

Design of Punching Tool and Stress Analysis Over Sheet Shaping Process

Sk. Mastan Vali¹, A. Siva Naga Raju²

¹M. Tech. Student, Dept. of Mechanical Engineering, Chinthalapudi Engineering College, Hyderabad, India

²Assistant Professor, Dept. of Mechanical Engineering, Chinthalapudi Engineering College, Hyderabad, India

Abstract: As technique for punching in oval shapes is not promptly accessible, squeeze apparatus for punching in oval shapes is required in different enterprises as per their applications. There are squeeze instruments for punching fit as a fiddle, however when shapes other than roundabout shape are craved, they need to configuration as per measurements required by industry. As application for oval shape punching varies from industry to industry since measurements required by businesses vary from each other. As a few enterprises require more than one oval molded gap that will be done in single stroke for diminishing the time required for creation, there is have to plan such press instrument which can perform such operation. Henceforth to have quick generation of required oval size openings, there is have to build up a press device gathering which can make precise oval gaps on metal sheets. For outlining such get together, it is exceptionally important to examine each part of existing plan of press apparatus with the goal that progressions required can be effortlessly characterize. Correlation of existing outline with the proposed empowers us to discover redress method for plan.

Keywords: Punching device, Sheet metal; Sheet metal assembling; Sheet metal components; Selection of material

1. Introduction to punching productions

Punching process is progressively utilized as a part of produce industry. Punching is among the most vital sheet metal in assembling process in large scale manufacturing of metal parts and segments. This operation has an extraordinary effect in assortment of ventures, for example, car industry. Lately, a further comprehension of the innovative parts of the punching procedure has been picked up particularly in punching instruments [1].

- Punching is the most financially savvy procedure of making openings in strip or sheet metal for normal to high creation.
- It can make numerous molded gaps.

As strategy for punching in oval shapes is not promptly accessible, squeeze instrument for punching in oval shapes is required in different ventures as indicated by their applications. There are squeeze instruments for punching fit as a fiddle, yet when shapes other than round shape are sought, they need to configuration as indicated by measurements required by industry. The cost of tooling in sheet metal ventures contributes an extensive part to the general cost of assembling a segment. It is hence basic to keep down this cost by guaranteeing that the

device works for a long stretch underway without intrusion. One method for accomplishing this goal is to decrease the weight on the apparatus amid punching.

A. Highlight arrangement in punching gaps in sheet metal

The arrangement of cutting and punching is a seven-organize process. Each stage includes settling on various decisions with respect to the item. These decisions ought to be made in view of shapes and components made in the outline organize. It is in this way a well ordered process towards the choice of an assembling strategy. All conditions included are at last contrasted with the conceivable outcomes of the procedures. In view of the correlation, certain assembling forms are proposed to the client for assembling the part being referred to. Now, the decision is construct exclusively in light of mechanical perspectives. Cost factors, which assume an essential part in the process determination, can be associated with the correlation after that. A large portion of the current PC supported bite the dust plan frameworks have still not completely managed the center pass on configuration issue of material determination of press device segments. Some current CAD/CAM frameworks can create bill of materials, however these frameworks don't consider the accessibility of other reasonable materials for the decision of client for better execution of press instrument segments and henceforth the long existence of press device. Further, these frameworks don't have even information base comprising of experienced learning of space specialists in material determination of press instrument parts. In choosing materials, architects and designers need to consider countless.

B. Choice of materials for squeeze apparatus parts

The choice of materials for squeeze apparatus parts for a given application rely upon which bite the dust disappointment components rules. It requires something beyond learning of materials properties. The step-by-step manual procedure of material determination is researched amid mechanical visits and discourse with kick the bucket plan specialists and is given as under.

Step 1. Recognize the sort of wear: This is the most central stride since it will figure out which wear resistance profiles the kick the bucket material ought to have. The accompanying elements are considered to set up the ruling wear (rough, glue or blended) not out of the ordinary:

- Type of sheet material
- Hardness of sheet material
- Presence of hard particles in the sheet material.

Step 2. Event of chipping or plastic disfigurement: The accompanying elements decide the degree of the hazard for chipping or potentially plastic twisting, i.e. regardless of whether high pliability and additionally high hardness are required:

- Type of operations to be performed on squeeze device
- Thickness and hardness of sheet material
- Geometrical many-sided quality of parts to be created.

The kick the bucket creator typically utilizes his huge experience here.

Step 3. Danger of splitting: The accompanying elements give a sign of the hazard for breaking, i.e. regardless of whether extreme material or potentially direct hardness levels must be utilized:

- Type of operations to be performed on squeeze device
- Geometry of part to be delivered
- Die outline and kick the bucket measure
- Thickness and hardness of sheet material

2. Literature review

The title plan and manufacture of a Punch toss and Precision Hole Punch requires a measure of good comprehension on the information of the CNC Turret Punch Machine. In this way, executing an exploration is important to get all the data accessible and identified with this theme.

A. Presentation

The machine utilized as a part of FKM lab is the Traumatic 2020R FMC display from Trump Organization. Traumatic 2020R FMC is a high-accuracy facilitate holing sheet-handling focus with numerical program control. It has: fast of removing and a pressure driven cutting head with 360° revolution and also a simple to-utilize straight device cushion. A proficient in the field of cutting produces parts without scratches and with high exactness and consistency. It is fabricated in the year 2005 in Germany. The most extreme punch limit of this machine is 180kN. The instruments related with this machine are normally produced using High Speed Steel (HSS) material and from Trump of Organization.

B. Wording

1) Turret punch machine

Turret Punch is broadly stated as CNC Turret Punch Press and generally known as NCT or punch machine is a numerical controlled plate handling machine. Charts are changed into CNC programs through programming or CAD/CAM programming can complete the handled things in brief time regardless of how muddled the plated thing is. It is significantly more productive than the conventional punch machine which requires shaping procedure. It is additionally utilized for bowing, punching and shaping sheet metals.



Fig. 1. Turret punch machine

2) Sheet metal

Sheet metal is essentially metal framed into thin and level pieces. It is one of the principal shapes utilized as a part of metalworking, and can be cut and twisted into an assortment of diverse shapes. Incalculable ordinary articles are built of the material. Thicknesses can change fundamentally, albeit to a great degree thin thicknesses are considered thwart or leaf, and pieces thicker than 6mm (0.25in) are considered plate. Sheet metal is accessible as level pieces or as a wound strip. The loops are shaped by running a ceaseless sheet of metal through a roll slitter. The thickness of the sheet metal is called its gage. The gage of sheet metal extents from 30 gage to around 8 gage. The higher the gage, the more slender the metal is.

3) Punching

Punching is performed by moving the sheet metal between the best and base instruments of a punch. The best device (punch) mates with the base instrument (kick the bucket), cutting a straightforward shape (e.g. a square, circle, or hexagon) from the sheet. A zone can be removed by making a few hundred little square cuts around the border. A punch is less adaptable than a laser for cutting compound shapes, yet quicker for tedious shapes. A common CNC punch has a decision of up to 60 instruments in a "turret" that can be turned to bring any apparatus to the dynamic punching position. A present day CNC punch can take 600 blows for each moment.

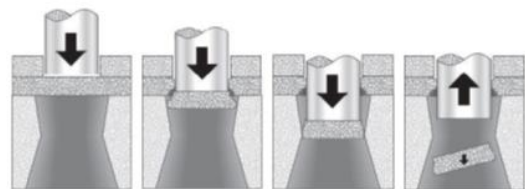


Fig. 2. Punching process

3. Guidelines

A. Modeling software catia

CATIA V5 is the main answer for item achievement. It tends to all assembling associations. CATIA can be connected to a wide assortment of businesses, from aviation, car, and modern hardware, to gadgets, shipbuilding, plant outline, and shopper products. Today, CATIA is utilized to outline anything from a

plane to adornments and attire. With the power and useful range to address the total item advancement process, CATIA bolsters item building, from beginning particular to item in-benefit, in a completely coordinated way. It encourages reuse of item plan information and abbreviates improvement cycles, helping undertakings to quicken their reaction to advertise needs.

B. Production of solid/surface bodies

We can make strong bodies by cushioning the outline geometry to make cooperative elements or Creating primitives for the essential building pieces, at that point including more particular components (for instance, gaps and spaces and so on. Padding the portray and non-outline geometry lets us to make a strong body with complex geometry. This strategy additionally gives us add up to control over the altering of the body. Altering is finished by changing the cleared creation parameters or by changing the portray. Altering the outline makes the cleared element refresh to coordinate the portray. Spruce up highlights are utilized to adjust the part bodies as indicated by given determinations these are the most vital components to alter the items.

C. Modeling procedure

Open catia software starts menu – mechanical design – part design – tools – options – select units as mm. It opens part module. Then select – front plane draw a sketch with required lengths and exit sketcher, as shown in below fig

- Die
- Sheet
- Punch

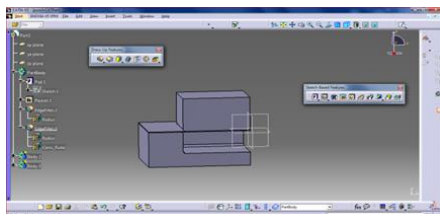


Fig. 3. Assembly of punch die and sheet

4. Introduction of ANSYS

ANSYS a product of ANSYS inc. Is a world's leading, widely distributed and popular commercial CAE package. It is widely used by designers/analysis in industries such as aerospace, automotive, manufacturing, nuclear, electronics, biomedical, and much more. ANSYS provides simulation solution that enables designers to simulate design performance directly on the desktop. In this way, it provides fast efficient and cost efficient product development from design concept stage to performance validation stage of product development cycle.

A. Project Objectives

After complete of this chapter,

- The basic concept and general working of FEA
- Understand the advantage and limitations of FEA

- Understanding the analysis type
- Understanding important terms and definition of FEA

The finite element analysis (FEA) is computing technique that is used to obtain approximately. Solution to boundary value problems. It uses a numerical method called finite element method(fem).in FEA involves the computer model of a design that is loaded and analysed for specific results, much as stress, deformation, deflection, natural frequencies, mode shapes, temperatures distribution and soon.

B. General procedure

To Conduct Finite Elements Analysis. To conduct the finite elements analysis you need to follow certain steps that are given next.

- Set the type of analysis to used.
- Create model
- Define the elements type
- Divide the elements types
- Divide the given geometry into nodes and elements
- Apply material properties and boundary conditions
- Drive elements matrices and equations
- Solve the unknown parameters at nodes
- Interpret the results

The general process FEA by using software is divided into three main phases:

C. Advantage and limitations of fea software

Following are some of the advantage and laminations of FEA software. Advantage

- It reduces the amount of prototype testing, thereby saving the cost and time.
- It gives the graphical representation of the results of analysis
- The finite elements modelling and analysis are performed in the preprocessor and solution phases which if done manually would consume a lot of time and in some cases, might be impossible to perform.
- Variables such as stress and temperature can be measured at any desired point of the model
- It helps optimise a design
- It is used to simulate the designs that are not suitable for prototype testing.
- Its helps you create more reliable, high quality, and competitive designs.

D. Limitations

- It does not provide exact solutions
- FEA package are costly
- An inexperienced user can deliver incorrect answer, upon which expensive decision will be based
- Results give solutions but not remedies
- Feature such as bolts, welded joints, and so on cannot be accommodated to model. This may lead to approximation and errors in the results.

- For more accurate results, more hard disk space, ram, and time are required.

5. ANSYS Workbench

A. Project objective

In this paper, we will be able to define:

- Understand the types of system
- Understand different types of cells
- Understand the graphic user interface of the workbench window
- Start a new project in ansys workbench windows
- Add the first and subsequent analysis system to a project
- Set units for the project

1) Starting ansys workbench 16.0

The workbenches windows help streamline an entire project to be carried out in ansys workbench 14.0. In this window, one can create, manage, and view the workflow of the entire project creates by using standard analysis system. The workbench windows mainly consist of the menu bar, standard toolbar, the toolbar windows, project schematic windows, and the status bar.

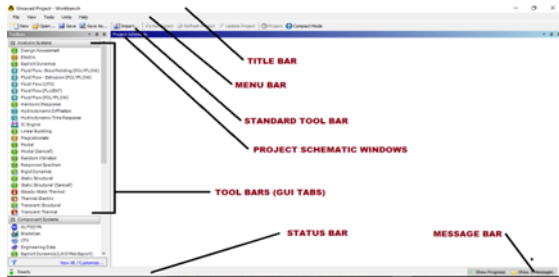


Fig. 4. The component of the workbench windows

B. Toolbox windows

The toolbox windows are located on the left in the workbench windows. The toolbox windows list the standard and customized templates or the individual analysis components that are used to create a project. To create a project, drag a particular analysis or component system from the toolbox window and drop into the project schematic windows or double click on GUI table it will add it into project schematic windows and to create the project.

C. Project schematic windows

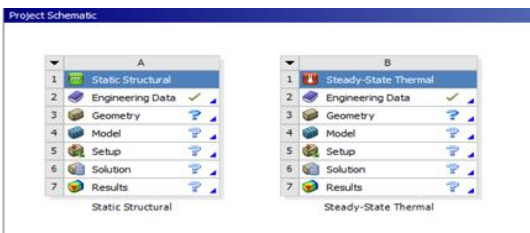


Fig. 5. Static analysis imported into project schematic

The project schematics windows help manage an entire project. It displays the workflow of entire analysis project. To

add an analysis system to the project schematic windows, drag the analysis system from toolbox windows and drop into the green coloured box displayed in the project schematic windows.

D. Custom system analysis

Table 1
Analysis and definitions

Name of analysis	loads	Solution determines
Static structural	Static load conditions	Deformation, Stresses and Strains, Fatigue tool, Life, Damages, Safety factor

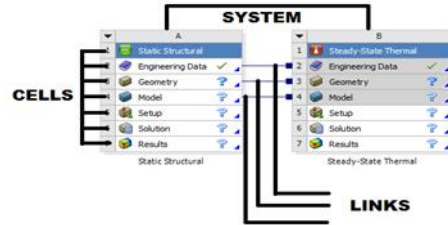


Fig. 6. Sharing of engineering data, geometry and model

6. Static structural analysis

A. Project objective

In this project, we will be able to define total deformation and stress, etc.

- Create the static structural analysis system
- Apply different types of materials
- Applying of boundary conditions
- Apply a different type of constraints
- Apply different loads
- Generate the results as per required
- Generate project reports

In this project, we imported the geometry of the component show the dimensions for the component with respect to the load applications. The material to be applied on the model is Stainless Steel. Next, you will run the analysis under two conditions and evaluate the Total Deformation, Directional Deformation, Equivalent Stress, Maximum Principal Stress, and Minimum Principal Stress.

B. Introduction to static structural analysis

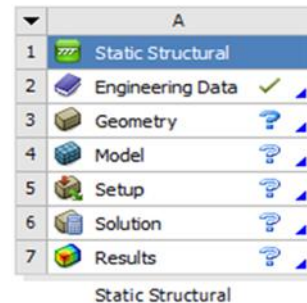


Fig. 6. The static structural analysis system added to the project window

The Static Structural analysis is one of the important analyses in ANSYS Workbench. It is available as Static Structural analysis system under the Analysis System toolbox in the

Toolbox window, this system analyses the structural components for displacements (deformation), stresses, strains, and forces under different loading conditions. The loads in this analysis system are assumed not to have damping characteristics (time dependent). Steady loading and damping conditions are assumed in this type of analysis system.

As discussed in previous sections, analysis can be carried out in three major steps: pre-processing, solution, and post-processing. The tools required to carry out these steps are discussed next.

C. Project overview

In this project, you will create the model, as shown below. The dimensions to create the model and its boundary and loading conditions are also given in the same figure. Run a Static Structural analysis on the model and evaluate the Total Deformation and the Directional Deformation. Determine Directional Deformation along the X, Y, and Z axes. After evaluating the results, interpret them.

- Start a new project and create the model.
- Generate the mesh.
- Set the boundary and loading _conditions.
- Solve the model.
- Duplicate the existing analysis system.
- Interpret results.
- Save the project.
- Starting a new project and creating the model
- Adding the material to the engineering data
- Table of the composite material
- Specifying the boundary conditions

D. Generating the Mesh

After the model is created in the Design Modeler window, you need to generate the mesh for the model in the Mechanical window. Choose the Generate Mesh tool from the Mesh drop-down in the Mesh contextual toolbar; the mesh is generated.

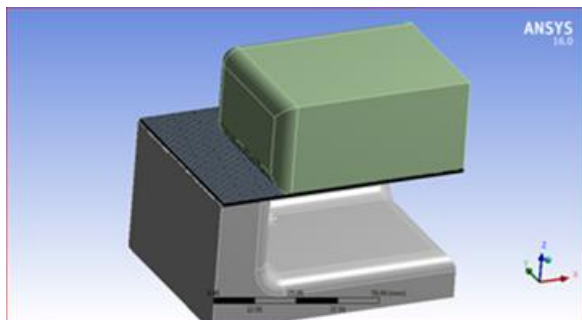


Fig. 7. Mesh generated with default mesh controls

Table 2
Material

S. No.	Material	Density	Young Modulus	Poison Ratio
1	Al Alloy	3720	3422.4	0.3
2	Steel	7480	504.152	0.29
3	Iron	7870	15.74	0.29
4	Copper	3180	10.176	0.23

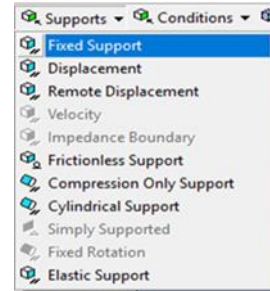


Fig. 8. Choosing the fixed support tool from the supports drop-down

After you mesh the model, it is required to specify the boundary and loading conditions. Choose the Fixed Support tool from the Supports drop-down in the Environment contextual toolbar; Fixed Support is added under the Static Structural node. Also, the Details of “Static Structural” window is displayed.

Choose the Force tool from the Loads drop-down in the Environment contextual toolbar; Force is added under the Static Structural node in the Tree Outline. Also, the Details of “Force” window is displayed.

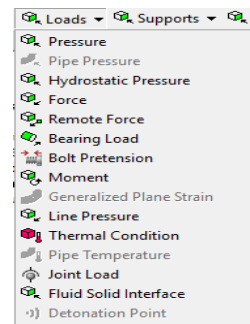


Fig. 9. The loads drop-down

E. Solving the fea model and analyzing the results

After the boundary and load conditions are specified for the model, you need to solve the analysis. After solving, you will get the Total and Directional Deformations due to the given condition. Also, you will get Equivalent Stress, life, and damage. Choose the Total tool from the Deformation drop-down of the Solution contextual toolbar; Total Deformation is added under the Solution node.

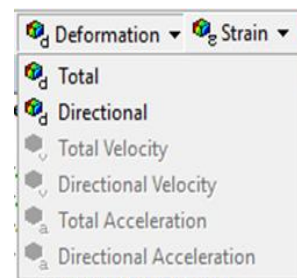


Fig. 10. Choosing the total tool from the deformation drop down

Choose the Equivalent (von-misses) tool from the Stress drop-down in the Solution contextual toolbar.

7. Results

A. Total deformations

In the Details of “Total Deformation” window, expand the Results node, if it is not already expanded. Note that the maximum and minimum deformations displayed are respectively. In the Details of “equivalent stress” window, expand the Results node, if it is not already expanded. Note that the maximum and minimum deformations displayed are respectively.

B. Stainless steel

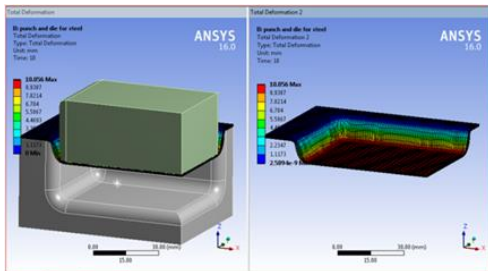


Fig. 11. The values of total deformation obtained from the legend

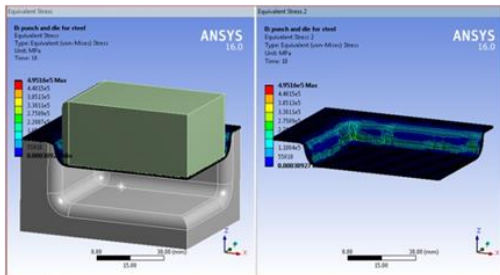


Fig. 12. The equivalent stress obtained from the display in color bands.

C. CIS

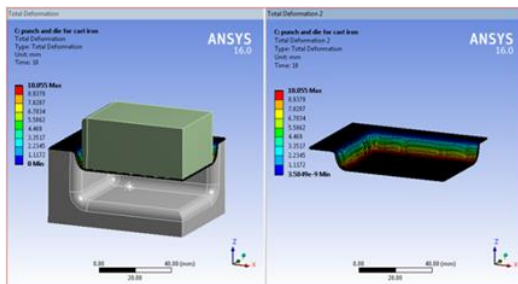


Fig. 13. The values of total deformation obtained from the legend

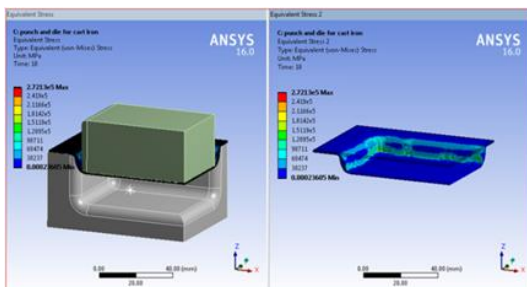


Fig. 14. The equivalent stress obtained from the display in color bands

D. Al Alloy

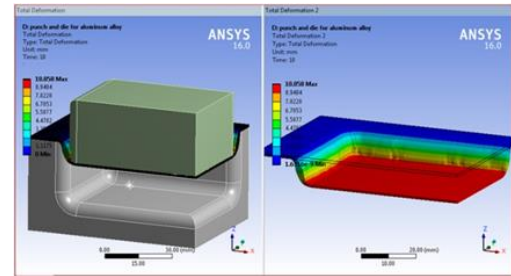


Fig. 15. The values of total deformation obtained from the legend

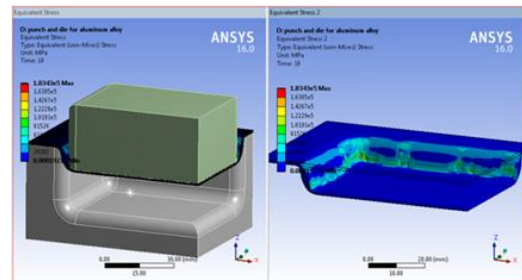


Fig. 16. The equivalent stress obtained from the display in color bands.

E. Cu Alloy

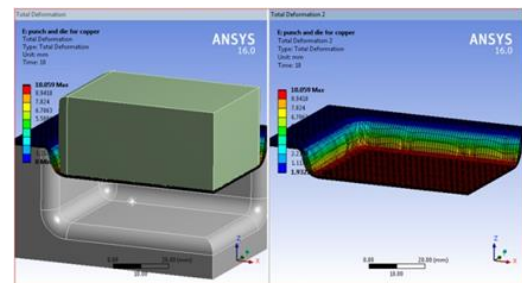


Fig. 17. The values of total deformation obtained from the legend

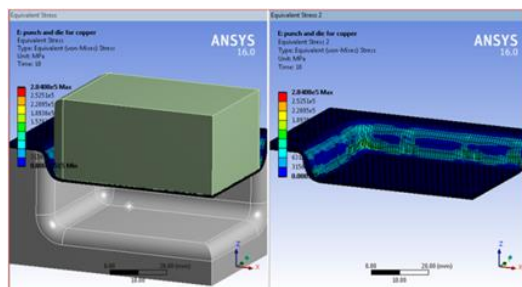


Fig. 18. The equivalent stress obtained from the display in color bands

Close the existing Mechanical window; the Workbench window is displayed. A body is called to be deformed if its shape is changed temporarily or permanently. The temporary change of shape is known as elastic deformation and a permanent change of shape is known as plastic deformation. In ANSYS Workbench, You can determine deformation in terms of Total and Directional Deformations.

8. Results and graphs

Table 4
 Total deformation, stress and weight

Total Deformation mm					
S. No.	Distance	Steel	CI	Al Alloy	Cu Alloy
1	10.00	10.06	10.06	10.06	10.06
Equivalent Stress Pa					
Distance	Steel	CI	Al Alloy	Cu Alloy	
1	10.00	495160	272130	183430	284080
Weight Kg					
	Steel	CI	Al Alloy	Cu Alloy	
1		0.0377	0.0346	0.0133	0.0399

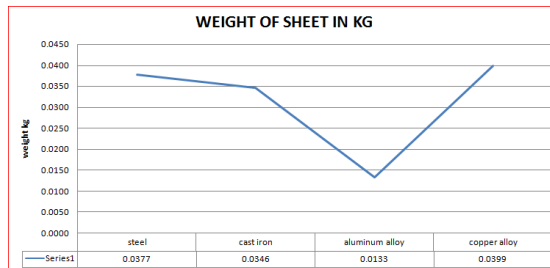


Fig. 19. Graph showing weight of sheet

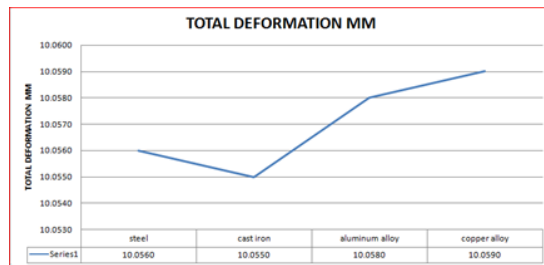


Fig. 20. Graph showing total deformation

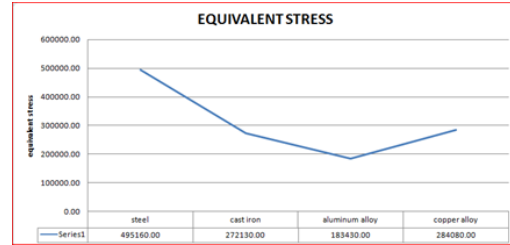


Fig. 21. Equivalent stress

9. Conclusion

From the straightforward perception that behind each state of a sheet metal item lays a limited gathering of machines, we can find that sheet metal shapes can be arranged into assembling highlights. Accordingly, each shape or highlight relates to a component class or highlight. By get ready principles and limitations for the element classes and elements, the outline of sheet metal items can be made additionally fabricating amicable. Information for determination of materials of press instrument segments gotten from kick the bucket originators, device makers, pass on plan handbooks what's more, mechanical leaflets has been broke down, arranged and fused into an arrangement of creation rules. The strategy of improvement of proposed astute framework has been depicted finally. The framework is fit for giving master exhortation on choice of materials of dynamic and dormant segments of press apparatus. There are very few press devices accessible for punching in shapes in the present situation. As application for shape punching varies from industry to industry since measurements required by ventures contrast from each other. As a few ventures require more than one formed opening that will be done in single stroke for lessening the time required for creation, this basic outline of Press Tool gathering empowers to

Table 3
 Total deformation and stress as per distance

Total Deformation mm					
S. No.	Dist. mm	Steel	CI	Al Alloy	Cu Alloy
1	1	1.0027	1.0026	1.0029	1.0029
2	2	2.0581	2.0594	2.056	2.056
3	3	3.0366	3.0366	3.0366	3.0366
4	4	4.0365	4.0366	4.0399	4.0399
5	5	5.0356	5.0357	5.0362	5.0357
6	6	6.0398	6.0404	6.0398	6.0399
7	7	7.0501	7.0499	7.0553	7.0538
8	8	8.0512	8.0508	8.0548	8.0557
9	9	9.0518	9.0517	9.0563	9.0577
10	10	10.056	10.055	10.058	10.059
Equivalent Stress Mpa					
S. No.	Distance mm	Steel	ci	Al Alloy	cu Alloy
1	1	13139	7117.1	4782.9	7477
2	2	50461	27752	18432	28858
3	3	73470	41197	25263	38868
4	4	79372	44602	27184	41648
5	5	72226	40681	25405	38914
6	6	1.76E+05	1.01E+05	64172	99605
7	7	2.30E+05	1.26E+05	85414	1.32E+05
8	8	3.19E+05	1.75E+05	1.18E+05	1.83E+05
9	9	4.07E+05	2.23E+05	1.51E+05	2.34E+05
10	10	4.95E+05	2.72E+05	1.83E+05	2.84E+05

perform such operation with security and less potential outcomes of scrap. In the wake of examining the punch in Ansys workbench 16 comes about demonstrates that there is less anxiety creating in punch that guaranteeing this outline is protected.

References

- [1] Mohd Khairul Mizan Bin Nasir (2009) Investigate the impact of shear point of fast steel punching apparatus in punching process. Postulation, Faculty of Mechanical Engineering, Universiti Malaysia Pahang.
- [2] U.P. Singh (1992) Design investigation of the geometry of a punching/blanking device. *Diary of Materials Processing Technology* 33: 331-345.
- [3] J. Ch. Lin, WS Lin, KS Lee, JL Tong (2008), The ideal leeway outline of miniaturized scale punching bite the dust. *Diary of Achievements in Materials and Manufacturing Building* 14.
- [4] M.Lohtander, J. Varis. Assembling Features in Cutting Shapes and Punching Holes in Sheet Metal. Nineteenth International Conference on Production Research, Department of mechanical building, Lappeenranta University of Technology, Lappeenranta, Finland.
- [5] S. Kumar (2011) An Intelligent System for Selection of Materials for Press Tool Components. *Diary of Engineering Research and Studies* 2: 119-130.
- [6] 'Principal of Tool Design' by American Society of Tool and assembling engineers.
- [7] 'Apparatus plan' by Cyril Donaldson Tata McGraw slope third release.