ISSN 2395-1621

Design Evaluation & Optimization of Bed of EDM using FEM tools

#1Nitin k.Rasal, #2P.H.Jain

[1]nitinrasal12@gmail.com
[2]phjain30@rediffmail.com

[1]M. E. (Design Engineering) Mechanical Engineering Department,
TPCT'S College of Engineering, Osmanabad
Maharashtra, India
[2]Assistant Professor, Mechanical Engineering Department,
TPCT'S College of Engineering, Osmanabad
Maharashtra, India

ABSTRACT

The Machine Tool industry aims for high precision and repeatability while it is in operation. The quality of the machine tool, in fact, is determined on this count. The structural components in the machine tool play a vital role in helping to achieve Consistent performance. The damping for the vibrations, load-bearing capacity for the over-hung members, the stable alignment between the mating parts forming links and for the components experiencing dynamic/ rotation movement in the given pair. The model of this EDM machine under review is now undergoing changes to the structure. The new design needs to be reviewed in the light of structural strength while subjecting the components/ sub-assembly to Analysis using CAE. The geometry of the machine frame/ structure is amenable to the usage of 3D modeling. The design of the structure would necessitate knowledge of the fundamentals for Machine Design. The information like weight of the structure and the relative position with respect to other elements of the machine tool can be readily offered by the three dimensional CAD interface. Although the physical design can be done using the facilities and/or the faculties above, the assessment of the geometry for Conformance to the conditions specified (test conditions) could be done through the utilization of a suitable tool – Software for Analysis in the domain of Structural Analysis. With the past experience of the Sponsoring Company in this field, `NASTRAN or ANSYS' appears to be a competent tool to pursue Analysis for this Project Work.

Experimentation: Life testing for fatigue would be conducted using Special Purpose Machine. This setup would be used to induce cyclic loads over the EDM table with a predetermined frequency. The results depicting reason/s for failure would be attempted to be identified over the Design and/or Process for the component (table). The problem would be explored from Design through Process and marked for future implementation on the field upon successful validation.

Keywords— EDM (Electrical discharge machining), FEM (Finite Element Method), FEA (Finite Element Analysis

ARTICLE INFO

Article History

Received : 26th June 2015

Received in revised form : 27th June 2015

Accepted : 3rd July 2015

Published online : 6th July 2015

I. INTRODUCTION

Electrical discharge machining (EDM) is one of the earliest non-traditional machining processes. The EDM process is based on thermoelectric energy between the work piece and an electrode. Various types of products, such as dies and moulds, can be produced by EDM. During EDM of metals, a large amount of heat is generated, which affects the surface characteristics of the metals. However, this phenomenon is unavoidable during EDM of metals, and some technical problems remain unsolved in the area of surface integrity of the machined work piece. Electrical discharge machining (EDM) is a process used to shape hard metals and form deep and complex-shaped holes by electro-erosion in all types of electro-conductive materials [1-4]. Surface finish and integrity are two different facets of the cavity quality, but both play an important part in the characteristics of the mould. Many of the machine parameters affect the integrity of the sub-layers of the cavity, and they also affect the surface finish. Among the authors who studied the influence of EDM parameters on surface roughness. A considerable research was carried out on the machining of the 40CrMnNiMo8-6-4 steel tool [5]. It was observed that the surface roughness of the work piece was influenced by pulsed current and pulse time. “Higher values of these parameters

Increased surface roughness. Lower Current, lower pulse time and relatively higher pulse pause time produced a better surface finish”. The relationship between EDM parameters and surface cracks was investigated by [2,6]. They analyzed the EDM of D2 and H13 steel tools. “The formation of surface cracks is explored by considering surface roughness, white layer thickness, and the stress induced by the EDM process”. They conclude that the white layer thickness is mainly influenced by the pulse-on duration and that it increases as the pulse-on duration increases.

If crack appear, they would be micro-cracks and exist in the white layer (WL), and the cracks would begin at the white layer’s surface and travel down perpendicularly towards the parental material. Some others have published results of an experimental investigation, which studied the effects of electrode material changes on the machining performance of steel EN-31 [7]. They concluded that the best rate of machining is carried out with an aluminum or copper electrode.

Material Properties for EDM work table:

Table 1 shows the material properties of the given EDM work table.

Table 1: Material properties for EDM table trim model

Sr. No.	Parameter	Descriptions
1	Material	Mild Steel
2	Young's Modulus E	$2.1x10^5$ N/mm^2
3	Density ρ	$7.86X10^{-6}$ kg/mm^3
4	Poisson's ration	0.3
5	Yield stress	580 N/mm^2

Boundary Conditions

The EDM work table is mounted on the different thickness channel support, the frame of the EDM table support is constrained as UX, UY, UZ, ROTX, ROTY,ROTZ.

Loads Applied

The load is applied along Fy direction. To apply load, it is necessary to select the nodes associated at the center along z direction. It is necessary to observe the number of nodes associated at the center along z direction, because the applied load needs to be divided with the number of nodes associated with it. After the preprocessing, the solution has to be done. From solution phase, choose the new analysis as static, then solve the current load step option, the solution will be done. EDM work table was modeled using commercial software and all the specification was accordingly followed the relevant drawing standard. The table geometry was shown in Figure 1. The EDM table assembly is modeled using competent CAD software CATIA V5 R20.

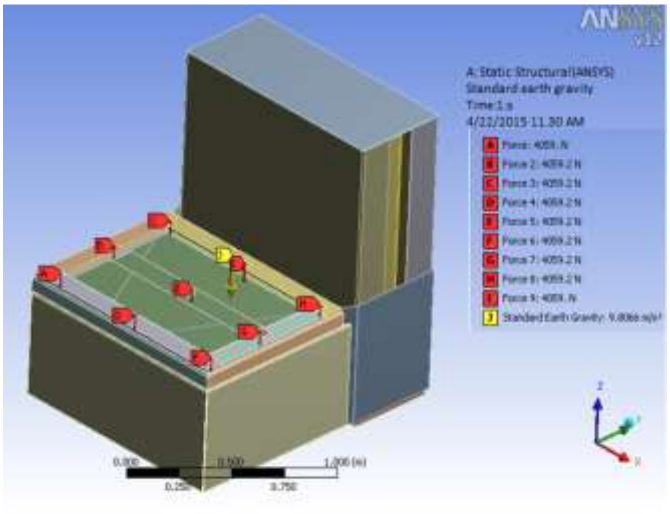

Figure 1: Load application on EDM Table assembly

Stress analysis has been carried out to evaluate structural integrity of the Table Assembly under given loading conditions as shown in figure 2

.

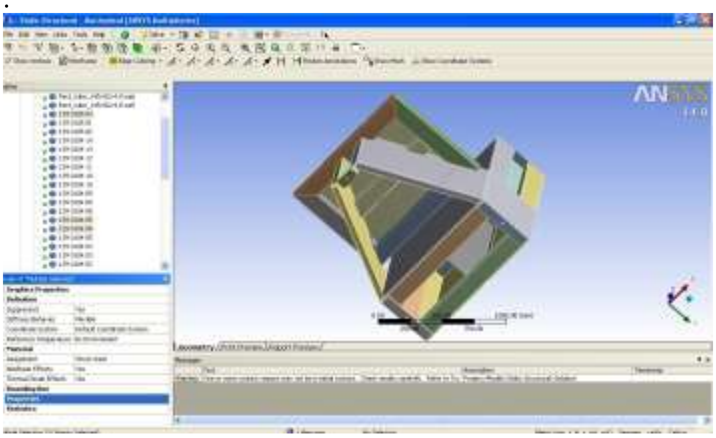

Figure 2: Geometry imported in the Ansys workbench For analysis

Loading Conditions:

Total mass of the assembly = 542.58 Kg

Table Assembly has been analyzed for the following

Loading conditions:

a. Static Load of 3500 Kg of the Job on it and The weight of T-Slot 224 kg, which is Equally distributed on each 9 pads.
b. Self weight of the assembly is considered to Be a standard earth gravity load and is Applied through the CG of the assembly.

Assumptions:

There is bonded face connection between all bodies. Various positions of Tool machine is not consider. Material of all the parts in main assembly, including the tool machine and machining part is assumed to be Structural Steel.

II. SELECTION OF ELEMENT TYPE

Selecting element type for FEA model is the most important decision in analysis, because element represent the actual properties of the material. We select Hex8 (Hexahedral shaped, 3 Degree of Freedom, linear shape element for analysis. The load and the constraints are applied on the mater node which is connected to the spring by 1D rigid RBE2

And RBE3 element as shown in the figure below

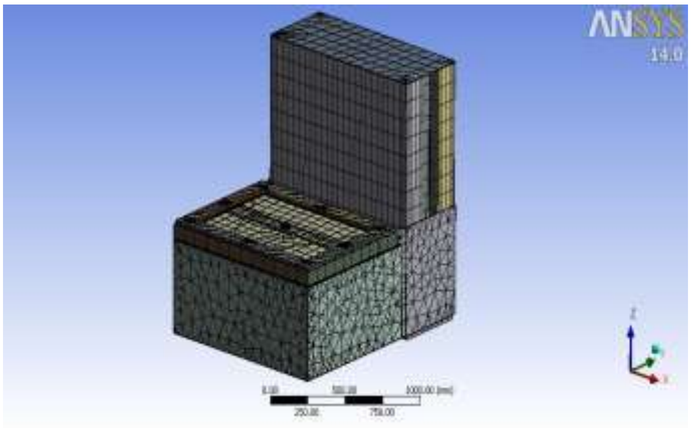

Figure 3: Hexahedral mesh model

Meshing details

Solid187 and Solid 185 element was used to create FE model. Mesh grading was done to control number of element as shown in figure 3.

A Static load of 3500 kg and 224 kg of T-Slot equally distributed on each pad, at shown location (A, B, C, D, E, F, G, H, I) as shown in figure 4.

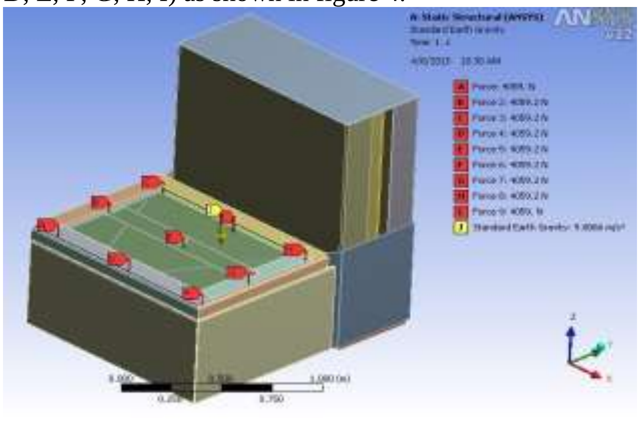

Figure 4: Load application on different pad faces with Gravity load

III.STATIC STRUCTURAL ANALYSIS METHOD

Static Structural Deformation Result:

A static structural analysis is carried out with the given loading condition in the ANSYS solver. Preprocessing of helical compression spring is done by using CATIVA V5R18 software. Where the 3D hexahedral mesh is done and the input deck is prepared for ANSYS solver. The result is shown in figure 5 and 6.

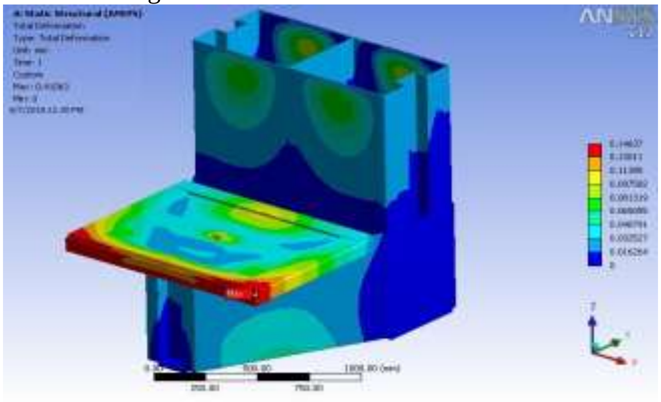

Figure 5: Maximum deformation at table end support is 0.146 mm

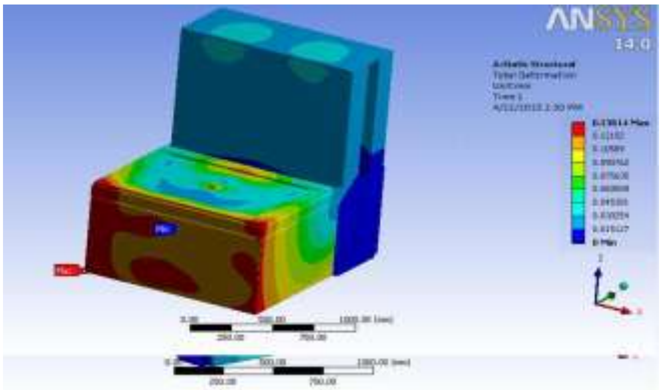

Figure 6: Maximum deflection of EDM table bed location

Static Structural Stress Result:

The critical region of failure is having maximum deformation of 0.146 mm and stress at the work table is 96.81 MPa will are much less then the yield stress of the material as shown in figure 7.

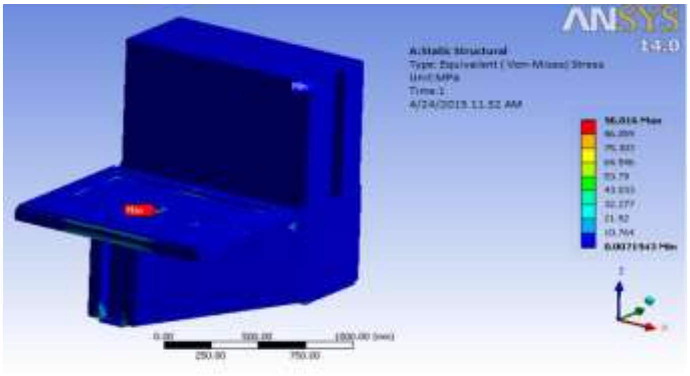

Figure 7: Maximum von Mises stress in EDM table Original design

Reaction Forces:

Total reaction forces at the fixity must be equal to the total load applied on the table assembly to validate the result for static stress condition is shown in figure8.

Details of "Force Reaction"	
Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Fixed Support
Orientation	Global Coordinate System
Options	
Result Selection	All
Display Time	End Time
Results	
X Axis	-3.9038e-005 N
Y Axis	5.6456e-005 N
Z Axis	49702 N
Total	49702 N

Figure 8: force reaction

IV. DESIGN MODIFICATION

The original design shows the maximum deformation of 0.146 mm and maximum von Misses stress as 96.81MPa. The modification is suggested to enhance the strength of the table support channel for different thickness variation. Hence a design modification with 5mm, 4mm and 3 mm channel section is used to predict the deformation and stress for a given loading condition.

FEM OF MODIFIED CHANNEL SECTION THICKNESS:

EDM work table was modeled using commercial software and all the specification was accordingly followed the relevant modification. The support channel thickness is modified from 6mm to 3 mm to predict the stress and deformation result as shown in Figure 9.

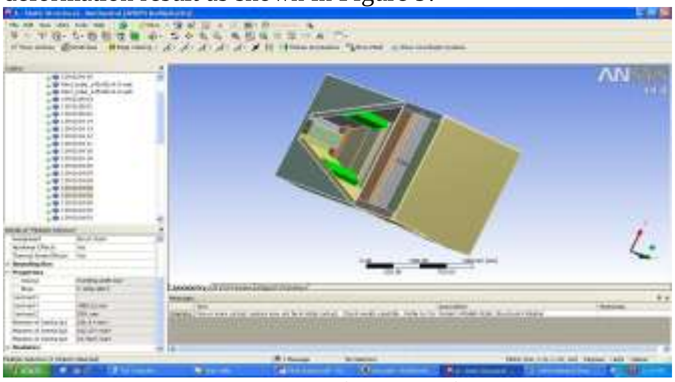

Figure 9: Bottom channel support modified to 5mm for Analysis

The modified design of table is modeled using UNIGRAPHICS software. The inner channel section for 5mm was modeled. The meshing is done again in the Ansys workbench to have the static load case condition result. FE Model of modified Design1 (5mm thickness channel): The loads are applied on the 9 pad as equally distributed load as that of original design loading condition as shown in figure 10.

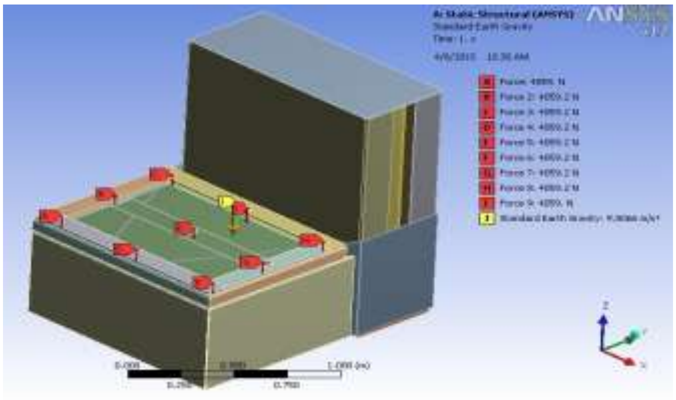

Figure 10: Loads and Boundary conditions for modified design

Static Structural Analysis Result:

Preprocessing of EDM table is done by using Ansys workbench software. Where the 3D hexahedral meshing is done on sweep able volume and for non sweep able volume a tetrahedral 3D element is used as shown in the figure 11.

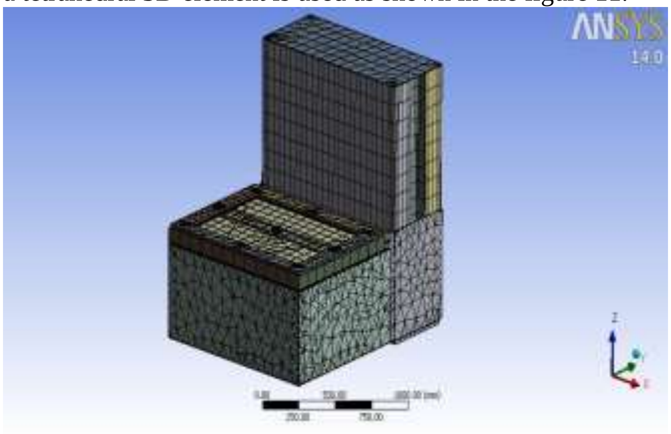

Figure 11: 3D solid mesh of the table assembly

A static structural analysis is carried out with the given loading condition in the ANSYS solver. The result for maximum deformation and maximum stress are plotted as shown in the figure 12 and 13.

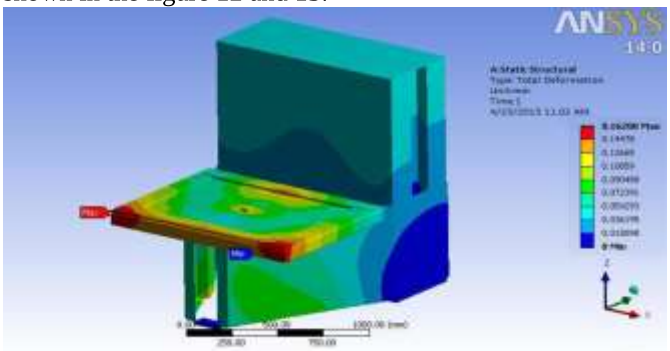

Figure 12: Table top deformation for 5mm thickness Channel

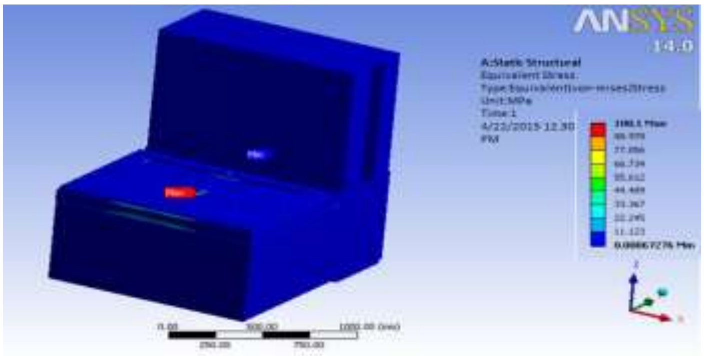

Figure 13: Maximum von Mises stress at 5mm channel Thickness

FE Model of modified Design2 (4 mm thickness channel):
A static structural analysis is carried out with the given loading condition in the ANSYS solver. The result for maximum deformation and maximum stress are plotted as shown in the figure 14 and 15.

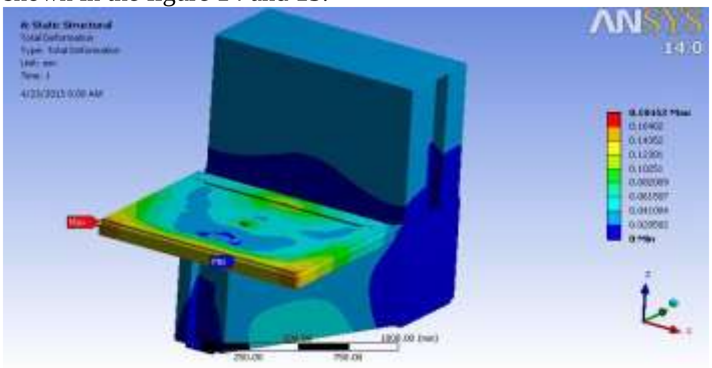

Figure 14: Table top deformation for 4mm thickness Channel

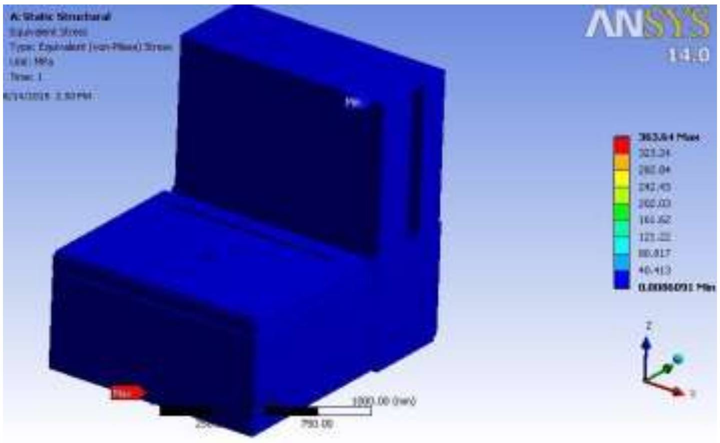

Figure 15: Maximum von Mises stress at 4mm channel Thickness

FE Model of modified Design3 (3 mm thicknessChannel):
A static structural analysis is carried out with the given loading condition in the ANSYS solver. The result for maximum deformation and maximum stress are plotted as shown in the figure 16 and 17.

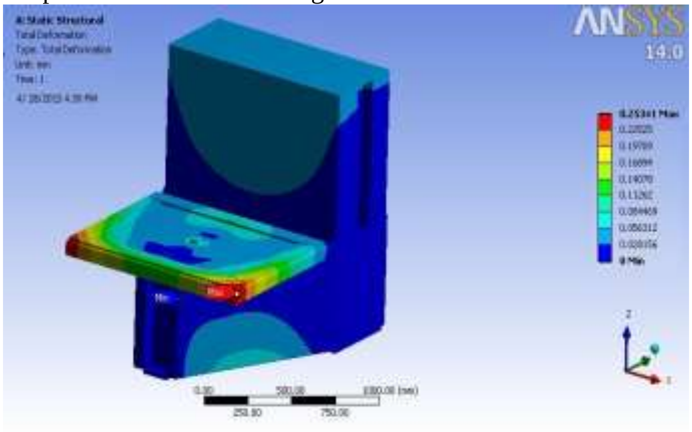

Figure 16: Table top deformation for 3mm thick Channel

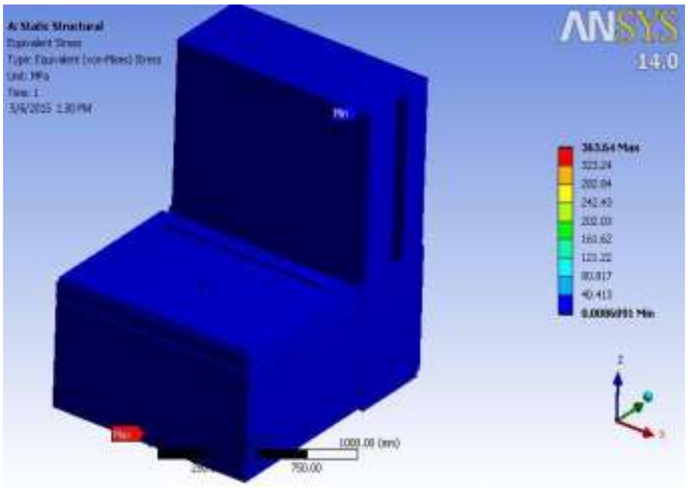

Figure 17: Maximum von Mises Stress at 3mm Thickness channel

V. RESULT

RESULT COMPARISION
EDM table is modified according to the various support channel thickness from 6mm to 3mm and the results are compared in tabular form as shown table 2.

Table 2: Result comparison between original and Modified Design

Sr. No.	Compression EDM table	Original 6mm	Modified Design		
			Channel thickness 5mm	Channel thickness 4mm	Channel thickness 3mm
1	Maximum Deformation of the EDM table (mm)	0.146	0.162	0.184	0.253
2	Maximum von-Mises Stress in the EDM table (MPa)	96.81	100.1	235.35	363.64

RESULT AND DISCUSSIONS

- Original EDM table assembly shows the maximum deflection of 0.146 mm at the table location which is well within the permissible limiting 2 mm.
- Maximum von Mises stress is observed to be 96.81 MPa near the centre pad of EDM work table.
- Maximum von Mises stress is observed at the bed location in the original design is to be 56.14 MPa.
- The modified design with 5mm thickness shows the maximum deformation of 0.253 mm and von Mises stress of 363.64 MPa which is less than the yield stress of the material.
- The modified design with 3mm thickness shows the good acceptance value to reduce the mass of the table assembly

VI. CONCLUSION

- The original design deformation is well within the permissible limit of maximum 2 mm.
- Maximum von Mises stress value is within the permissible limit of the yield stress of the material.
- The modified table design shows the deformation is well within the acceptable limit of the requirement
- Maximum von Mises stress value is well within the permissible limit of the material
- Modified design can be implemented in the new version of EDM table manufacturing.
- The Simulation results show good agreement with actual test data available
- The maximum table deformation though increased, which means that the table stiffness is over design can be reduced to the 3mm channel section.

REFRENCES

1. Adam A. Cardia, Hiram A. Firpia, Matthew T. Bementb,Steven Y. Liang, “Workpiece dynamic analysis and prediction during chatter of turning process” a WoodruffSchool of Mechanical Engineering, Georgia Institute ofTechnology, Atlanta, GA 30332-0405, USA, MechanicalSystems and Signal Processing 22 (2008) 1481–1494.
2. . Taik-Min Lee a,*, Dong-Yoon Lee b,1, Ho-Cheol Lee c,2,Min-Yang Yang d, “Design of cam-type transfer unitassisted with conjugate cam and torque control cam”, 3aKorea Institute of Machinery and Materials, 171 JangdongYusonggu, Daejeon 305-343, Republic of Korea.,Mechanism and Machine Theory 44 (2009) 1144–1155.
3. Daniel Kanaan, Philippe Wenger, Damien Chablat,“Kinematic analysis of a serial–parallel machine tool: TheVERNE machine” Institut de Recherche enCommunications et Cybernétique de Nantes, UMR CNRS6597, 1, rue de la Noë, BP 92101, 44312 Nantes Cedex 03,France., Mechanism and Machine Theory 44 (2009) 487–498
4. J.R.Bakera; K.E. Rouchba Department of Mechanical Engineering, “Use of Finite element structural models in analyzing machine tool chatter”, University of Kentucky—Extended Campus, Paducah, KY, USA., Finite Elements inAnalysis and Design 38 (2002) 1029–1046.
5. Alberto Carpinteri, Marco Paggi, “Numerical analysis of fracture mechanisms and failure modes in bi-layeredstructural components”,Department of Structural andGeotechnical Engineering, Politecnico di Torino, CorsoDuca degli Abruzzi 24, 10129 Torino, Italy, FiniteElements in Analysis and Design 43 (2007) 941 – 953.
6. R. Neugebauer1 (1), B. Denkena2 (2), K. Wegener3 (2), “Mechatronic Systems for Machine Tools”,Fraunhofer-Institute for Machine Tools and Forming Technology IWU,Chemnitz, Germany Institute of Production Engineeringand Machine Tools (IFW), Hannover University, Garbsen,Germany.
7. Y.H. Lee, C.L. Li “Automation in the design of EDM electrodes”, Department of Manufacturing Engineering and Management Hong Kong,41 (2009) 600_613.

