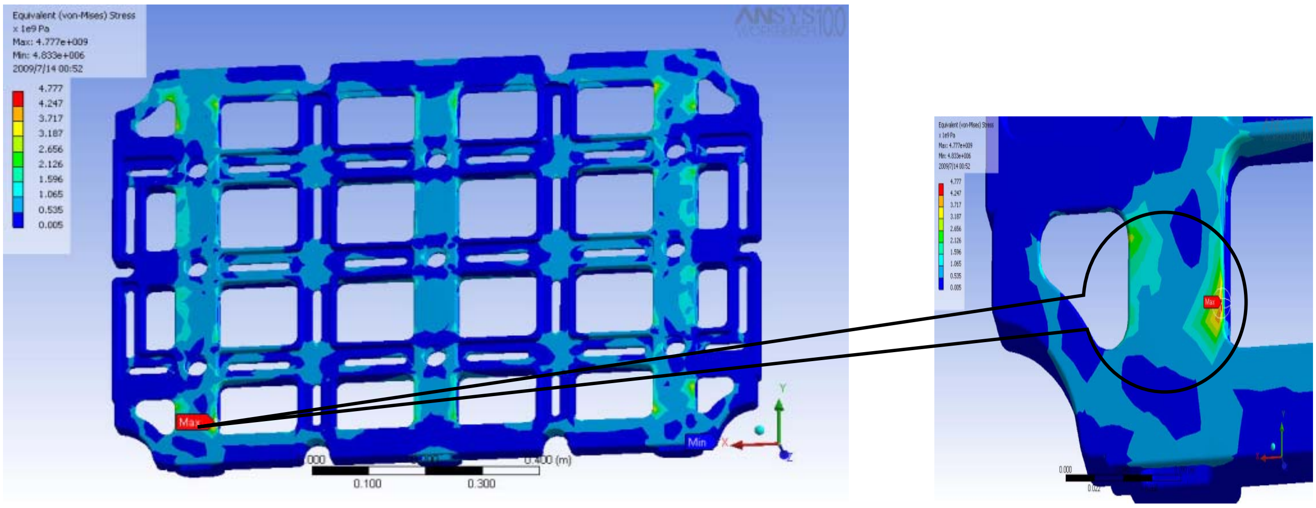

Figure 16: Stress plot of modified tray(right side shows the magnified view)

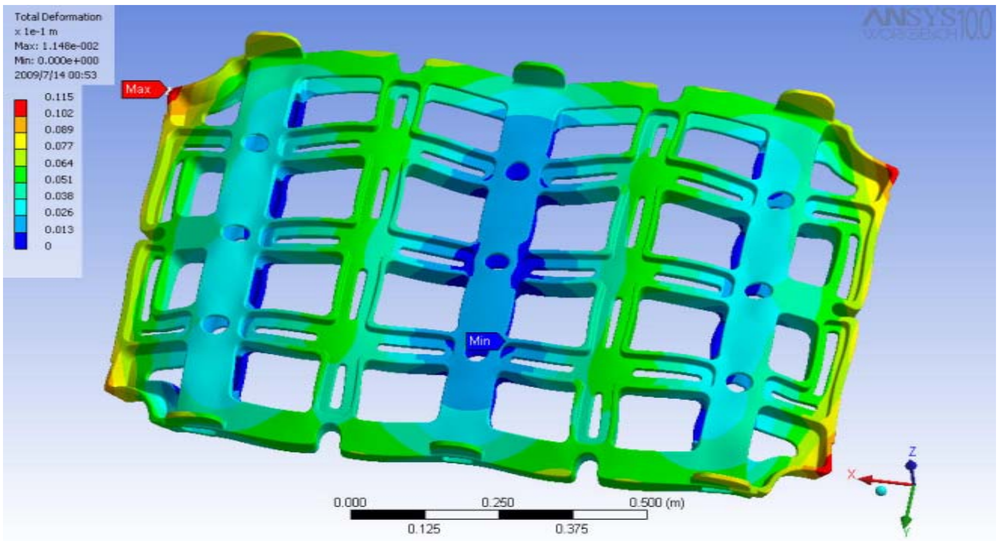

Figure 17 :Deformation plot of modified tray(magnified X1000)

ISBN: 978-988-18210-7-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)