CASE STUDY-1

SIMULATION OF A WEIGHING SCALE

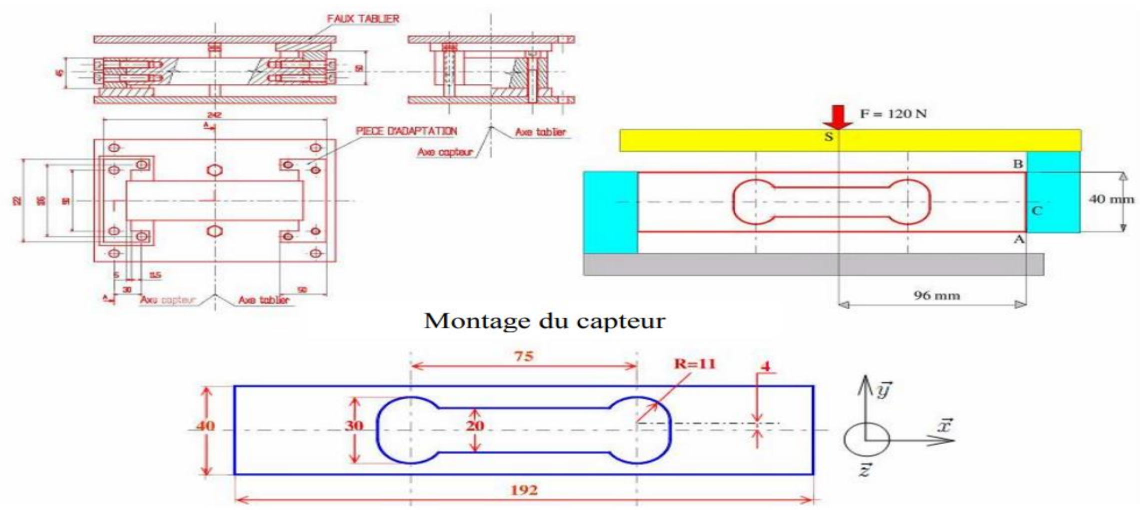
1. INTRODUCTION

The Weigh scale is an electronic device to measure weighs of various things. It consists of a load cell, the base and the electrical or mechanical connections. The load cell consists of various types such as: strain gauge, hydraulic, pneumatic and other load cells. They work on the principal of change in deformation for load cells and in turn provide them in electrical signals for output of weights.

In the present case study, the load cell made of aluminium alloy is subjected to static structural analysis. The modelling choices are to set up a 2D model as well as a 3D model. The simulation is carried out under steady conditions, post which the results are presented and analysed.

2. DESIGN

The sketch used for this load cell has been shown in Figure 1. The part has been created in 3D shell element and also in 3D solid for 2D and 3D configurations respectively in analysis software Abaqus 2020. The sketch has been designed in mm and respective material properties are inserted such as Young's Modulus as 74000 MPa and Poison's ratio as 0.3.



CASE STUDY-1

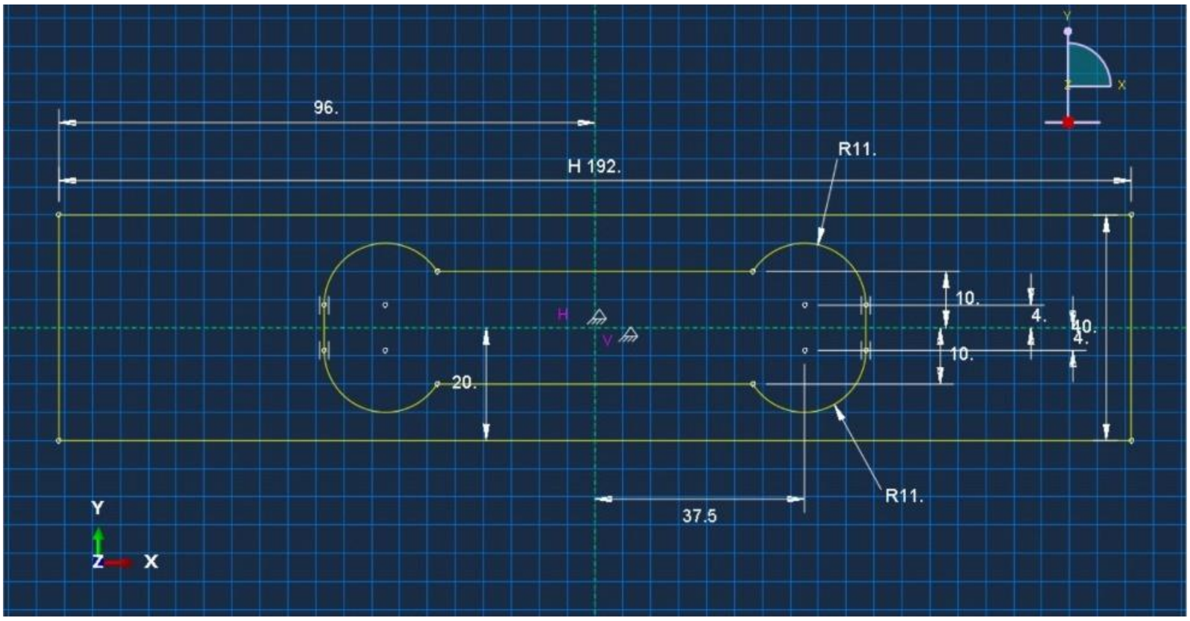


Fig. 1: Sketch of the load cell in mm

During analysis many assumptions are made before the analysis. Some of them are:

- The left part of the sensor is completely fixed given the arrangement of the base of the weight sensor.
- In order to apply an equivalent mechanical load without modelling other surrounding parts (only the sensor part), we choose to apply: a) A downward vertical surface load of 120N (y-axis, negative value). b) Two linear forces applied to the horizontal edges of this surface: - a horizontal load along –x axis with an intensity of 288N (upper edge) - a horizontal load along +x axis with an intensity of 288N (lower edge)
- The longitudinal deformations are measured by the strain gauges located in points A and B

The entire model has been divided into 2 partitions and contain points A & B where the sensors are placed to determine the deformation. The Figure 2 represents the partitions and mentions the points A & B where all properties need to be determined after analysis.

CASE STUDY-1

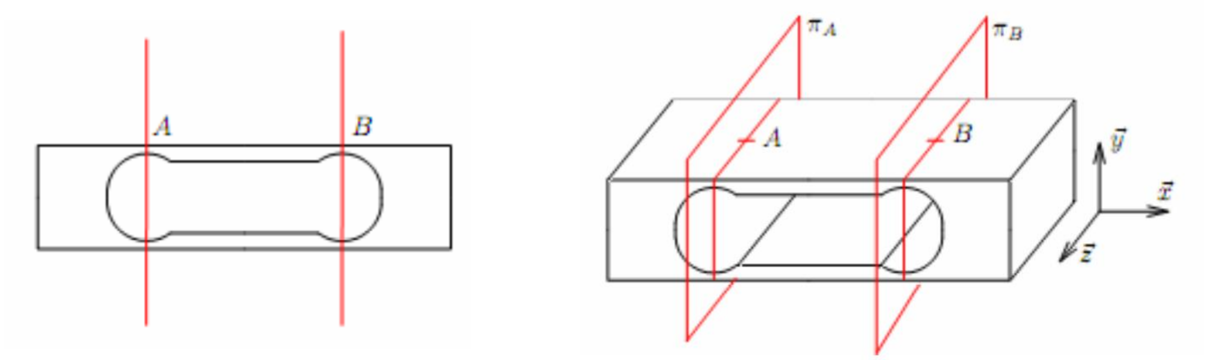


Fig. 2: a) Points A & B in 2D Model **b) Points A & B in 3D Model**

3. MODELLING

The modelling has been done for 2D and 3D analysis and a mesh has been generated to give an optimum result. To get optimum results mesh convergence has to been studied hence mesh convergence is performed and found that at 1350 elements the percentage is less than 5% and least among all others hence this mesh is approved for further study of analysis. The graph shown in Fig. 3 depicts the strain value in x-direction along with the element size. The mesh images have been shown in Fig. 4.

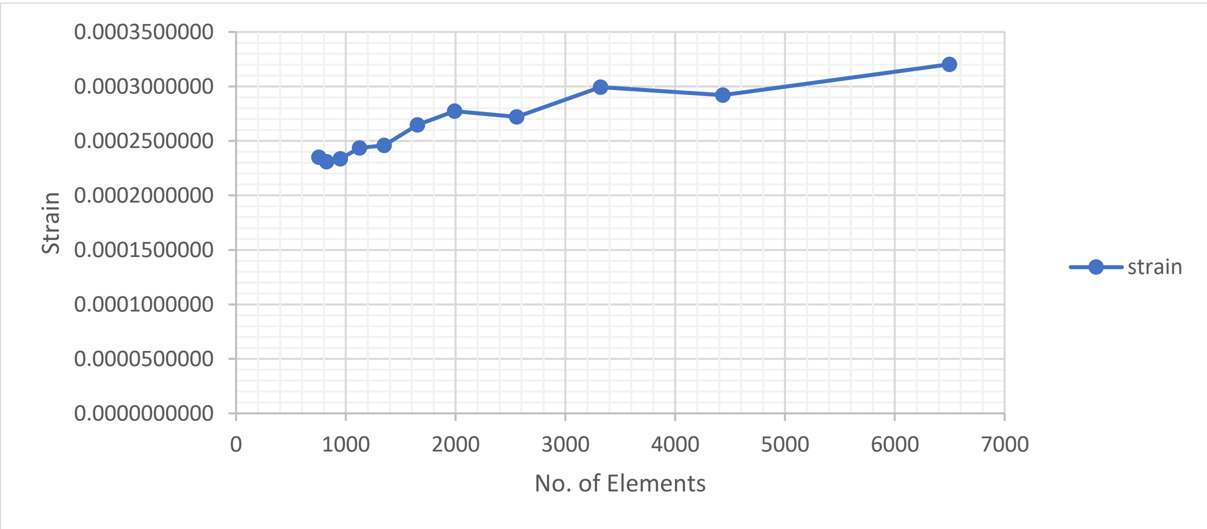


Fig. 3: Graph depicting Mesh Convergence Study

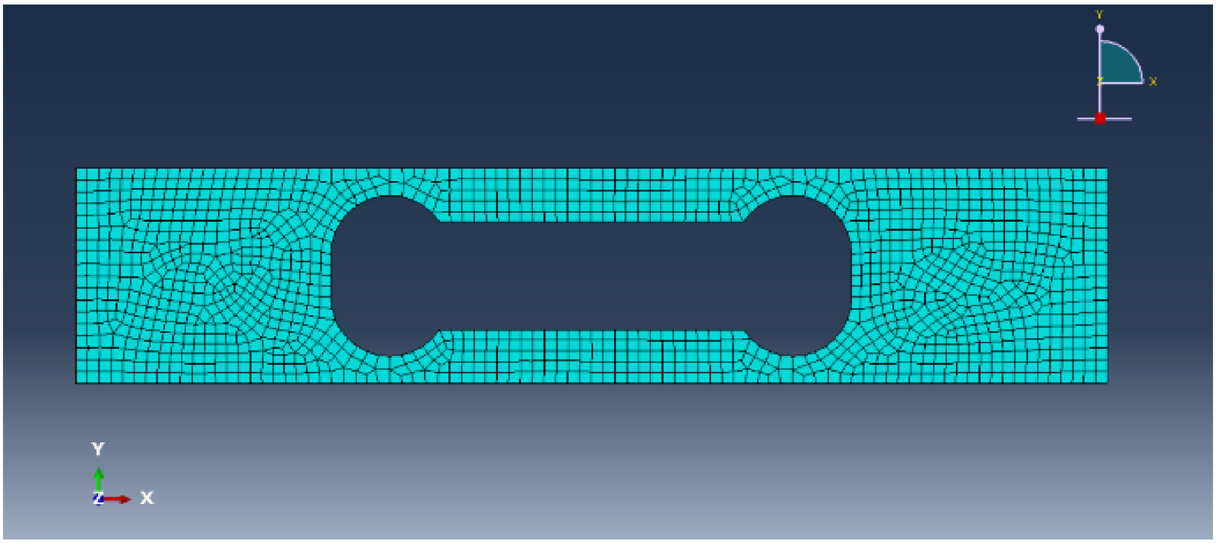
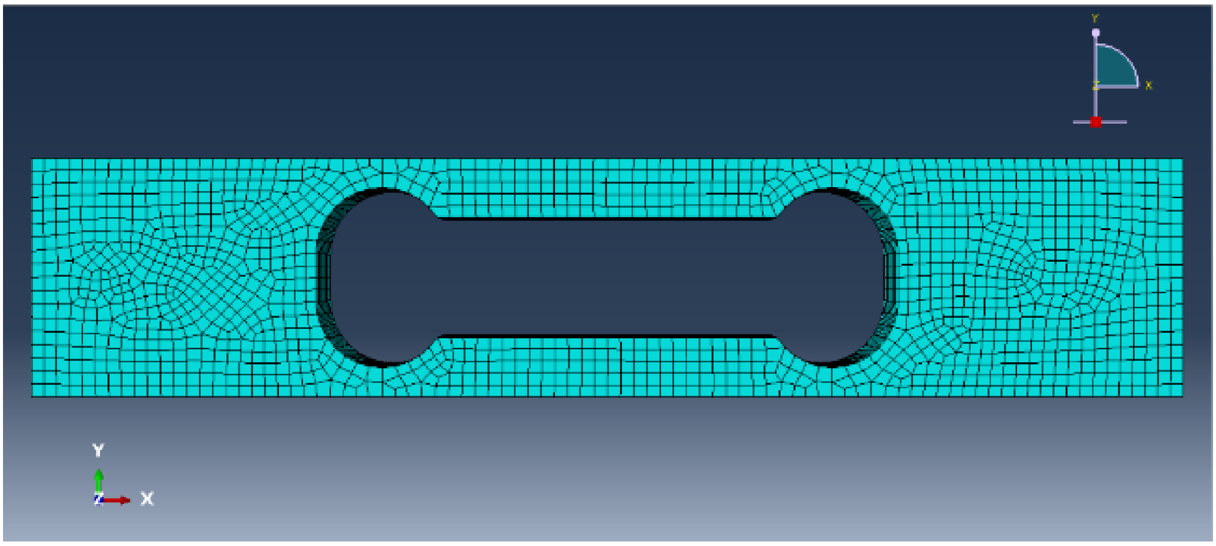


Fig. 4: a) Mesh for the 2D model of load cell



b) Mesh for 3D model of load cell

4. <u>ANALYSIS</u>

The analysis for the 2D and 3D model of load cell is performed. In analysing we consider only the load cell; hence we apply forces directly to the load cell. Considering load having applied with concentrated loads on edges of top edge of free end as -288N and bottom edge of free end as 288N and a shear force or surface traction force of 120N through the surface on the free end.

Figure 5 shows the Boundary Conditions and loads acting on the load cell. Partition also has been created to get the exact point where the stress, deformation and strain needs to be measured.

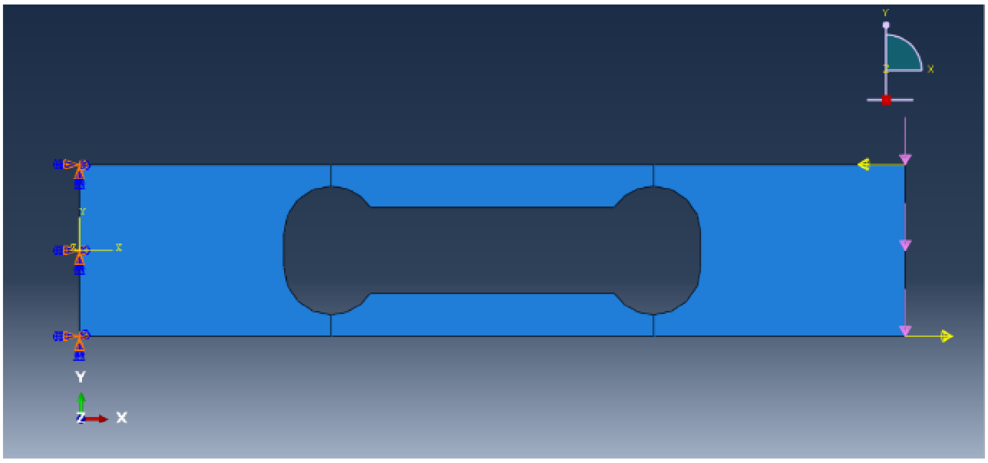


Fig. 5: a) Boundary Conditions for 2D Model

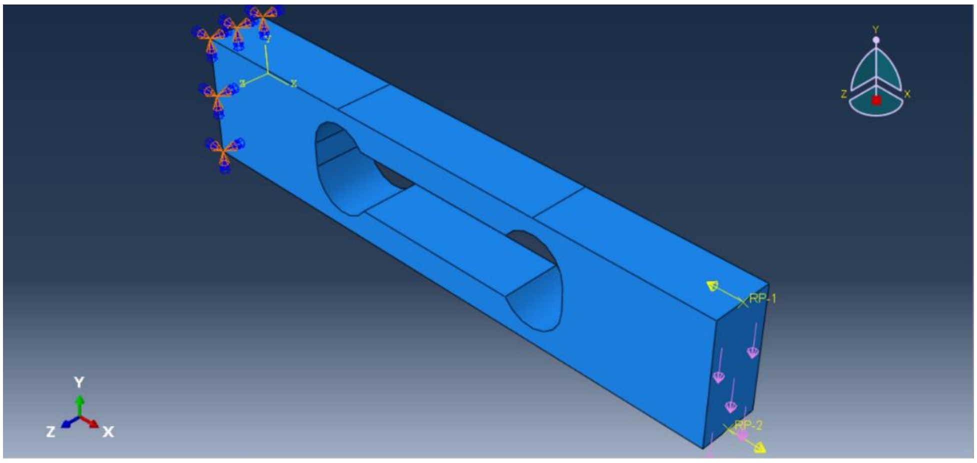


Fig.5: b) Boundary Conditions for 3D Model

5. RESULTS:

5.1. 2D AND 3D MODEL COMPARISON

The results have been obtained for 2D and 3D analysis of the model. Hence the results are very close which can be observed in the Figures 6, Figure 7 and Figure 8 showing the Von Mises Stress, Strain and Displacement respectively. Hence, we use 2D analysis since it reduces computation time, storage and give results very close to reality.

CASE STUDY-1

The analysis is done based on vertical displacement, Von Mises stress and strain. These values give us information particularly at points A and B by keeping the strain gauges at those nodes and hence giving us the info about the weight of an object. Due to this we compare results for 2D and 3D models which tells us the deflection are very similar in spite of thickness being very less.

The strain gauges were kept at points A and B as in Figure 2 so that the deformation obtained at these places gives the accurate output of the weight of an object.

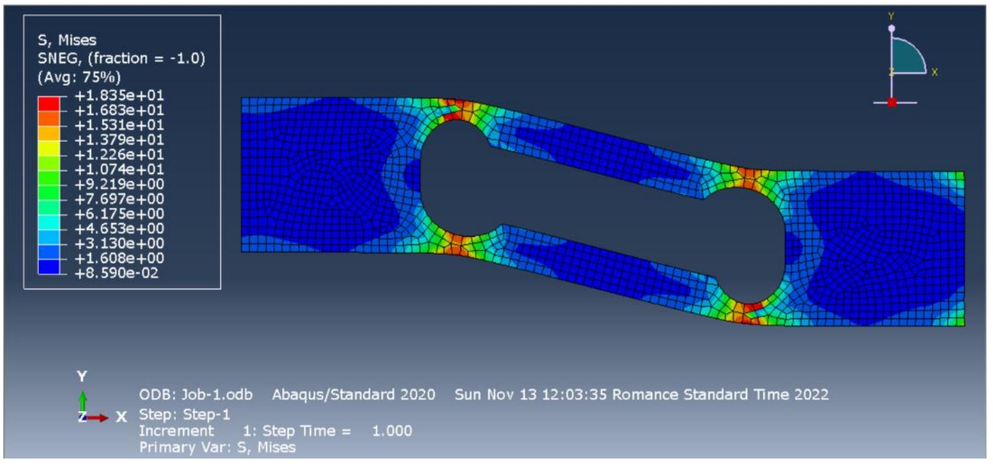


Fig. 6: a) Von Mises Stress for 2D Model

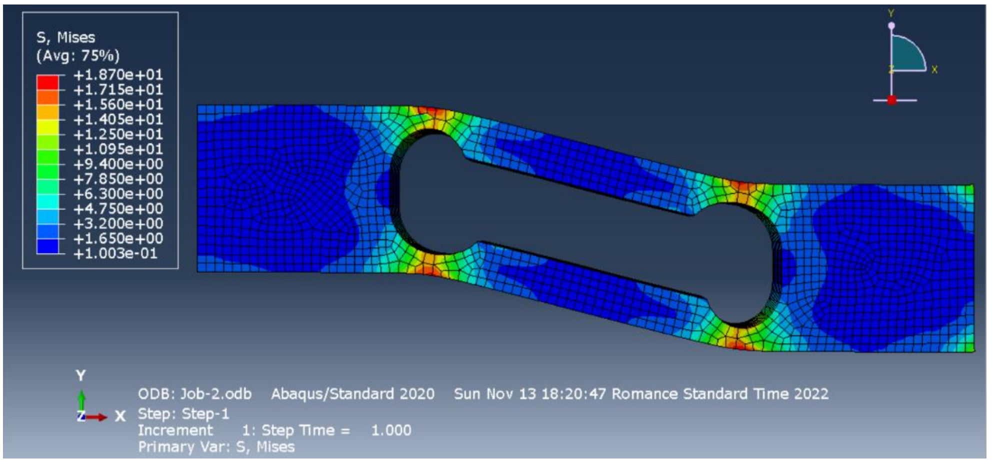


Fig. 6: b) Von Mises Stress for 3D Model

CASE STUDY-1

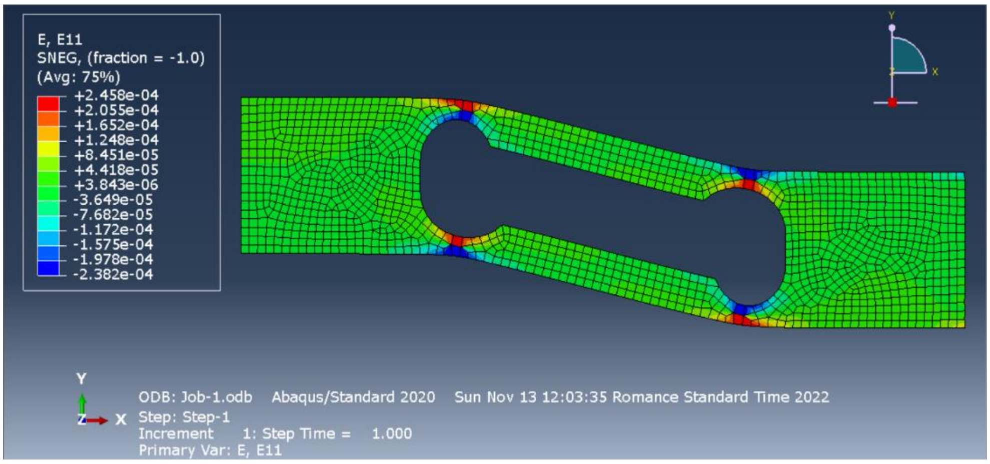


Fig. 7: a) Strain E11 for 2D Model

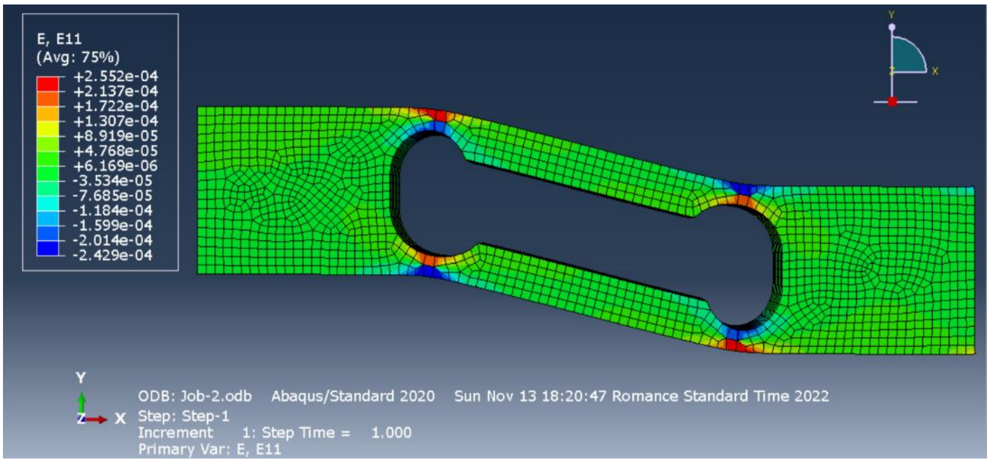


Fig. 7: b) Strain E11 for 3D Model

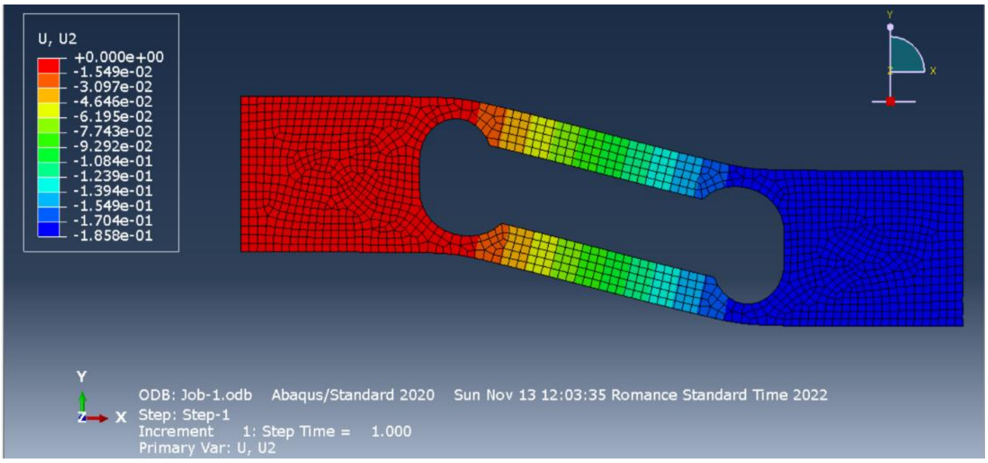


Fig. 8: a) Deflection in U2 for 2D Model

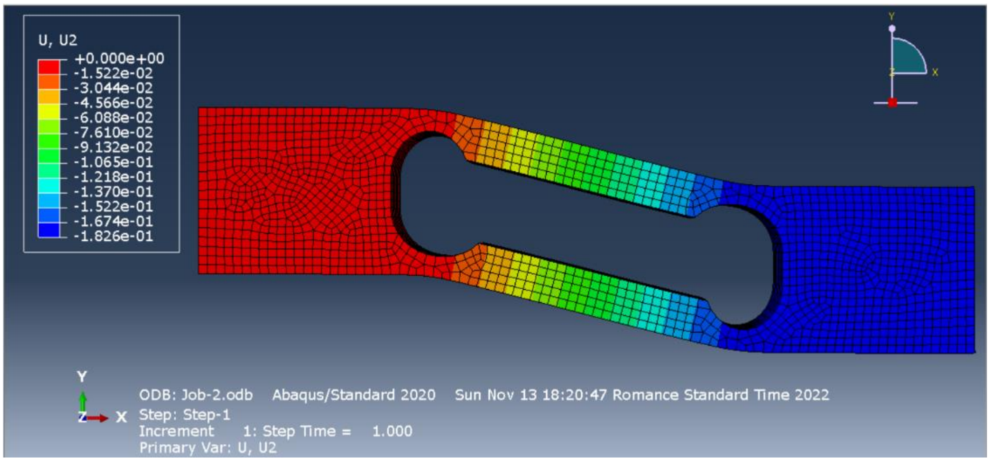


Fig. 8: b) Deflection in U2 for 3D Model

5.2. CONTOURS WITH RESPECT TO TIME

The contours have been plotted for Von Mises stress, strain and deformation with respect to time respectively as shown in Figure 8, Figure 9 and Figure 10. As time interval increases stress, strain and displacement keeps on increasing. There is a deviation at point A and B in stress and strain response in 3D model due to meshing done on it due to student version of the analysis software. During analysis the strain measured at points A and B are found to be 0.0002453 and -0.0002382 respectively at the mesh convergence point.

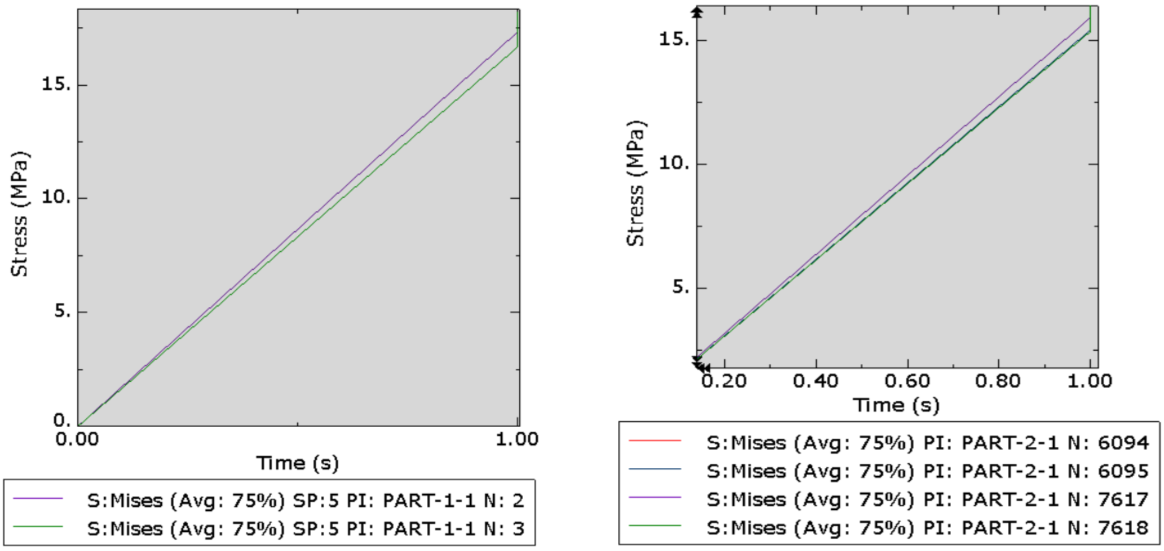


Fig. 9: Stress Response at strain gauge points for a) 2D Model and b) 3D Model

CASE STUDY-1

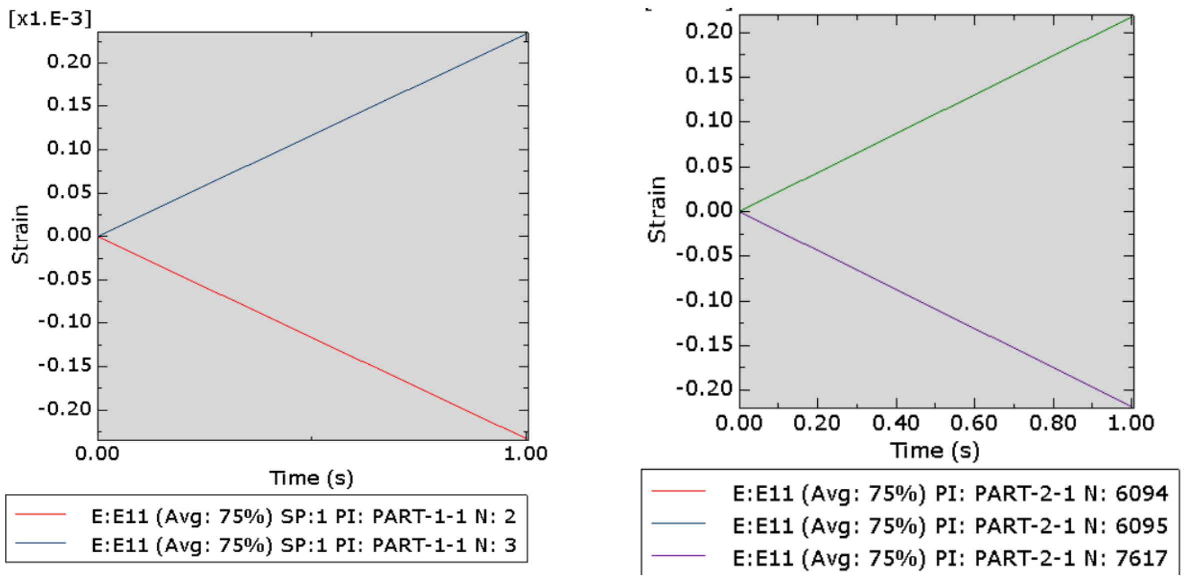


Fig. 10: Strain Response at strain gauge points for a) 2D Model and b) 3D Model

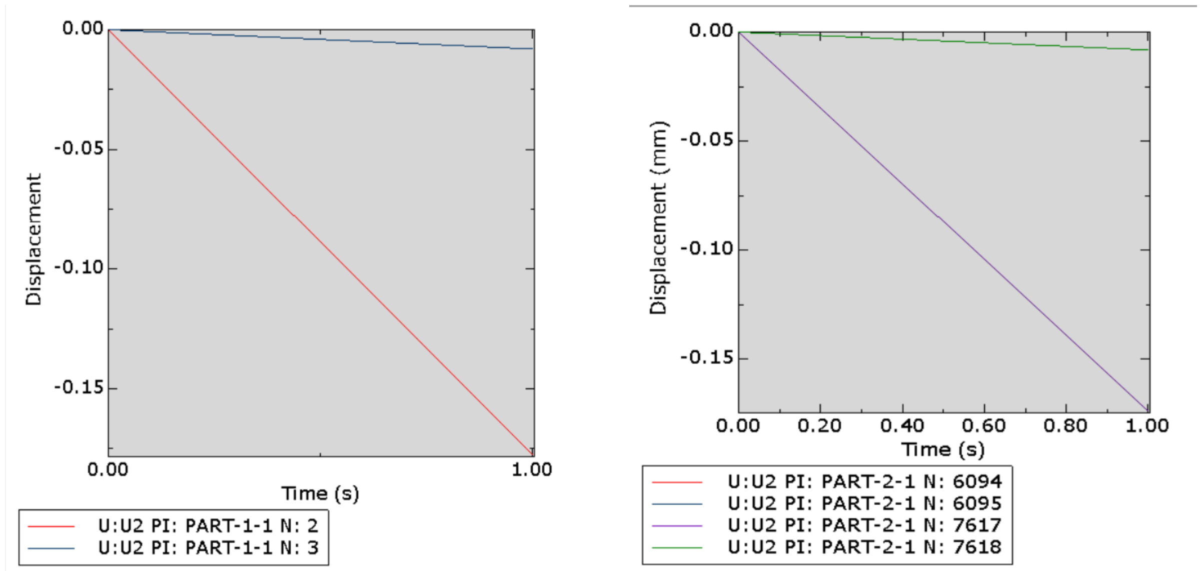


Fig. 11: Displacement Response at strain gauge points for a) 2D Model and b) 3D Model

5.3. <u>APPLYING LOADS OF DIFFERENT MAGNITUDE</u>

After applying different loads on the loadcell we get their respective values for strain. We apply a load of 150N, 180N, 210N and check for the strain values, the values of moment forces change to 360N, 432N, 504N and the shell shear edge load changes to 3.75N/mm, 4.5N/mm, 5.25N/mm respectively.

CASE STUDY-1

Fig. 12, Fig. 13 and Fig. 14 shows the strain contour for the load 150N, 180N, 210N respectively and deformation at those points

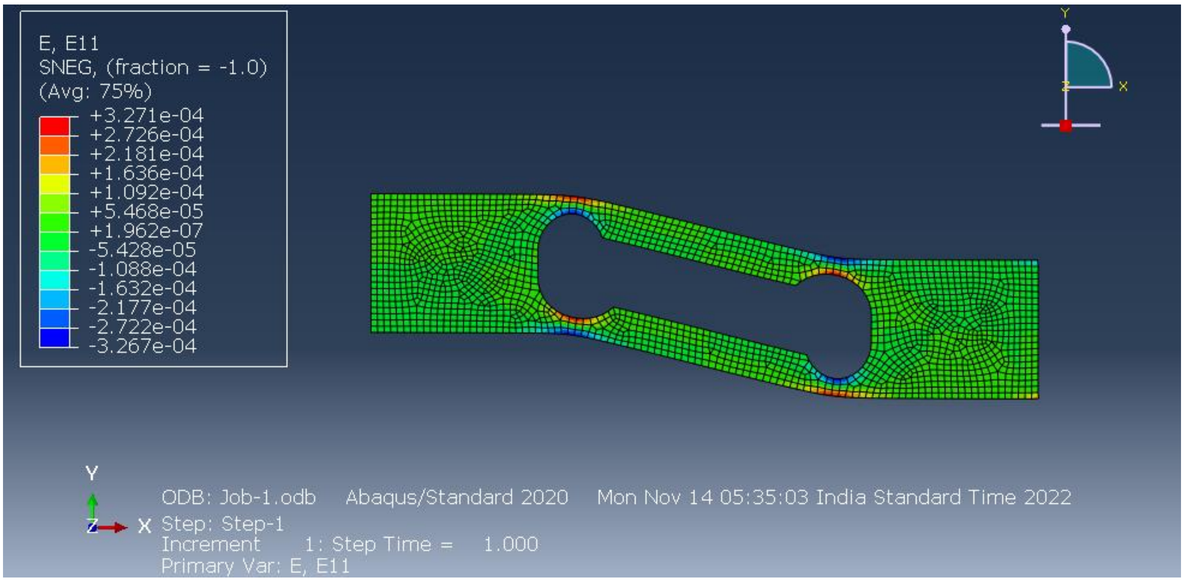


Fig. 12: Strain value for 150N load

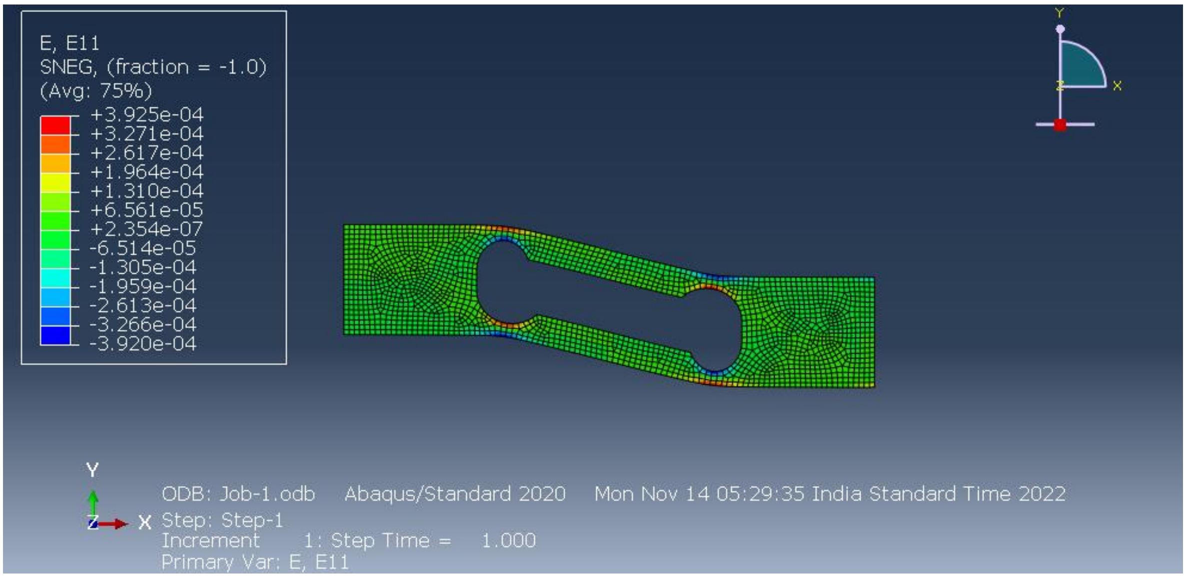


Fig. 13: Strain value for 180N load

CASE STUDY-1

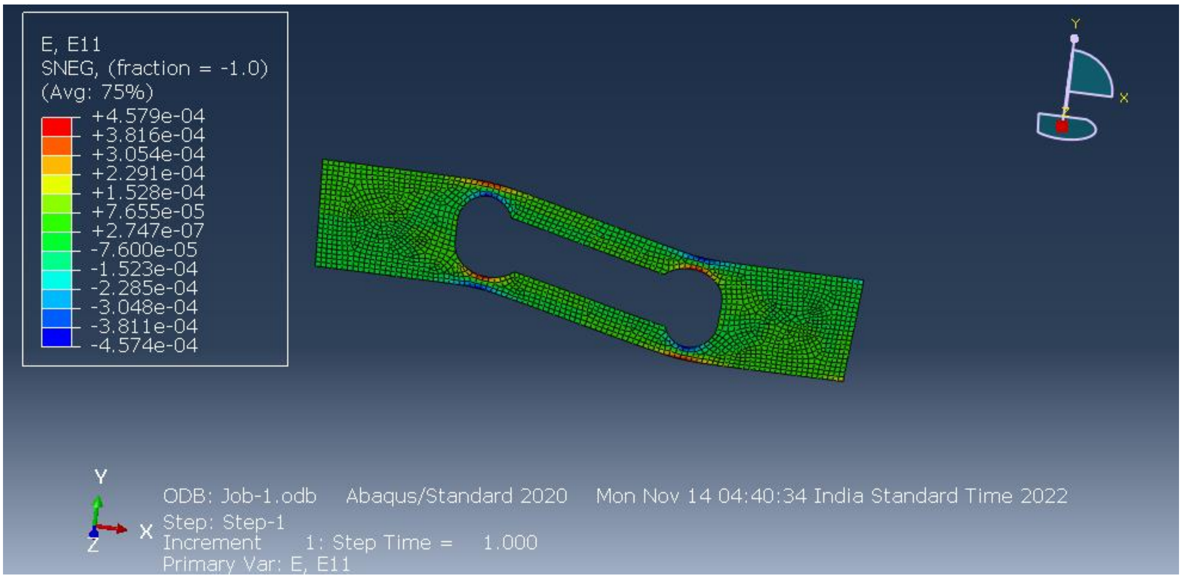


Fig. 14: Strain value for 210N load

The variation of different magnitude of loads with respect to strain has been plotted in the Fig. 15.

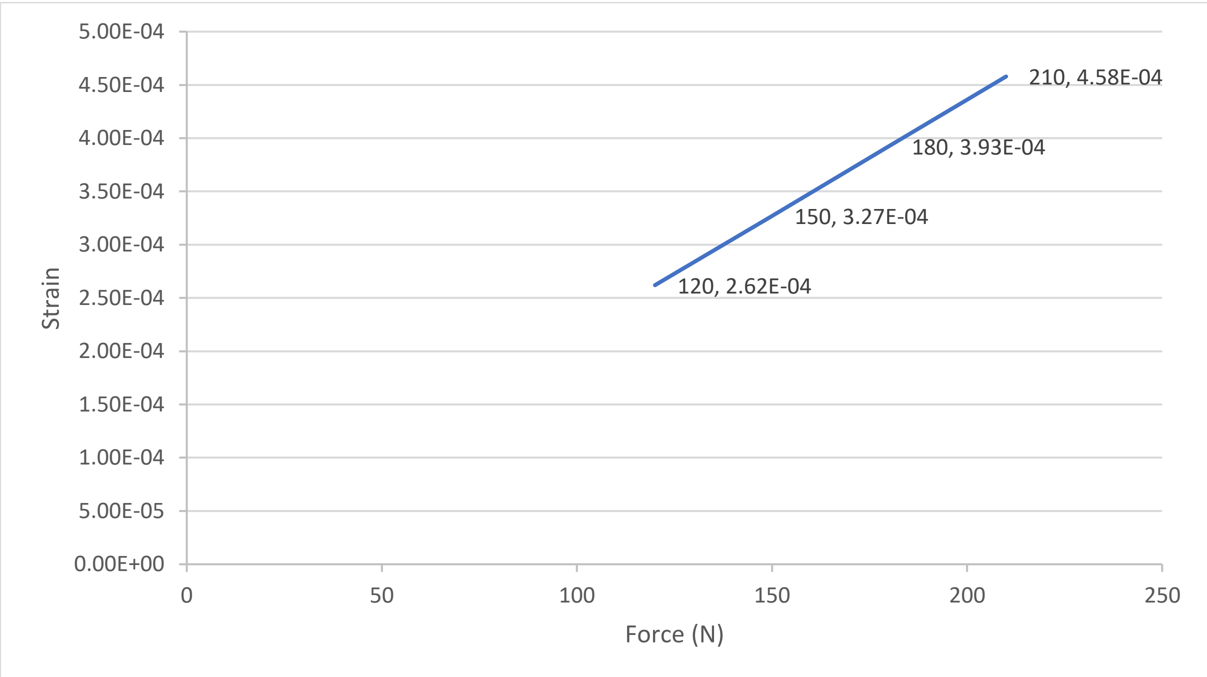


Fig. 15: Graphical representation of Load versus Strain

As the graphical representation shows that the load is linearly related to the strain, we can get a linear equation representing this line.

$$Y = (2.16\,X + 1.6) * 10^{-6}$$

CASE STUDY-1

It is the form of Y=mX+c

Where Y = y-axis

X = x-axis

m = slope

c = constant intersection at y-axis when x=0

5.4. DIFFERENT LOCATION OF THE LOAD APPLIED

When load is applied at middle of the model i.e., at 96mm the Strain values obtained are given from above figures, i.e., at Node A 0.0002369 and at Node B -0.000286, when the location of load is changed to left end of the model at 192mm, the moment values change from 288N to 576N, and the shear force remains the same that is shell shear edge load of 3N. This is represented in Fig. 16.

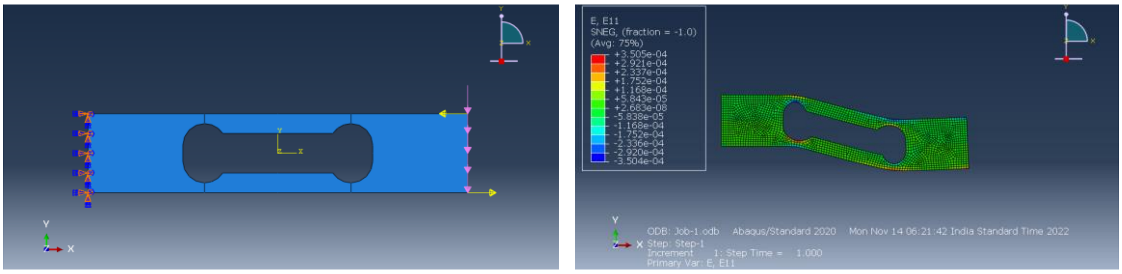


Fig. 16: Load applied at the top left corner of the loadcell and respective strain results

The Values of E11 at Node A is 0.0002844 and Node B is -0.0002388 And when load is applied at right end of model, i.e., no moment forces act and only a shell shear edge load is applied on the right end of the model, as shown in the below Fig. 17.

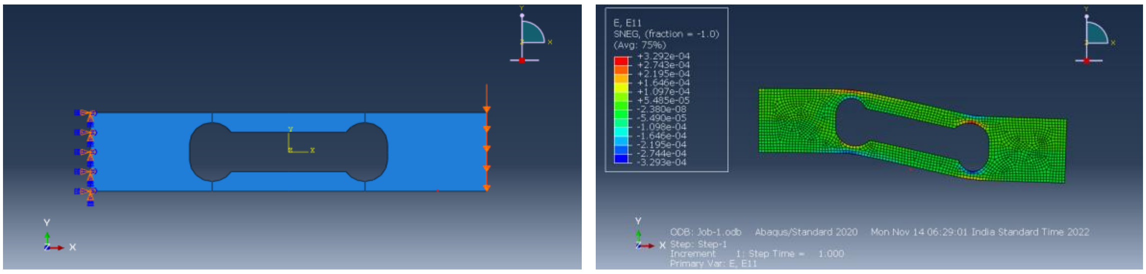


Fig. 17: Load applied at the right end corner of the loadcell and respective strain results

For the same amount of load applied at different locations, the values of strain measured is changing, which shows that the accuracy of the model to measure weight is relatively low. i.e., the weight measure accuracy is low for this model.

Conclusion of deformation mode, there is no permanent or plastic deformation, the model is not changing its shape permanently, here the body undergoes elastic deformation or also known as temporary deformation, to measures the weight by using the information from strain gauges. The shape of the model is very practical and easy to manufacture, the shape is made in such a way that strain gauges can be placed at location where the stress concentration factor will be high. The four strain gauges on the cell will measure the bending distortion, two measuring compression and two tension. When these four strain gauges are set up in a Wheatstone bridge formation, it is easy to accurately measure the small changes in resistance from the strain gauges. The Strain gauges at Node A and B are very efficient because the stress and strain concentration is very high at point A and point B, which results in accurate measure of strain at those respective point regardless of magnitude of load applied and location of load applied.

6. CONCLUSION

The linear static analysis of a load cell, the mechanical component of an electronic balance was carried out to determine the strains in two regions A and B of its top surface. 2D and 3D model were considered for analysis. The analysis was carried out for mechanical behaviour of load cell and results have been obtained very similar to real life scenarios with little deviations possible. Since the load is applied in different ways in real life hence it might contain some deviations.

CASE STUDY 2

Simulation of a Belleville Spring Ring

1. INTRODUCTION

The Belleville spring ring is a type of washer having an inclination and some height to provide higher stiffness compared to other washers available. These washers do not possess only elastic behaviour they operate in elastoplastic behaviour unlike the springs. When a compressive force is added the spring tends to deform in y-direction as well as in x-direction. This behaviour of the washer helps to provide a greater grip and stability to the nuts and bolts.

In the present case study, the Belleville spring is considered to be made of steel in elastic and elastoplastic behaviour which is subjected to static structural analysis. The modelling choices are to set up in Axisymmetric Shell element. The simulation is carried out under steady conditions, post which the results are presented and analysed. The objective of the simulation is operating the structure within the elastic limits.

2. DESIGN

The image of Belleville spring with its dimensions is depicted in Fig. 1. The sketch used for Belleville spring has been shown in Fig. 2. The part has been created in Axisymmetric shell element configuration in Abaqus 2020. The sketch has been designed in mm.

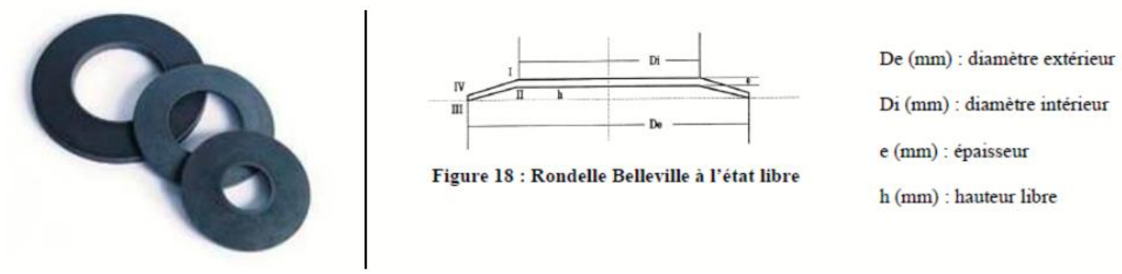


Fig. 1: Belleville Spring

Where, Outer Diameter De=25mm

Inner Diameter Di=12.2mm

Height h=1mm

Thickness of the washer e=0.5mm,

CASE STUDY 2

The spring has been studied in Asymmetric shell system as for analysis the change in quadrant depicts the entire system characteristics which in-turn reduces computation time and provides same results as the entire body.

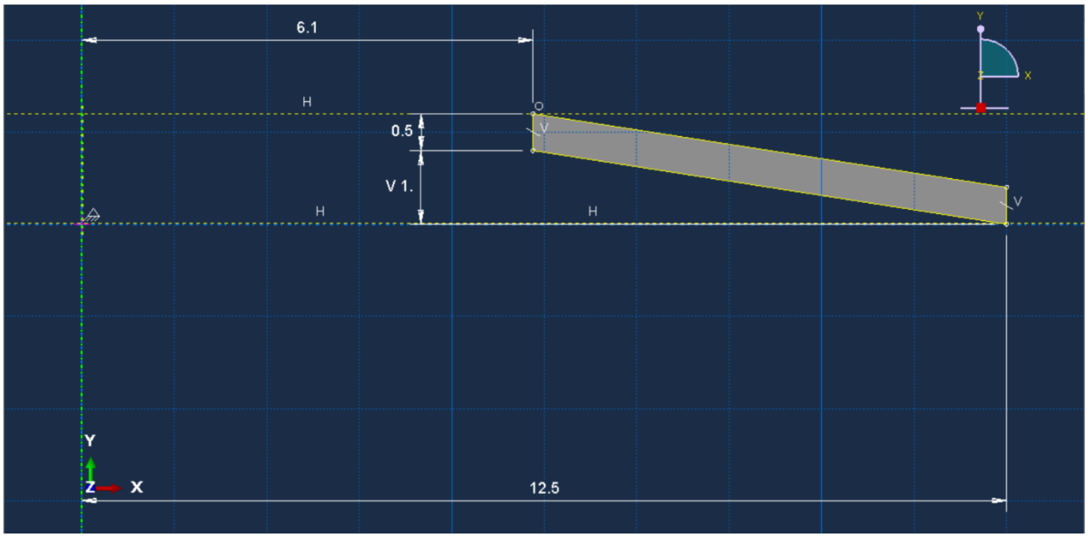


Fig. 2: Sketch of Belleville spring in mm

The values for elastic and plastic properties are mentioned in table 1.

Table 1: Elastic & Plastic Behaviour

- **ELASTIC BEHAVIOUR**

YOUNG'S MODULUS	210 GPa
POISON'S RATIO	0.3

- **PLASTIC BEHAVIOUR**

YOUNG'S MODULUS	**STRAIN**
400	0
477.495949	0.002
495.408705	0.004
507.749529	0.006
517.461894	0.008
525.594322	0.01
532.65523	0.012
538.93396	0.014
544.612555	0.016
549.813778	0.018
554.624747	0.02
559.109756	0.022
563.317746	0.024
567.286929	0.026

CASE STUDY 2

During analysis many assumptions are made before the analysis. Some of them are:

- Define two horizontal rigid planes (that can be wires for a 2D geometric space). Apply a rigid body constrain to these two planes.
- The bottom part is encastered.
- The top part has a unique mobility towards the axis of the Belleville ring. A displacement u=1mm is applied to compress the ring (full compression up to a flat shape).
- A frictionless contact is imposed between the planes and the ring.

3. MODELLING

The modelling has been done and the materials properties has been inserted with elastic and elasto-plastic material analysis and a mesh has been generated to give an optimum result. To get optimum results mesh convergence has to been studied hence mesh convergence is performed and found that at 4158 elements at 0.028 mesh size, the percentage is less than 5% and least among all others hence this mesh is approved for further study of analysis. The Plot shown in Fig. 3 depicts the strain value in x-direction along with the element size. The mesh images have been shown in Fig. 4.

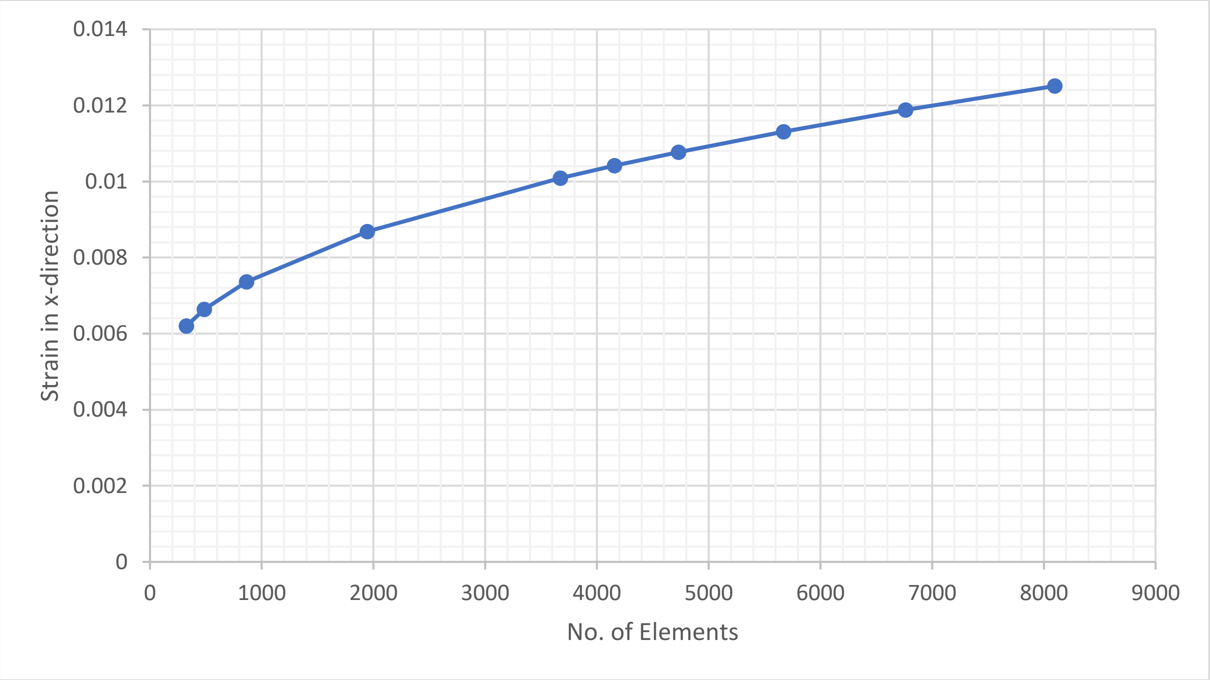


Fig. 3: Plot depicting Mesh Convergence Study with respect to E11

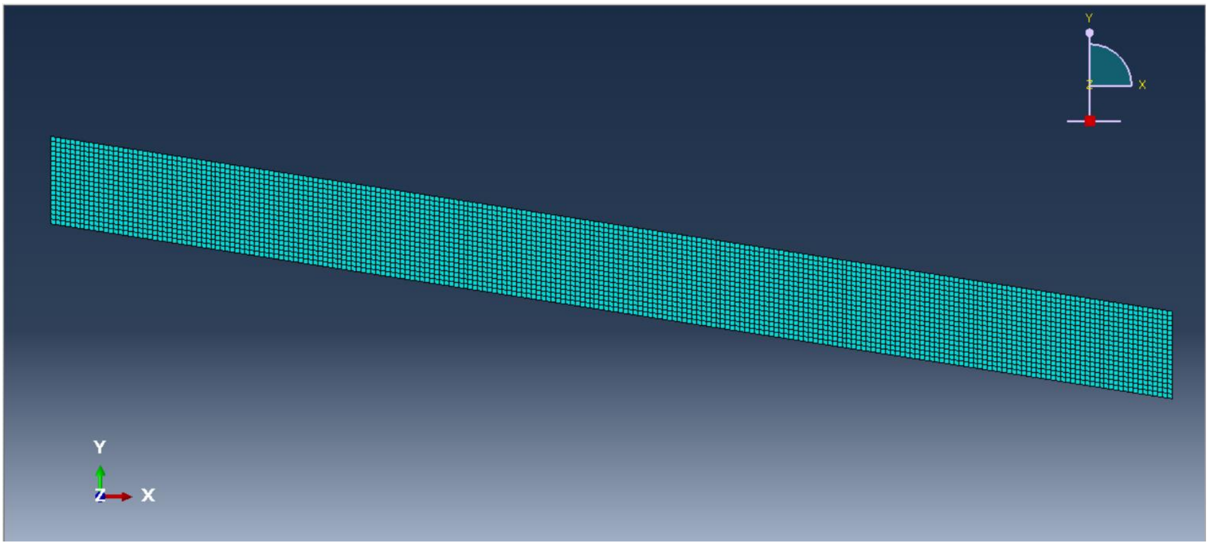


Fig. 4: Mesh for the washer

4. ANALYSIS

The analysis for the Belleville spring is to be done with the help of boundary condition. The spring is attached to 2 rigid surfaces which analytical rigid wire bodies by definition. The bottom rigid surface has been encastered for restricting it further movement than that surface. The upper surface has been provided with deformable and movement with a force of 2.5N in y-direction towards the spring to flatten the spring to its maximum extent (in this case 1mm). To apply the forces reference points has been assigned to each rigid surface where the boundary conditions have to be applied. Fig. 5 shows the boundary conditions for Belleville spring.

Fig. 5: Boundary Conditions for Belleville spring

5. RESULTS:

5.1 ANALYTICAL RESULTS

Using analytical formula, we find the force values at different strain values with time steps of 0.1 using the equation 1. This equation is formulated to calculate force for Belleville springs.

$$F = 4\frac{E}{(1-\mu)\alpha De} \cdot s.e. \left[(h-s)\left(h-\frac{s}{2}\right)+e\right] \text{, with } \alpha = \frac{1}{\pi}\frac{\left(\frac{\delta-1}{\delta}\right)}{\frac{\delta+1}{\delta-1}-\frac{2}{\ln(\delta)}} \quad \ldots(1)$$

Where, F is Force in N

E is Young's Modulus in MPa

μ is Poisson's ratio

De is the outer diameter in mm

s is displacement in mm

h is the depth of the spring in mm

e is the thickness of the spring in mm

δ is the ratio of outer to inner diameter of the spring

Hence, we plot the variation of force with displacement and observe there is a linear relationship between them. The Fig. 6 shows the Plot of reaction force with respect to displacement analytically.

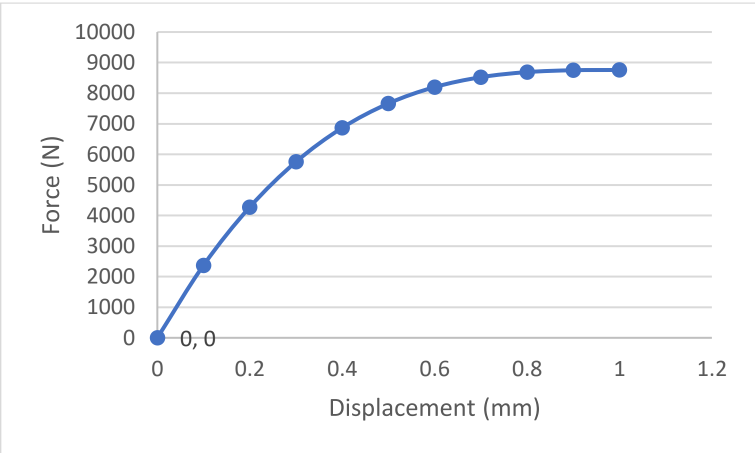


Fig. 6: Plot depicting the Force versus Displacement relation analytically

CASE STUDY 2

5.2 FEM RESULTS

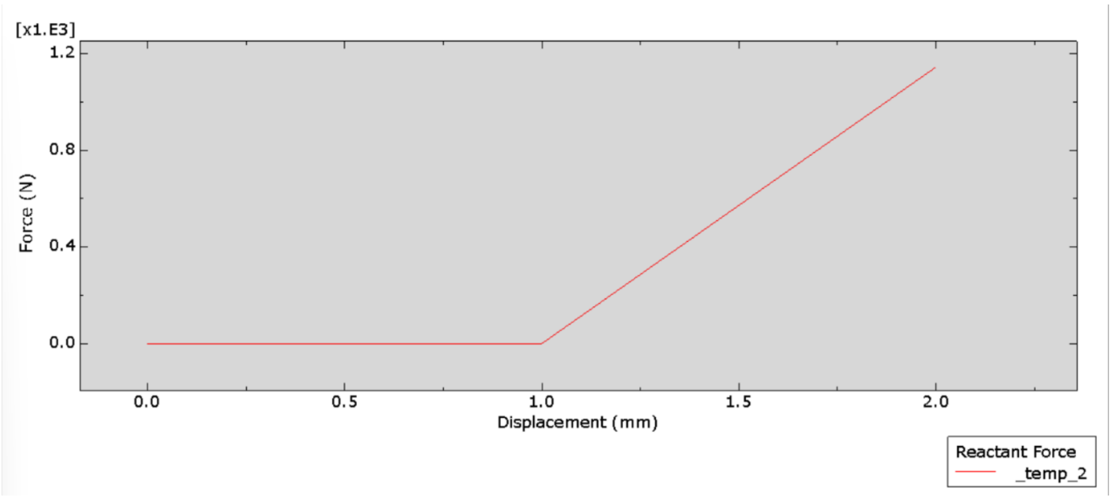


Fig. 7: Plot depicting the Force versus Displacement relation by FEM

The results of Reactant force with respect to displacement in mm have been obtained. Analysing the results, the software observe deformation till 1 mm where reactant force does not have a value. After 1 mm till maximum deformation the reactant force has a linear behaviour as in the analytical solution. The analytical solution does show the similar force at the same deformation but still poses a linear relation. The reason for non-matching of force values is because of the assumptions made in analytical solutions where it does take care of materials at each node and regular intervals which gives precise values than analytical solution.

The results are very close which can be observed in the Figures 8, Figure 9 and Figure 10 showing the Von Mises Stress, Strain E11 and Displacement respectively.

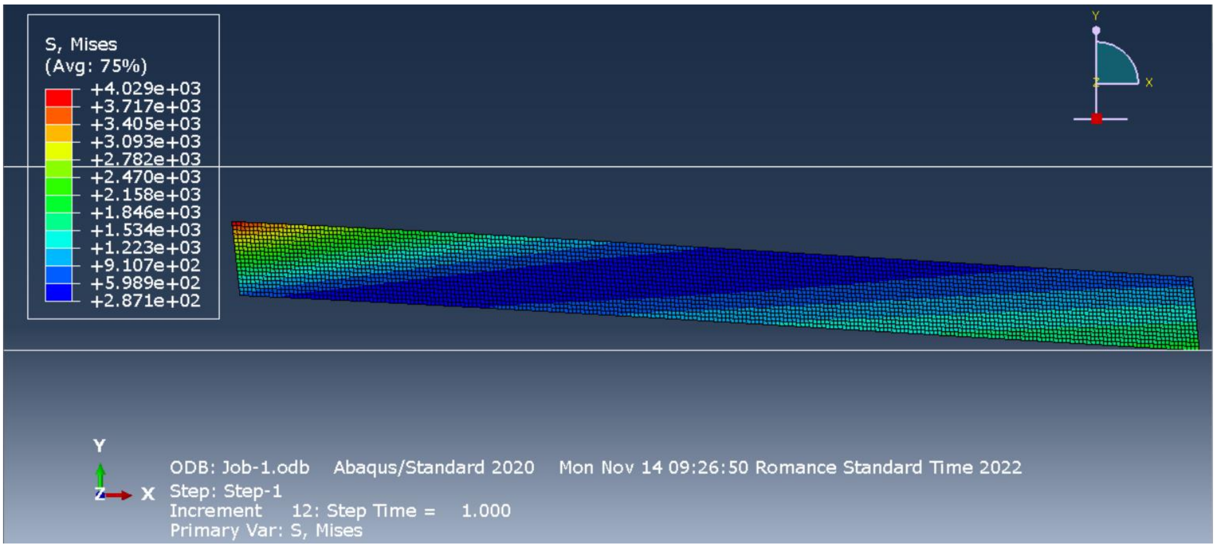


Fig. 8: Von Mises Stress for Belleville Spring

CASE STUDY 2

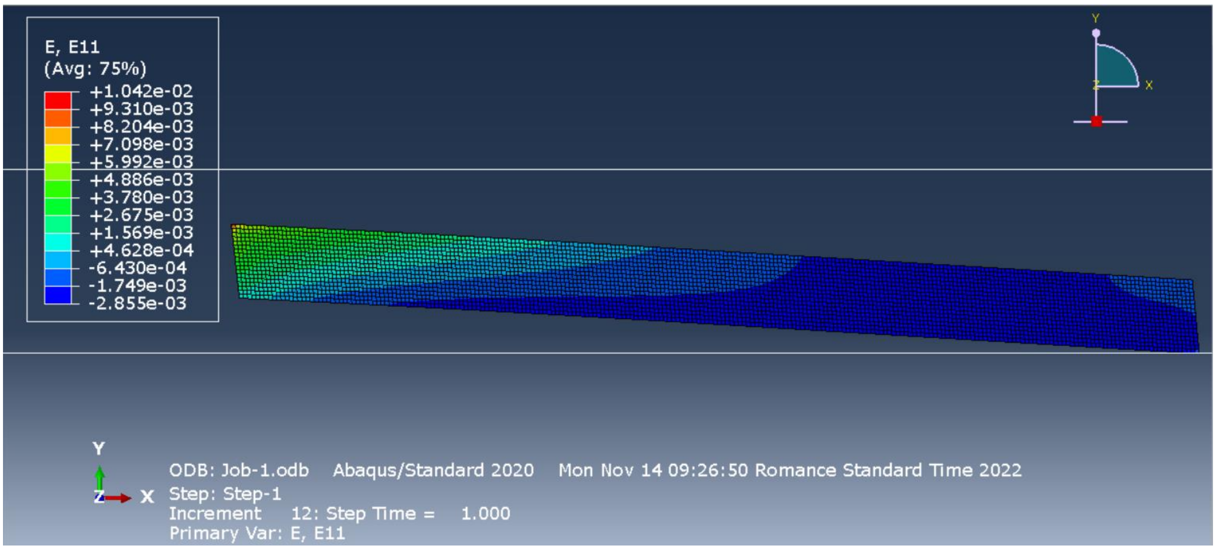


Fig. 9: Strain E11 for Belleville Spring

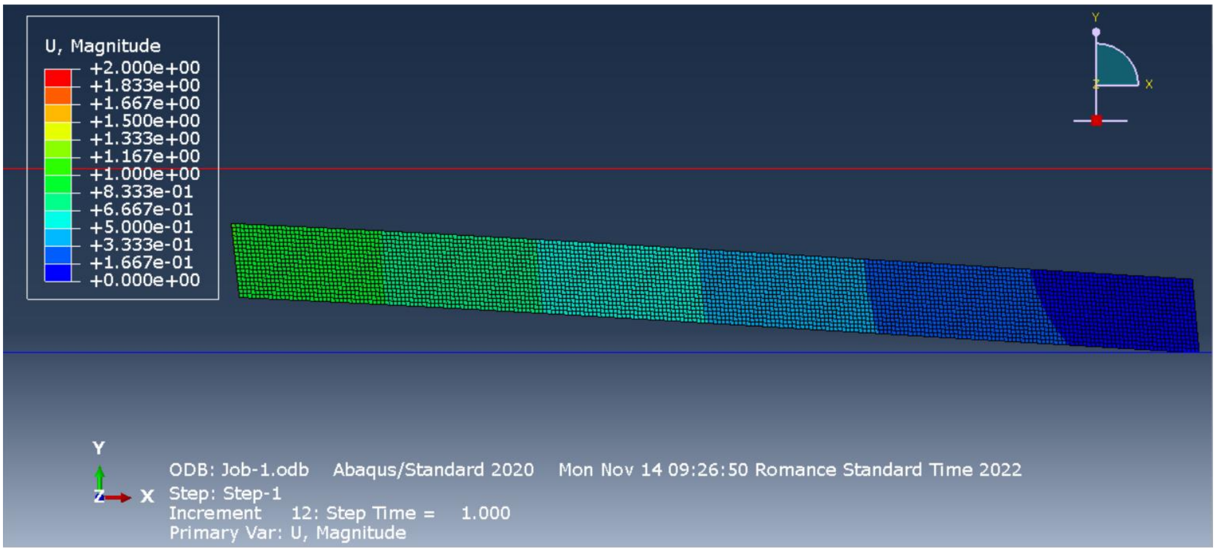


Fig. 10: Deflection U for Belleville Spring

5.3 DEFORMATION AT 400MPA OF YIELD STRESS

The objective for the force to reach the yield stress 400MPa move step wise to get the force to be near to 400MPa and we find out that at time step t=0.5476s the force reaches 383.6N and the displacement to be 1.095mm. The Fig. 11 shows the deflection at the time step of 0.5476 seconds.

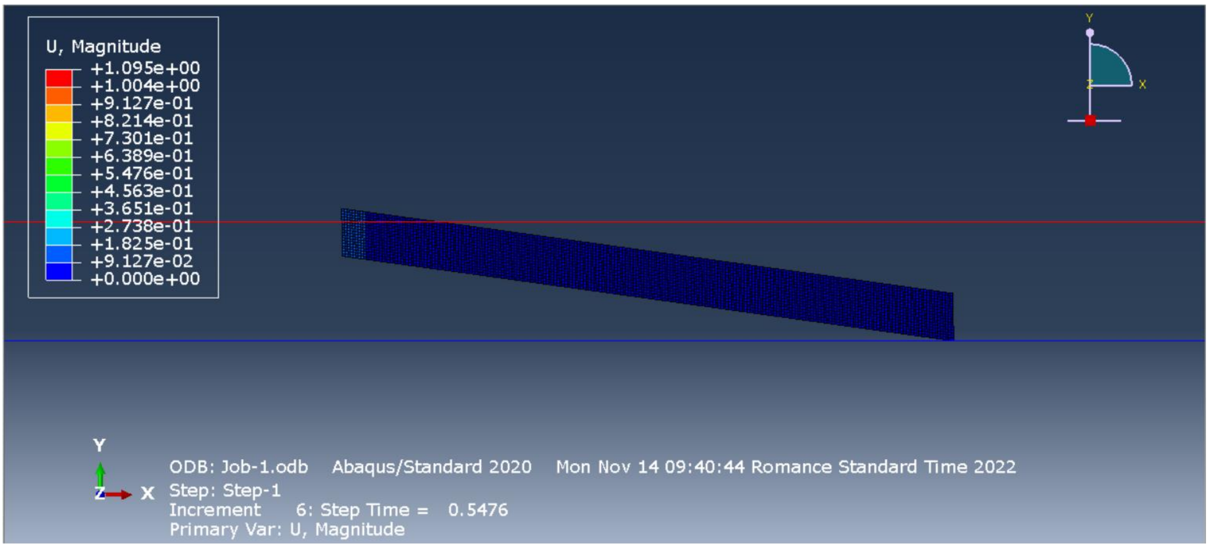


Fig. 11 Deflection U of Belleville Spring at time step 0.5476 seconds

5.4 ELASTO-PLASTIC BEHAVIOUR

The Belleville spring is subjected to elasto-plastic behaviour based on the values mentioned in the introduction. The Reactant force versus the displacement Plot is plotted to show the results in elasto-plastic behaviour in Fig. 12.

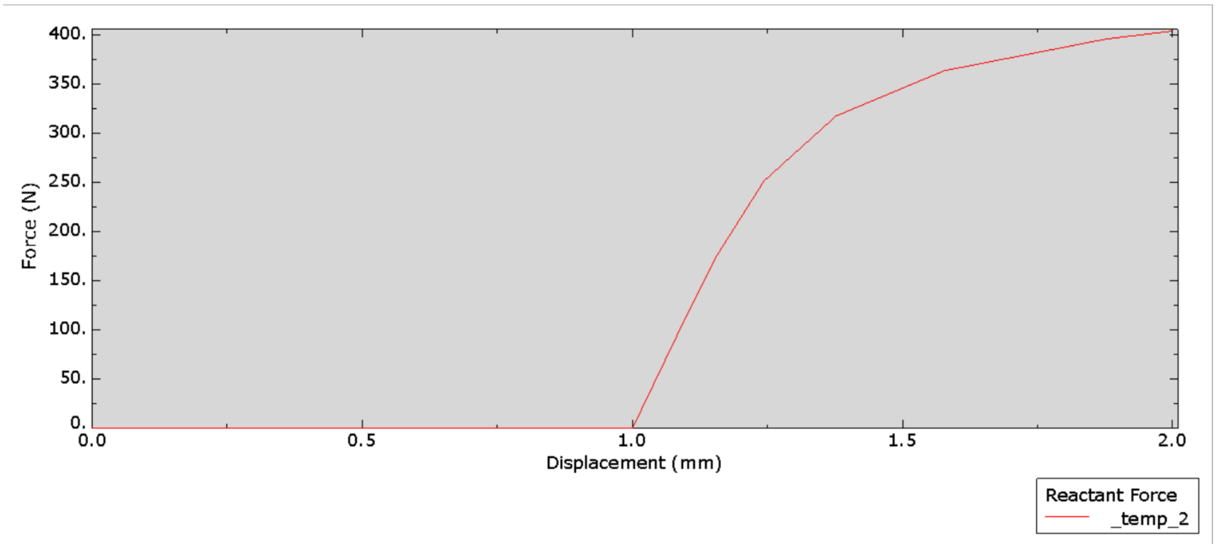


Fig. 12: Plot depicting the Force versus Displacement relation for Elasto-Plastic Behaviour

5.5 UNLOADING THE SPRING

The spring loading cases have been studied and analysed but due to its elasto plastic behaviour the spring needs to be analysed in unloading process. Hence a Plot is plotted of force versus displacement and analysing we find out that there is some energy loss of the material and the spring has been gone to plastic zone since it does return to the same zero displacement. This happens if the spring contain elasto-plastic behaviour. Hence, we need to change the washer after several uses. Fig. 13 depicts the unloading of spring.

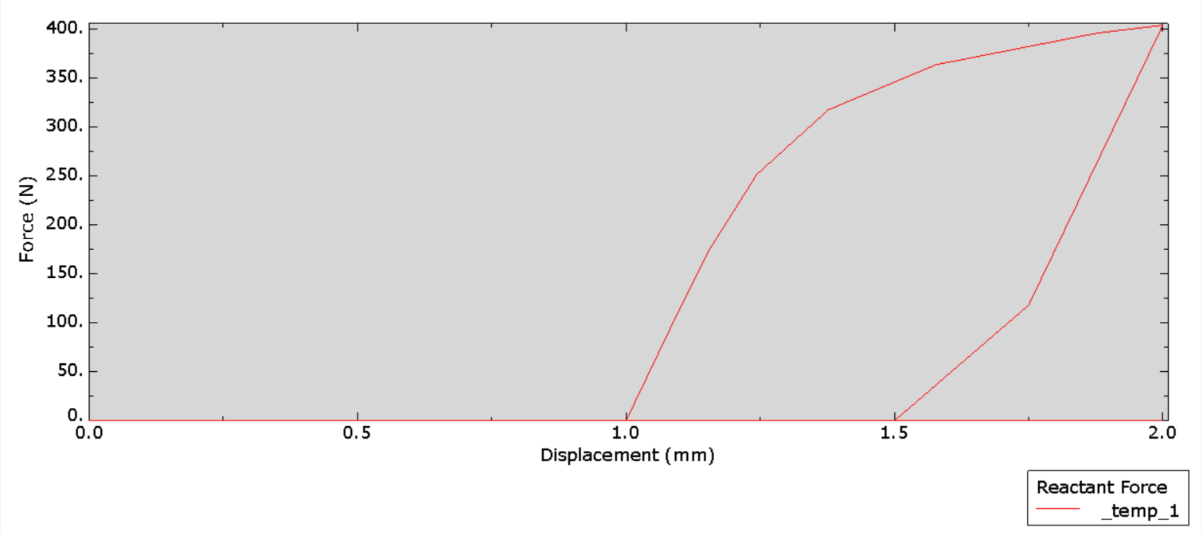


Fig. 13: Plot of force versus displacement with unloading for elasto-plastic behaviour

6. CONCLUSION

This simulation shows that the usage of the elastoplastic behaviour for the spring is much more realistic than just the elastic behaviour. The elastic behaviour can be used to simulate the behaviour of a material in the elastic region which is below the yield strength; however, in the case where the yield strength is exceeded, elastoplastic behaviour should be used to simulate true material behaviour. In addition, it is shown that in the case of a spring, one should not exceed the yield strength since elasticity is essential for a spring application, and exceeding this would make our system in the plastic region and thus not favourable.

CASE STUDY 3

Simulation of a Composite Plate

1. INTRODUCTION

In today's world use of single material does not fulfil the criteria for various purposes. Composite material when combined provides better mechanical properties than individual. They are a combination of a matrix and fibre. Some of the natural material possessing composite properties are wood and bone.

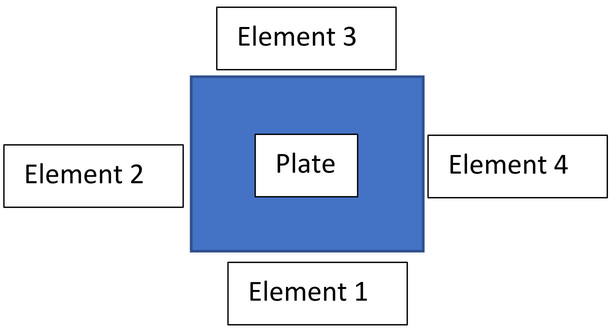
This assembly phenomenon, which makes it possible to improve the quality of the material for a certain use (lightness, mechanical rigidity, etc.) explains the growing use of composite materials in many industrial sectors. During the 20th century, the development of computers allowed the precise calculation of mechanical properties and the field of use of these artificial materials exploded. Nevertheless, the fine description of composites remains complex from a mechanical point of view due to the non-homogeneity of the material.

A composite material is composed as follows: matrix + reinforcement + optionally: filler and/or additive. Examples: reinforced concrete = concrete composite + steel reinforcement, or fiberglass composite + polyester resin.

The composite material is subjected to various frequencies in various events. Hence, they are subjected to modal analysis to determine the deformation at various natural frequencies.

2. DESIGN

The image of composite plate with its dimensions is depicted in Fig. 1. The sketch used for composite plate has been shown in Fig. 2. The entire structure is focussed on 16 layers of composite. The part has been created in 3D deformable shell element configuration in Abaqus 2020. The sketch has been designed in mm. The demonstrator panel is 100x100mm with a 6mm thickness.



CASE STUDY 3

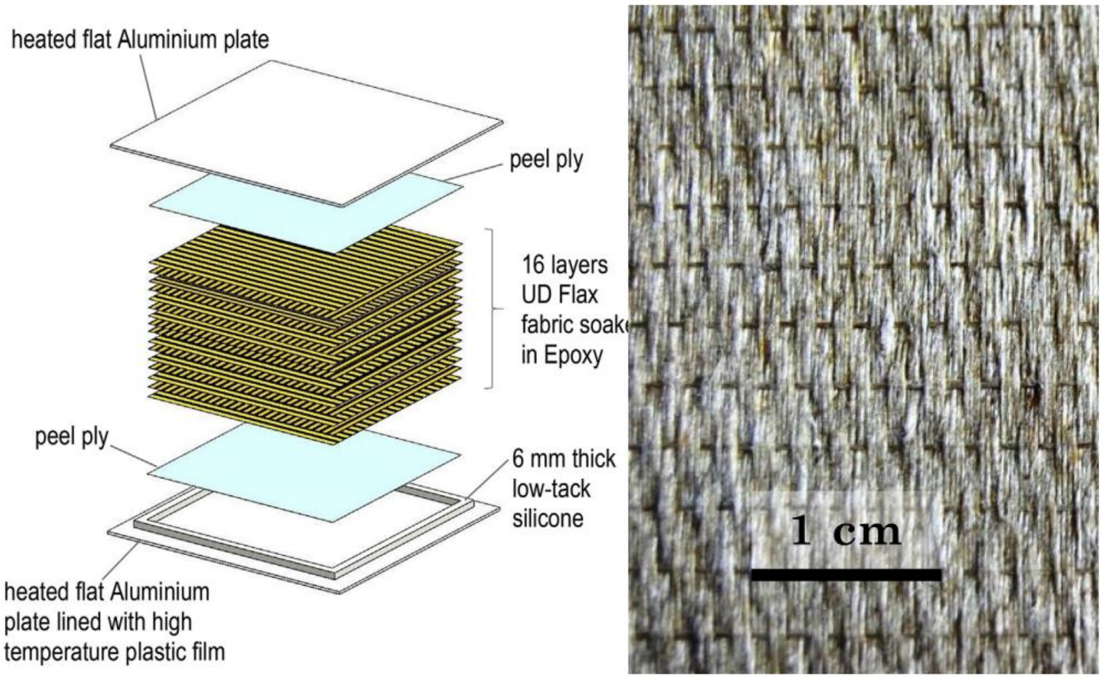


Fig. 1: Layup of the composite material (left) and dry flax fabric (right)

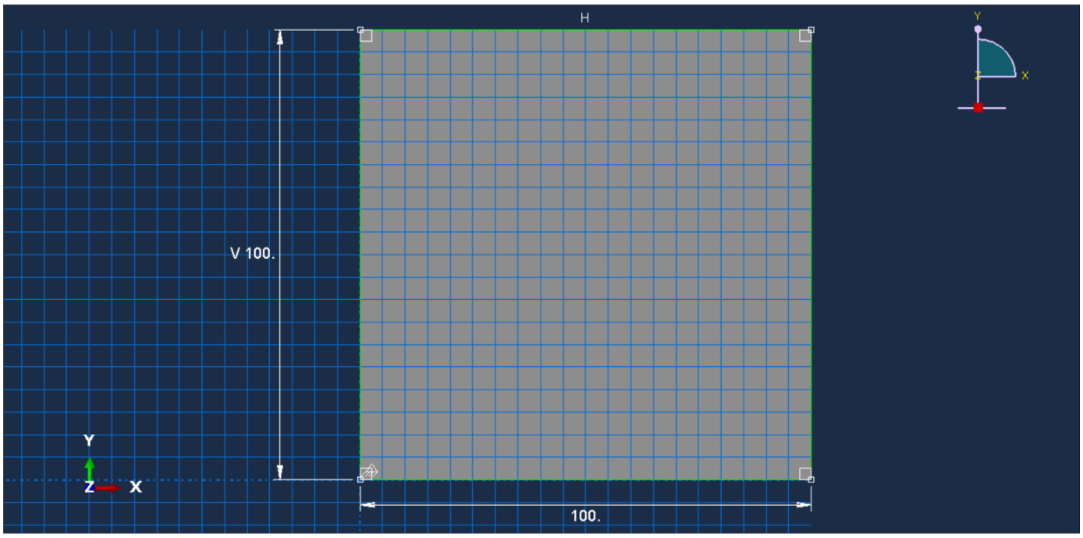


Fig. 2: Sketch of Composite Plate in mm

The values for elastic and composite properties are mentioned below:

- Elastic Properties:

  E (Young's Modulus) = 30 GPa

  Nu (Poison's Ratio) = 0.4

  Density = 1400 kg/m3

- Composite Properties:

  Density = 1400 kg/m3

Table 1: Composite properties of the material

E_1	31 GPa
E_2	4.6 GPa
E_3	4.6 GPa
ν_{21}	0.063
ν_{13}	0.063
ν_{23}	0.353
G_{12}	2000
G_{13}	2000
G_{23}	2000

3. MODELLING

The modelling has been done and the materials properties has been inserted with elastic and composite material for analysis and a mesh has been generated to give an optimum result. To get optimum results mesh convergence has to been studied hence mesh convergence is performed and found that at 25 elements of the entire part for quadratic element type the mesh converges. The percentage is less than 5% and least among all others hence this mesh is approved for further study of analysis.

The plot shown in Fig. 3 depicts the displacement magnitude along with the element size for linear and quadratic element types respectively. The mesh image has been shown in Fig. 4. The linear and quadratic element are similar but linear elements do not capture bending. A quadratic element, or a higher order element utilizes a non-linear shape function. Hence, provide better results.

CASE STUDY 3

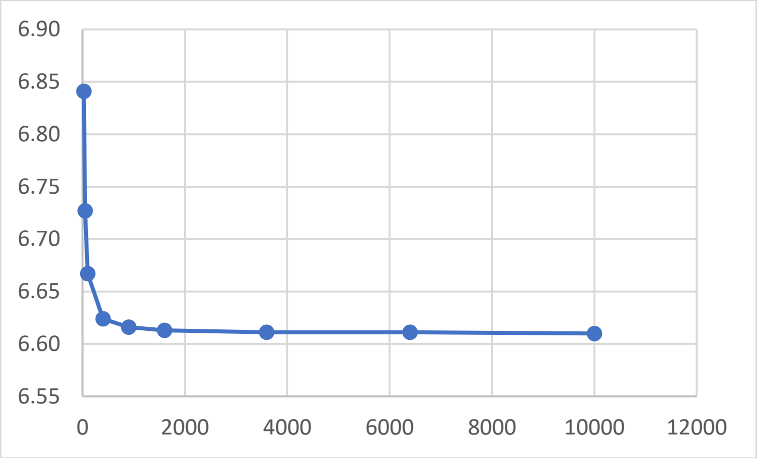


Fig. 3: a) Plot depicting Mesh Convergence Study with respect to U (Magnitude) for Linear Mesh Element Type

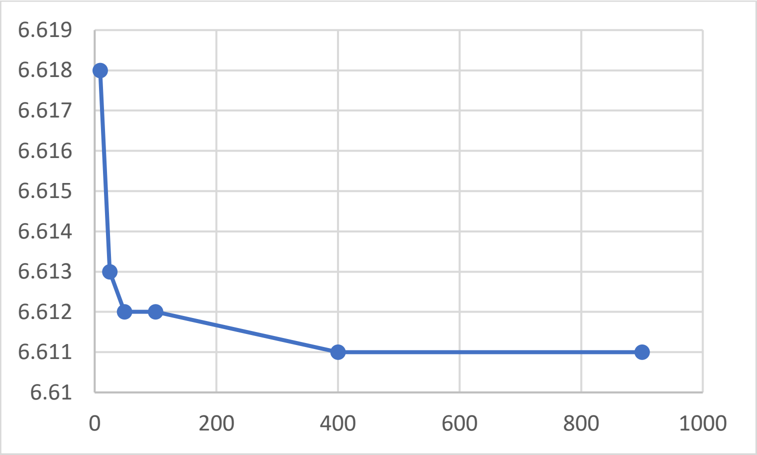


Fig. 3: b) Plot depicting Mesh Convergence Study with respect to U (Magnitude) for Quadratic Mesh Element Type

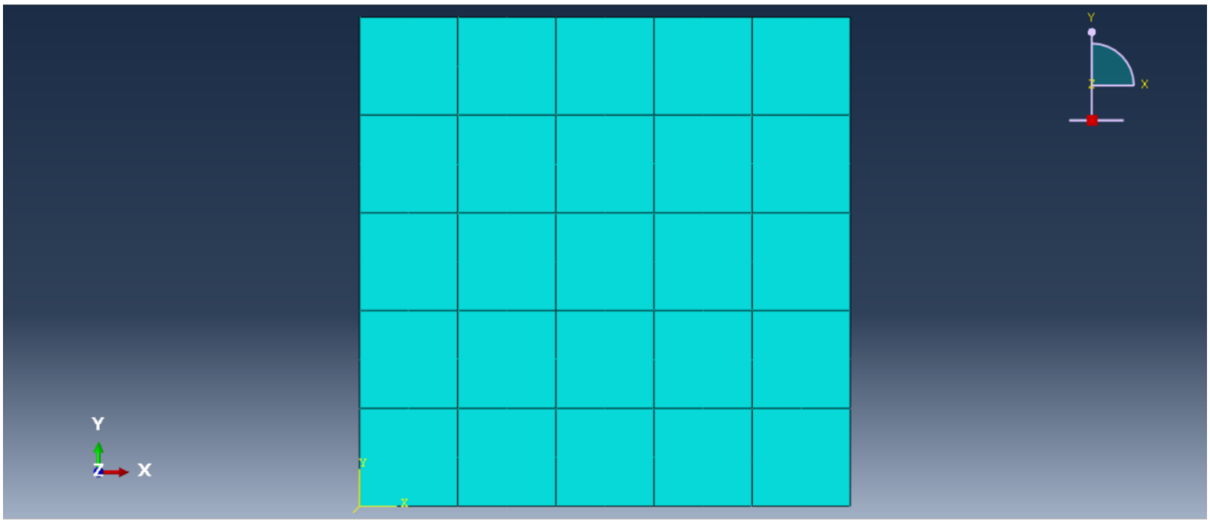


Fig. 4: Mesh for the Composite plate

4. ANALYSIS

The analysis for the composite plate is to be done with the reference material properties with boundary conditions of simply supported nd encastered at the 4 edges of the square plate and determining the deformation at different natural frequencies as in Case 1. Later the plate is subjected to composite material layup to give properties of a composite and assign orientation of 0, 90, +45/-45 and +67.5/-67.5 degrees respectively symmetrically and determining the deformation for respective angles and to design a layup for least deformation.

The boundary conditions for simply supported and encastered are mentioned in table 2 and table 3 respectively and Fig. 5 and Fig. 6 represent these boundary conditions in abaqus.

Table 2: Simply Supported Boundary Conditions

U1	U2	U3	UR1	UR2	UR3
Element 1					
-	0	0	-	0	-
Element 2					
0	-	0	0	-	-
Element 3					
-	-	0	-	0	-
Element 4					
-	-	0	0	-	-

Table 3: Encastered Boundary Conditions

U1	U2	U3	UR1	UR2	UR3
Element 1					
0	0	0	0	0	0
Element 2					
0	0	0	0	0	0
Element 3					
0	0	0	0	0	0
Element 4					
0	0	0	0	0	0

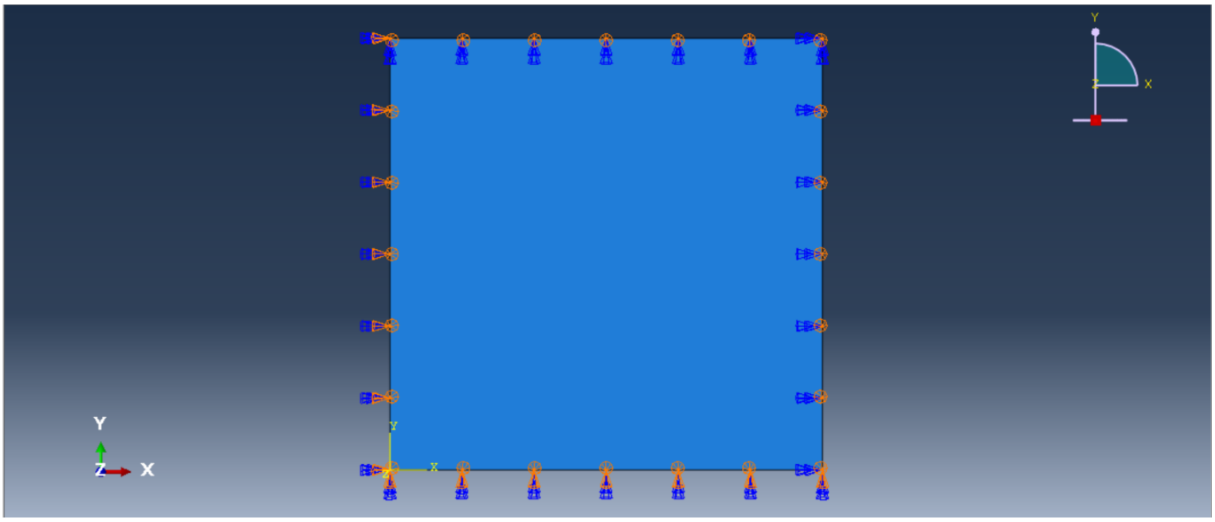


Fig. 5: Boundary Conditions for Composite plate for simply supported condition

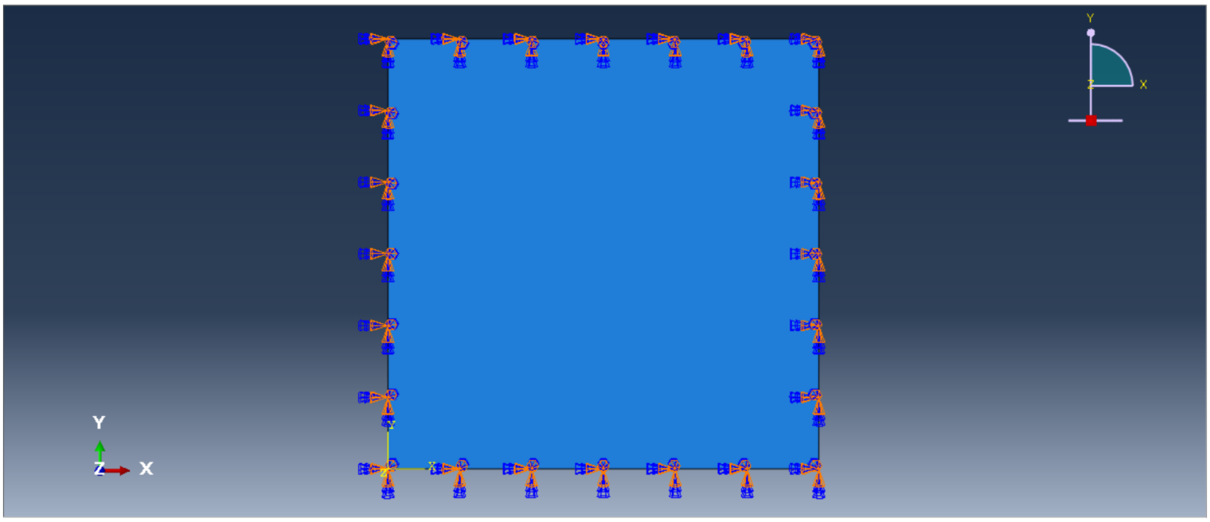


Fig. 6: Boundary Conditions for Composite plate for encastered condition

5. RESULTS:

5.1 EIGEN MODES FOR ELASTIC MATERIAL

In simply supported structure the composite plate with elastic properties the eigen modes have been determined and have been tabulated with modes and deformation magnitude in table 4 and Fig. 7 represents the deformation contour for mode 3 for similar case.

CASE STUDY 3

Table 4: Eigen modes for simply supported condition for elastic material

Eigen Mode	**Eigen Frequency (Hz)**	**Max Deformation (mm)**	**Max Von Mises Stress (MPa)**
1	211.19	3.962	1368
2	308.87	2.993	1783
3	361.99	6.613	2220
4	445.64	3.592	1574
5	474.88	4.132	3432

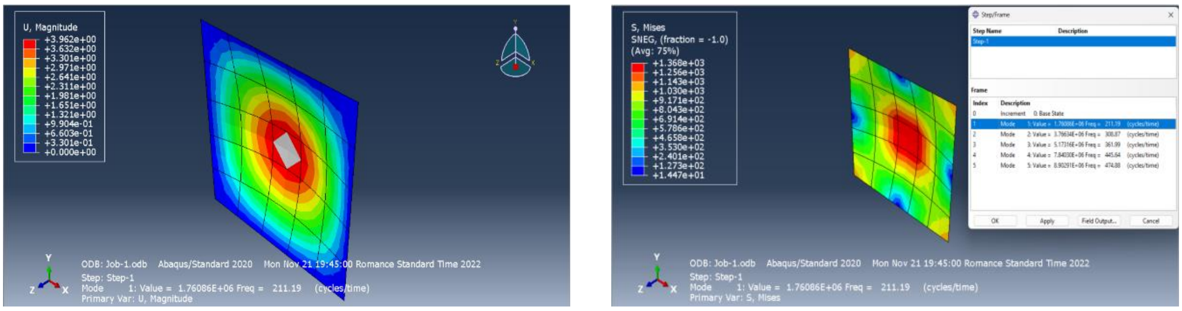


Fig. 7: Deformation and stress contour for elastic material in Mode 1

5.2 <u>EIGEN MODES FOR COMPOSITES AT DIFFERENT LAYUPS</u>

The composite material is assigned to the material and modal analysis is performed and deformation magnitude is found for different layups such as $[0]_{16}$, $[90]_{16}$, $[+45/-45]_{4S}$, $[+67.5/-67.5]_{4S}$ for simply supported boundary conditions.

For different Eigen Modes the frequency and deformation have been found for each layup for the composite material and tabulated in the table 5. The Fig. 8, Fig. 9, Fig. 10 and Fig. 11 represent the contour for deformation for $[0]_{16}$, $[90]_{16}$, $[+45/-45]_{4S}$, $[+67.5/-67.5]_{4S}$ respectively for mode 1.

CASE STUDY 3

Table 5: Eigne Modes, Frequencies and Max Deformation for each Composite layup with unique symmetric fibre direction

Eigen Mode	Eigen Frequency (Hz)	Max Deformation (mm)
For Layup $[0]_{16}$		
1	46.991	6.552
2	83.107	6.573
3	143.30	4.902
4	152.12	6.580
5	153.78	6.674
For Layup $[90]_{16}$		
1	46.991	6.552
2	83.107	6.573
3	143.30	4.902
4	152.12	6.580
5	153.78	6.674
For Layup $[+45/-45]_{4S}$		
1	59.161	6.555
2	129.87	6.749
3	133.61	4.879
4	135.26	6.717
5	168.52	10.500
For Layup $[+67.5/-67.5]_{4S}$		
1	53.471	6.555
2	102.65	6.571
3	140.34	5.220
4	149.06	6.571
5	179.48	6.702

CASE STUDY 3

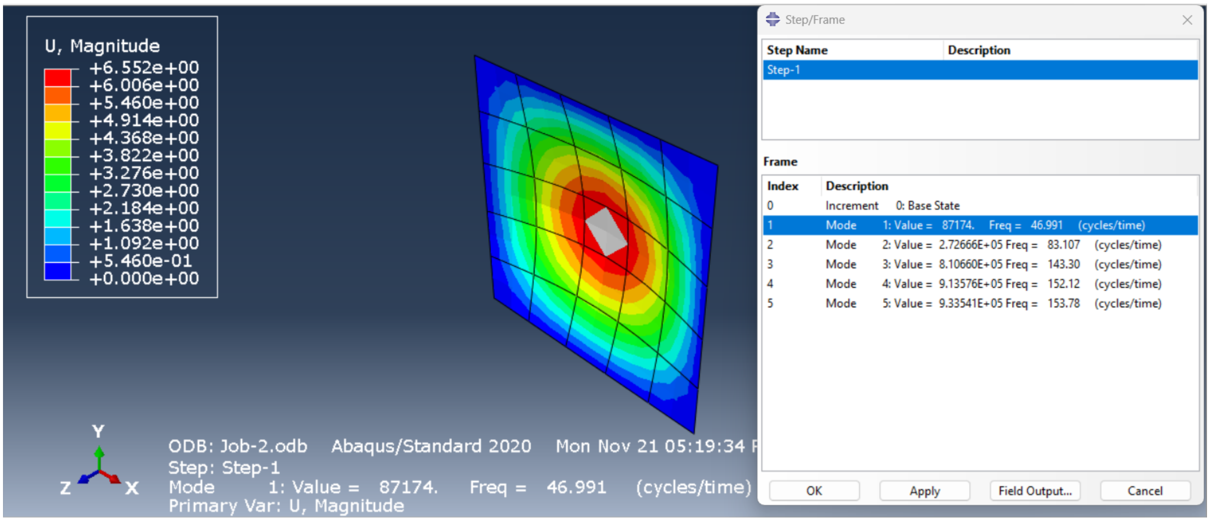


Fig. 8: Deformation contour for composite material in Mode 1 at $[0]_{16}$

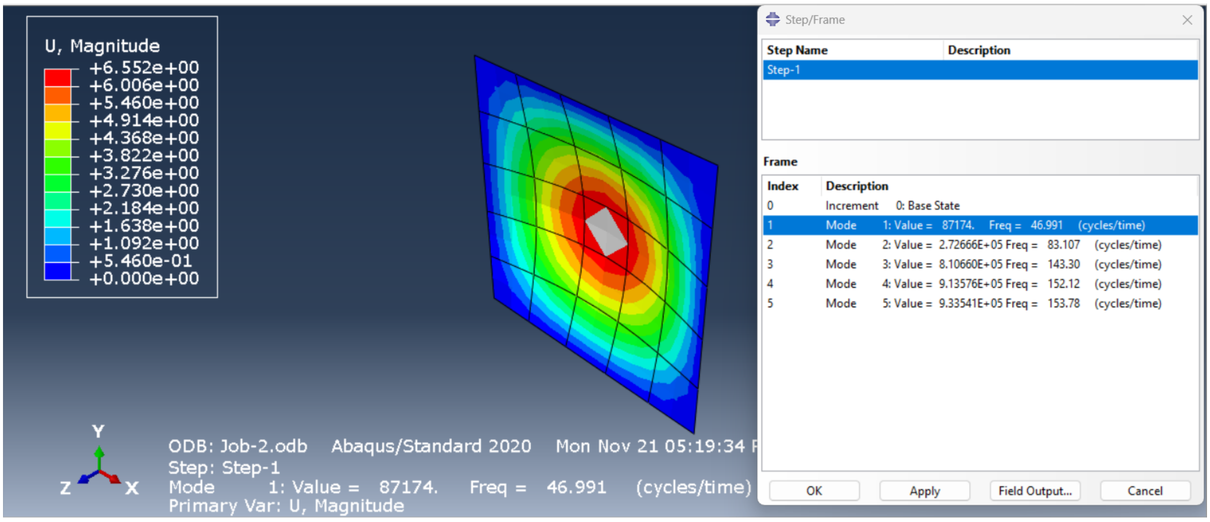


Fig. 9: Deformation contour for composite material in Mode 1 at $[90]_{16}$

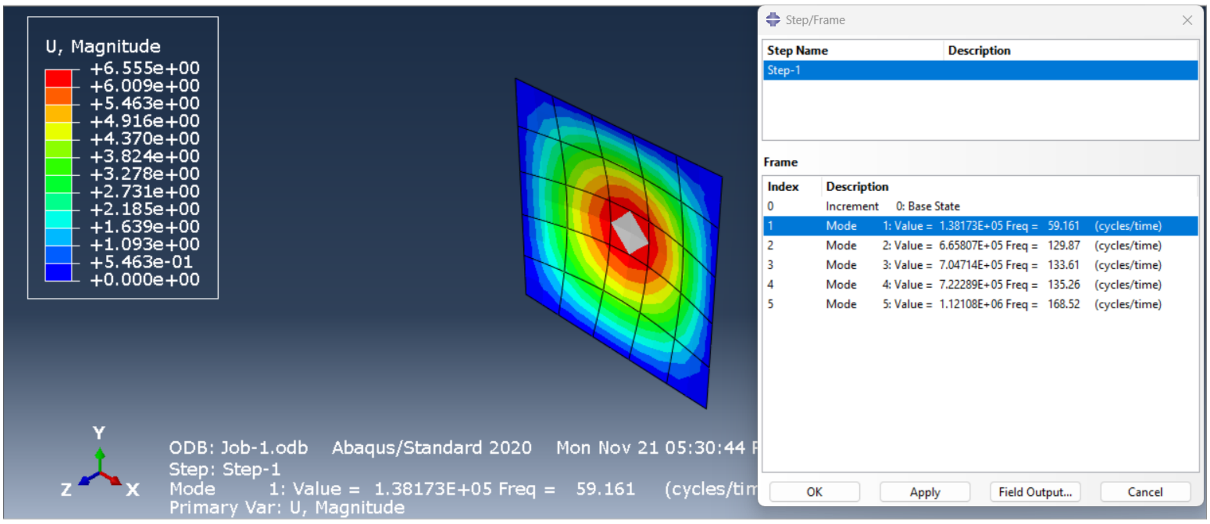


Fig. 10: Deformation contour for composite material in Mode 1 at $[+45/-45]_{4S}$

CASE STUDY 3

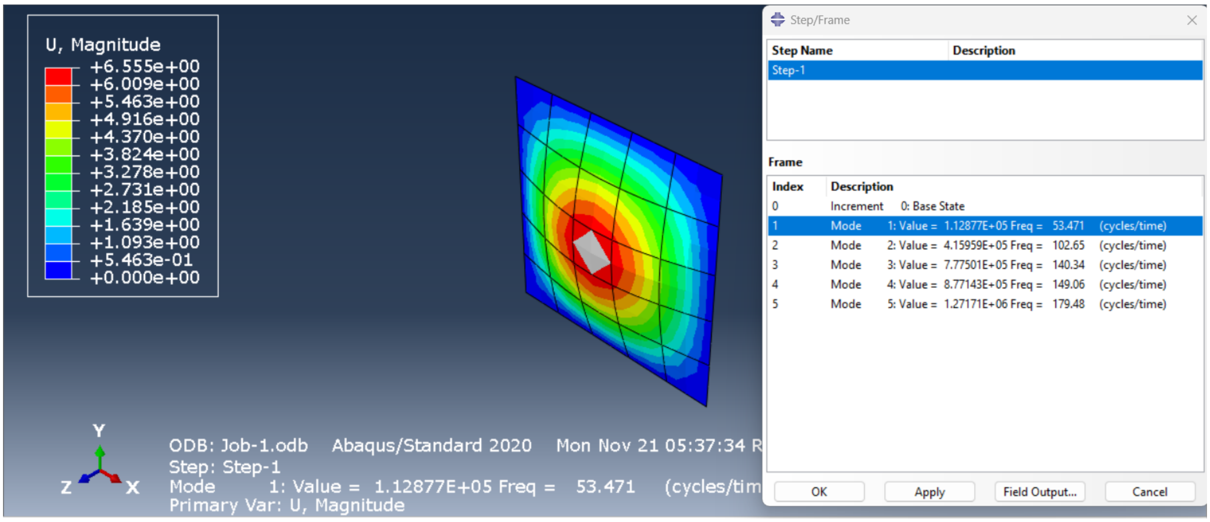


Fig. 11: Deformation contour for composite material in Mode 1 at $[+67.5/-67.5]_{4S}$

5.2.1. COMBINATION LAYUP FOR DECREASING MAXIMUM DISPLACEMENT

Using various analogies theoretically we try to combine some of the layups to determine greater properties.

The symmetrical orientation of 0 and 67.5/-67.5 degree is chosen since the mechanical strength in x-direction is taken care by 0 degree placed fibres but in y-direction and rotational part need to be managed by other fibre in almost y-direction. By this thought we would also like to determine the properties in x and y-direction solely so 0 and 90 degrees. Later we also think about 0,45,90 degrees combination and also 45,67.5 degrees for increasing the natural frequency since it gives more range for entire part to reach the natural frequency and hence probability of failure is reduced to greater extent. All combinations with values have been listed in table 6.

Table 6: Different Layup Combination with different modes, frequencies and deformation

Modes	Frequencies (Hz)	Max Deformation (mm)
1. 0 and 67.5/-67.5- degree layup		
1	51.005	6.555
2	125.21	6.945
3	129.98	6.917
4	190.56	6.754

5	244.42	5.554
2. 0 and 90-degree layup		
1	47.290	6.552
2	118.96	6.576
3	131.57	6.577
4	178.81	6.292
5	239.22	6.684
3. 0, +45/-45, 90-degree layup		
1	52.386	6.555
2	120.79	6.683
3	135.87	6.721
4	195.04	6.937
5	219.13	4.883
4. +45/-45, +67.5/-67.5-degree layup		
1	56.402	6.564
2	119.44	6.827
3	139.84	6.896
4	159.07	5.439
5	203.18	8.114

5.3 COMPARISON OF SIMPLY SUPPORTED AND ENCASTERED BOUNDARY CONDITION

The composite plate is now subjected to simply supported and encastered boundary conditions with the elastic properties and a comparison is made with eigen modes, frequencies and the magnitude of deformation obtained which is tabulated in table 6.

Table 6: Eigen Modes, Frequencies and deformation for different cases

Eigen Modes	Frequency (Hz)	Deformation (mm)
Simply Supported		
1	211.19	3.962
2	308.87	2.993

CASE STUDY 3

3	361.99	6.613
4	445.64	3.592
5	474.88	4.132
Encastered		
1	330.83	4.520
2	592.92	4.353
3	592.92	4.353
4	803.00	3.890
5	882.06	4.250

Comparing the results using the vibration equation which is used to find modal frequency analytically is

$$[M]\,[\ddot{U}] + [K]\,[U] = 0 \qquad \text{………..(1)}$$

Where, M = mass in kg

U: = acceleration due to gravity in m^2/s

K = Spring stiffness in kg*m/s

U = Displacement in m

6. DISCUSSIONS OF RESULTS

The analysis performed for isotropic material show different displacement at different modes and frequencies. This acts as a reference using the simply supported condition.

The composite material with layups such as: $[0]_{16}$, $[90]_{16}$, $[+45/-45]_{4S}$, $[+67.5/-67.5]_{4S}$ show different properties with [0] and [90] degree fibre orientation shows exact properties of frequencies and deformation at the same modes. But as we increase the inclination to 45 degrees, frequency hypes up and displacement also increases massively than other layups. When we turn the inclination to 67.5 degrees the frequency and deformation also increases but to a little extent than 0 and 90 degrees. So to get the higher frequencies and lower deformations the layup's are combined and found that +45/-45 and +67.5/-67.5 degree orientation provides greater frequencies and displacements as before. So, to decrease displacement to some values

0- and 90-degree layup are combined but frequency does not increase. So based on regular applications required we finalize a layup.

The comparison of simply supported and encastered we only change the boundary conditions hence only mass and acceleration cancel out the other part to get final force as zero and we remain with only [K] and [U] in which displacement is found using FEA analysis and we find deformation is almost half of the displacement values obtained using simply supported conditions and hence we can infer that spring stiffness need to be doubled if displacement is halved. Also, the frequency is increased to greater extent. Due to this phenomenon the encastered is more preferred for modal analysis to safeguard the system for higher frequencies and to get more safety for the application of the part.

7. CONCLUSION

The simulation of composite plate with isotropic material and composite material is performed and for different modes the deformation and frequencies are observed and compared. Also, the comparison for simply supported and encastered boundary condition tells us that for modal analysis for very critical application to be created in encastered boundary condition. Also, the composite plate needs to be designed according to application of that component and its maximum properties required in the preferred direction along with the frequency it can reach based on the surrounding and in which mode is it going to be operated.

CASE STUDY 4

Simulation of a Stamping Process

1. INTRODUCTION

Stamping or pressing is a manufacturing process used to convert flat metal sheets into final shapes. In this process, a flat sheet of metal, either in the form of a blank or in the form of a coil, is placed in a stamping press where a tool and die surface forms the metal into a final shape. Stamping encompasses a variety of sheet metal forming manufacturing processes, such as punching using a machine press or stamping press, blanking, embossing, bending, flanging, and coining. This could be a one-step process where each stroke of the press creates the desired shape on the sheet metal part, or it could be done through a series of steps.

The first step is to simulate the process and find the various interactions, boundary conditions and loadings applied on the plate for the stamping operation which is beneficial to know the resistance of the material which it can resist so it does fracture.

2. DESIGN

The image of the plate with its dimensions is depicted in Fig. 1. The sketch used for stamping plate, punch, blank holder and die has been shown in Fig. 2, Fig. 3, Fig. 4 and Fig. 5 respectively. The part has been created in 2D wire for the plate which is deformable and stamping plate, die and blank holder as an analytic rigid body with wire. The entire sketch has been designed in mm.

The distance between die, blank holder and the punch have been given distance based on the thickness of the plate. The model has been symmetrical so the sketch has been performed for half of the model to reduce the computation time and complexity of the problem. The model has 4 main parts consisting a plate, die, blank holder and punch. The plate has been assigned dimensions as shown in Fig. 1.

CASE STUDY 4

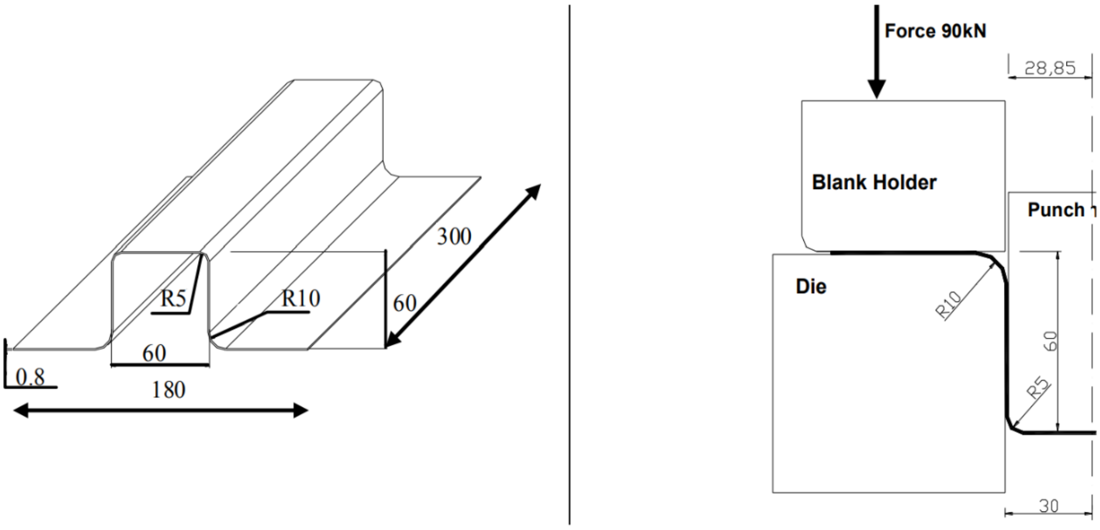


Fig. 1: Plate with Stamping process

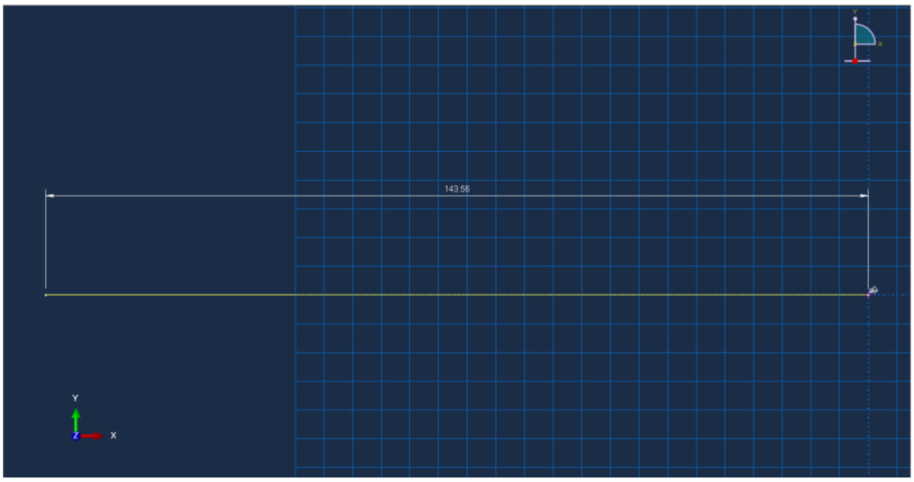


Fig. 2: Sketch of Plate

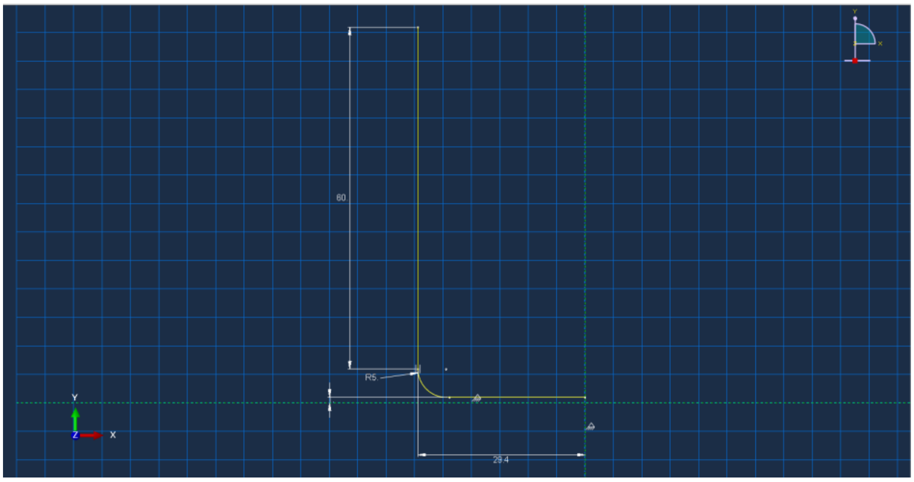


Fig. 3: Sketch of Punch

CASE STUDY 4

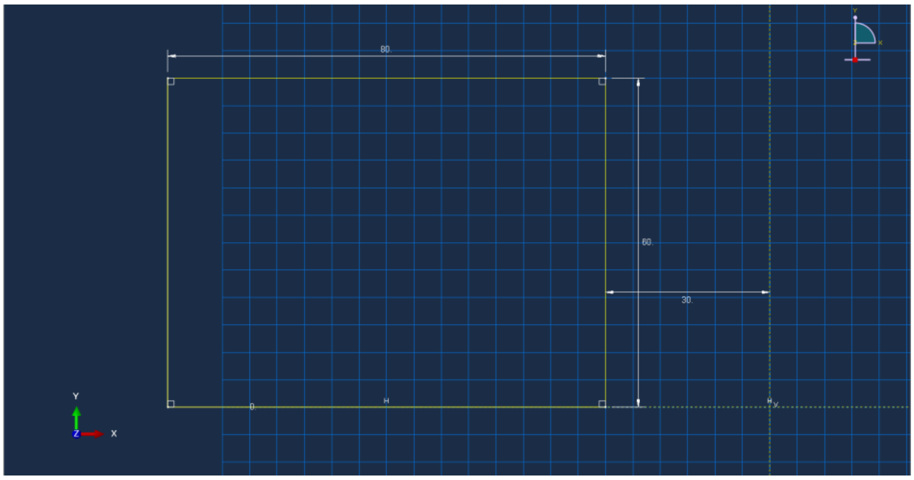


Fig. 4: Sketch of Blank Holder

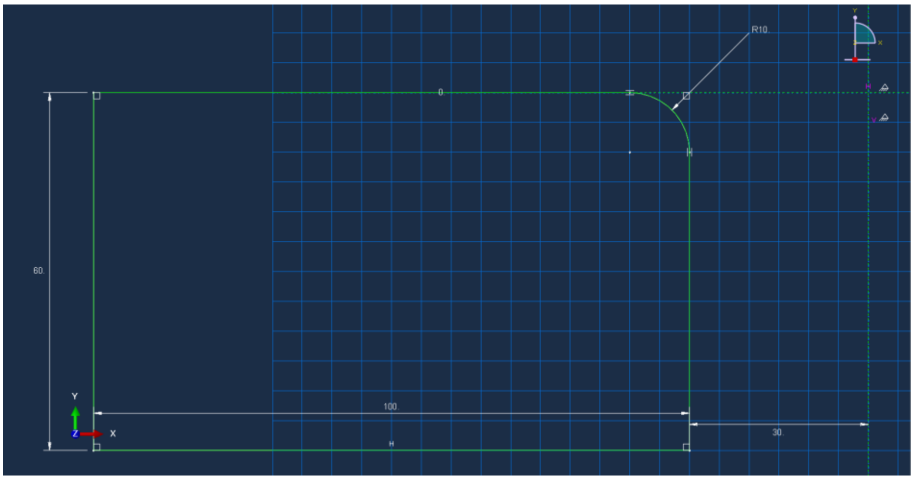


Fig. 5: Sketch of Die

The values for elastic and plastic properties are mentioned in table 1.

Table 1: Elastic & Plastic Behaviour

- **<u>ELASTIC BEHAVIOUR</u>**

YOUNG'S MODULUS	210 GPa
POISON'S RATIO	0.3

- **<u>PLASTIC BEHAVIOUR</u>**

YOUNG'S MODULUS	**STRAIN**
400	0
477.495949	0.002
495.408705	0.004
507.749529	0.006
517.461894	0.008
525.594322	0.01

532.65523	0.012
538.93396	0.014
544.612555	0.016
549.813778	0.018
554.624747	0.02
559.109756	0.022
563.317746	0.024
567.286929	0.026

3. MODELLING

The modelling has been done and the materials properties has been inserted with elastic and elasto-plastic material analysis and a mesh has been generated to give an optimum result.

Mesh has been generated for global size as 1. The mesh type considered as beam elements with B21 linear which is giving good results than quadratic type. Hence mesh has been generated with 144 elements. The mesh has to be assigned only for plate and none of the other bodies since other parts have been assigned rigid bodies.

4. ANALYSIS

All parts need to be assembled and need to provide a tolerance of the thickness of the plate so it does not overlap. In this model thickness of 0.6, 0.8 and 1.1 mm have been analyzed and similar tolerance has been provided between the plates and rigid bodies for optimum condition.

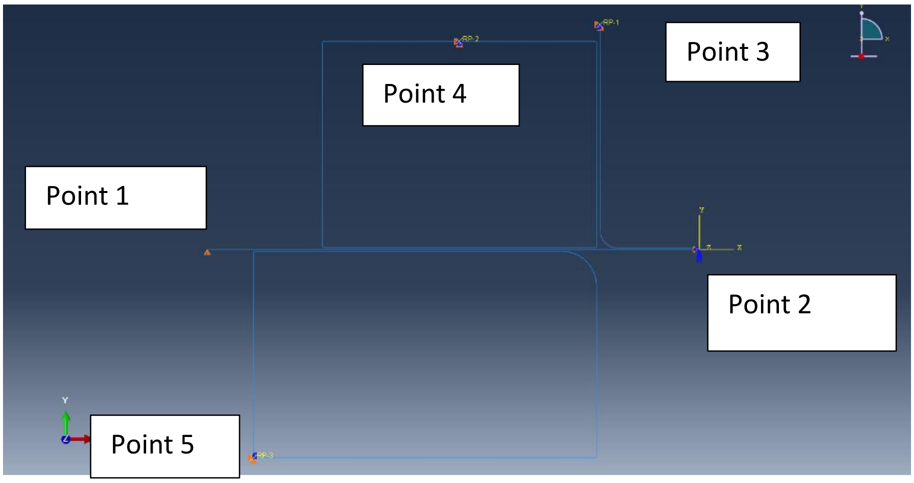


Fig. 6: Points depicting various points with boundary conditions

CASE STUDY 4

Reference points are needed to add the force and other various boundary conditions. The reference points are given rigid body constraint on the rigid body and interactions are provided between the surfaces of hard contact with friction co-efficient of 0.2. The various boundary conditions in U1, U2 and UR1 have been shown in the table 2 below.

Table 2: Boundary condition of various points based on the Fig. 6

Points	U1	U2	UR1	Comments
1	-	0	-	-
2	0	-	-	X-Symmetric
3	0	-60	0	Displacement
4	-	0	-	Load of 90kN is applied
5	0	0	0	Encastered

5. RESULTS:

5.1 FORCE ON THE PUNCH

We apply a load of 90kN on the Blank and find the various reaction forces and the variation with the various thickness of the plate with the same tolerance through a graphical representation which is observed in the Fig. 7.

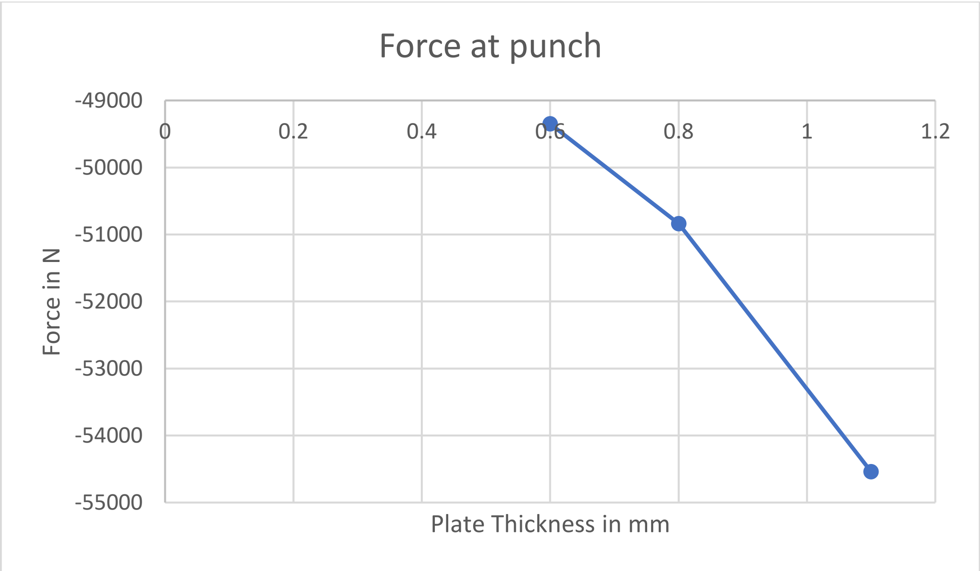


Fig. 7: Plot depicting the Force versus Plate thickness

CASE STUDY 4

From this we observe that as when plate thickness increases the reaction force at punch increases in magnitude.

5.2 MAXIMUM STRESS AND MAXIMUM PLASTIC STRAIN

Once we obtained variation with reactant force, we find the variation with maximum stress and plastic strain with respect to thickness in Fig. 11 and Fig. 12 respectively.

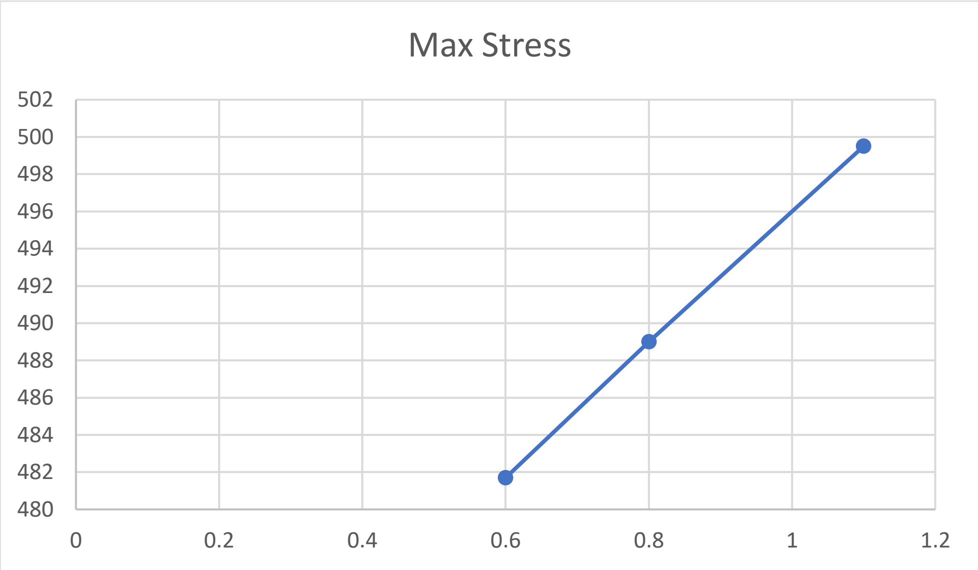


Fig. 11: Max stress with respect to thickness

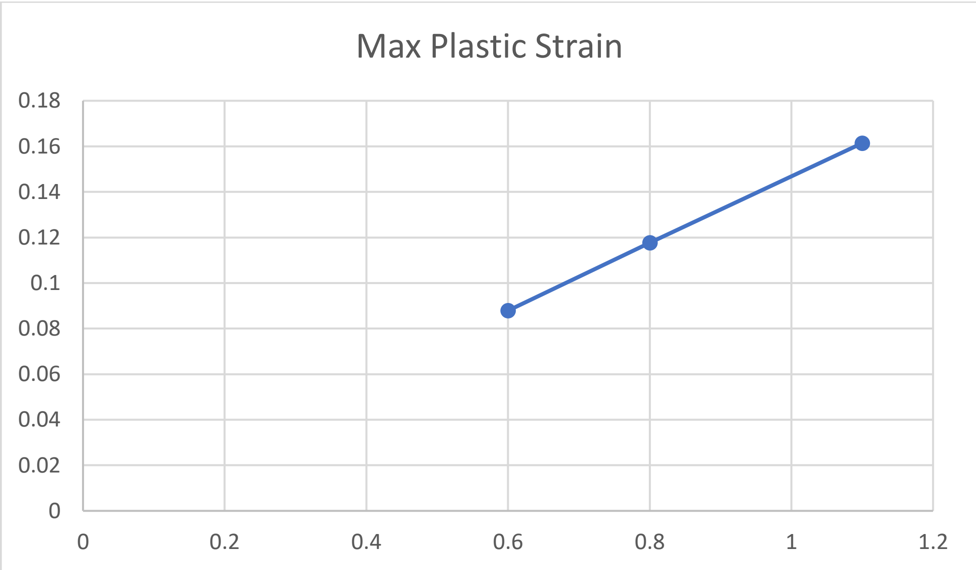


Fig. 12: Max Plastic strain with respect to thickness

The Fig. 13, 14 and 15 shows the contours of Max stress, max plastic strain and Force for 0.8 thickness. Fig. 16 shows the reaction force variation with stamping with time

CASE STUDY 4

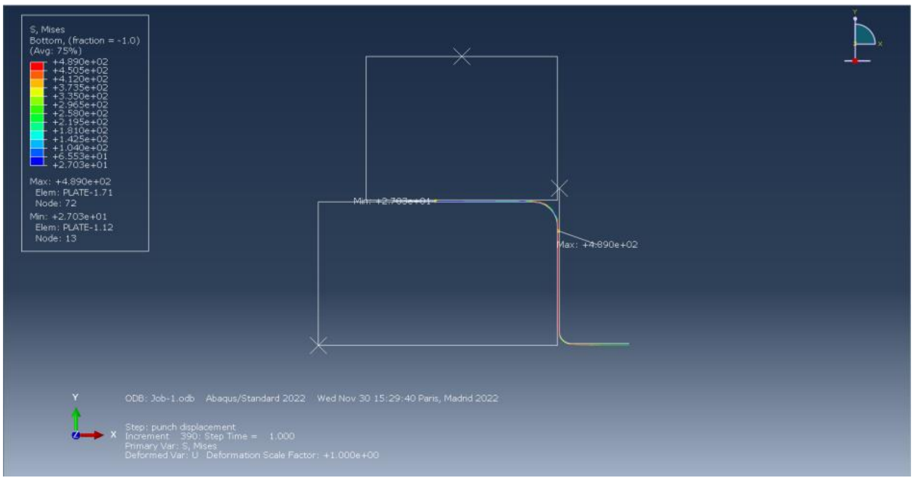


Fig. 13: Stress contour for 0.8 mm thickness

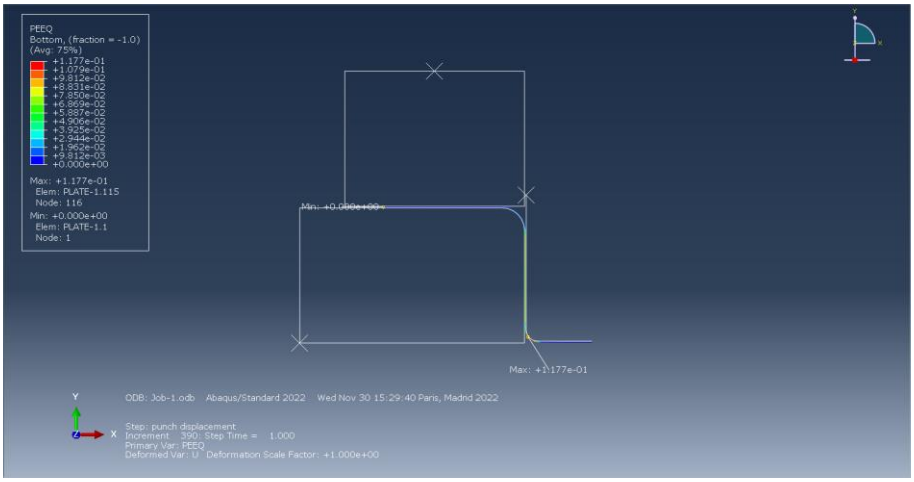


Fig. 14: Plastic Strain contour for 0.8 mm thickness

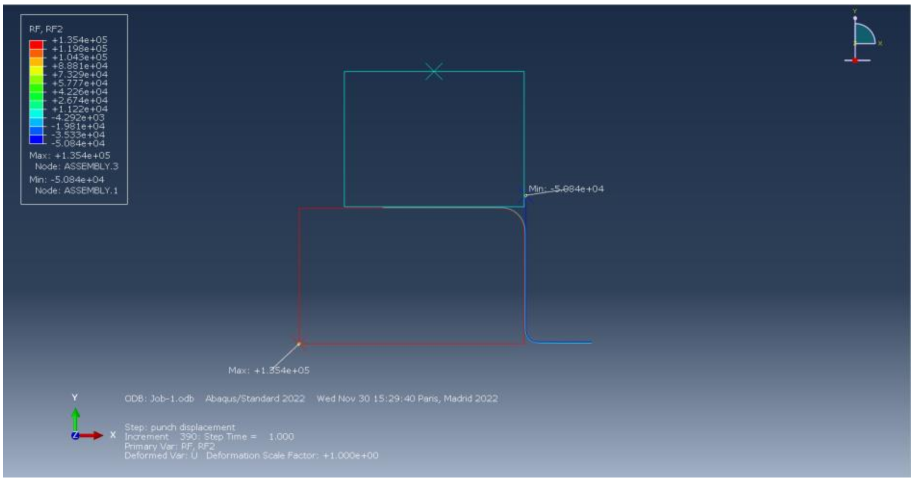


Fig. 15: Reaction Force contour for 0.8 mm thickness

CASE STUDY 4

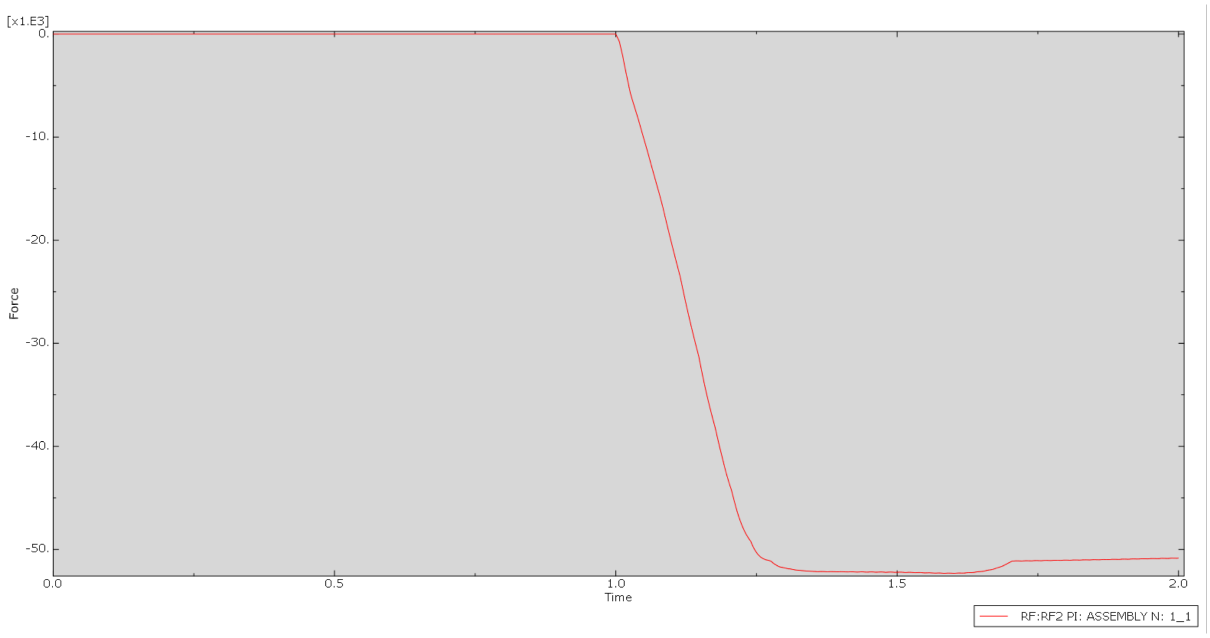


Fig. 16: Reaction Force variation with respect to time for 0.8 mm thickness

6. CONCLUSION

This simulation shows the variation of force increases with respect to thickness of the plate and also when time increase that is from initial position to the position where stamping is made, force is increased and the force comes back to original state. Also, the stress is max at place where the plate bends and maximum plastic strain where the end of the stamping attached to plate to leave some energy to the material as plasticity. Thickness place a very importat role while defining the stamping process. Hence also considering a beam element is also more sensible option to opt for good results. And also considering symmetric option reduces time and also similar results as entire part.

CASE STUDY 4

Simulation of a Stamping Process

1. INTRODUCTION

Stamping or pressing is a manufacturing process used to convert flat metal sheets into final shapes. In this process, a flat sheet of metal, either in the form of a blank or in the form of a coil, is placed in a stamping press where a tool and die surface forms the metal into a final shape. Stamping encompasses a variety of sheet metal forming manufacturing processes, such as punching using a machine press or stamping press, blanking, embossing, bending, flanging, and coining. This could be a one-step process where each stroke of the press creates the desired shape on the sheet metal part, or it could be done through a series of steps.

The first step is to simulate the process and find the various interactions, boundary conditions and loadings applied on the plate for the stamping operation which is beneficial to know the resistance of the material which it can resist so it does fracture.

2. DESIGN

The image of the plate with its dimensions is depicted in Fig. 1. The sketch used for stamping plate, punch, blank holder and die has been shown in Fig. 2, Fig. 3, Fig. 4 and Fig. 5 respectively. The part has been created in 2D wire for the plate which is deformable and stamping plate, die and blank holder as an analytic rigid body with wire. The entire sketch has been designed in mm.

The distance between die, blank holder and the punch have been given distance based on the thickness of the plate. The model has been symmetrical so the sketch has been performed for half of the model to reduce the computation time and complexity of the problem. The model has 4 main parts consisting a plate, die, blank holder and punch. The plate has been assigned dimensions as shown in Fig. 1.

CASE STUDY 4

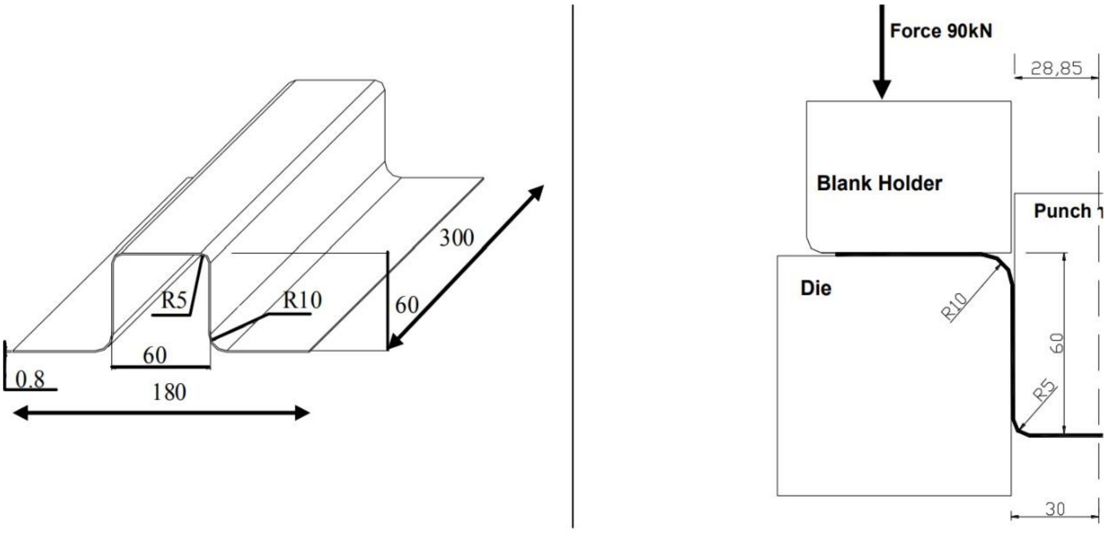


Fig. 1: Plate with Stamping process

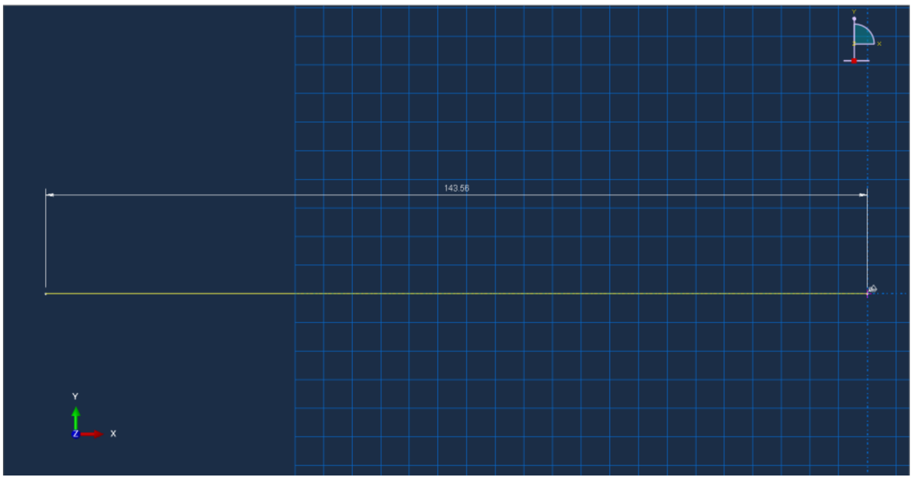


Fig. 2: Sketch of Plate

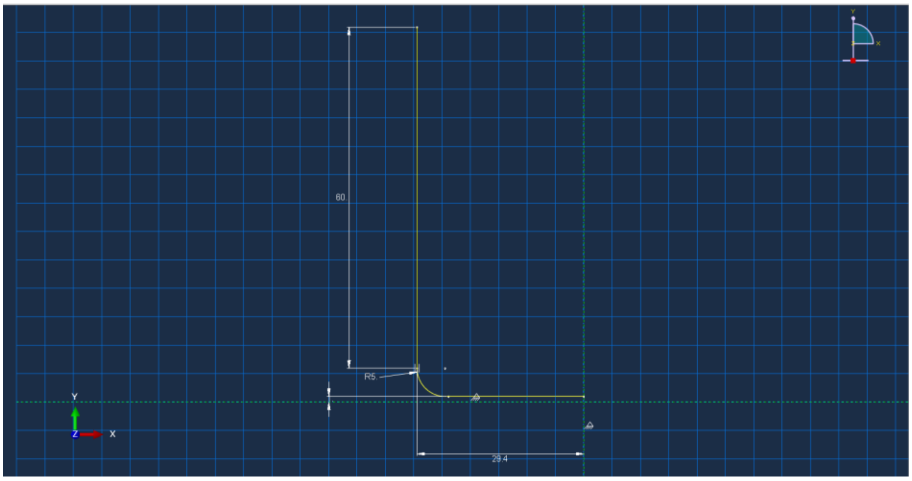


Fig. 3: Sketch of Punch

CASE STUDY 4

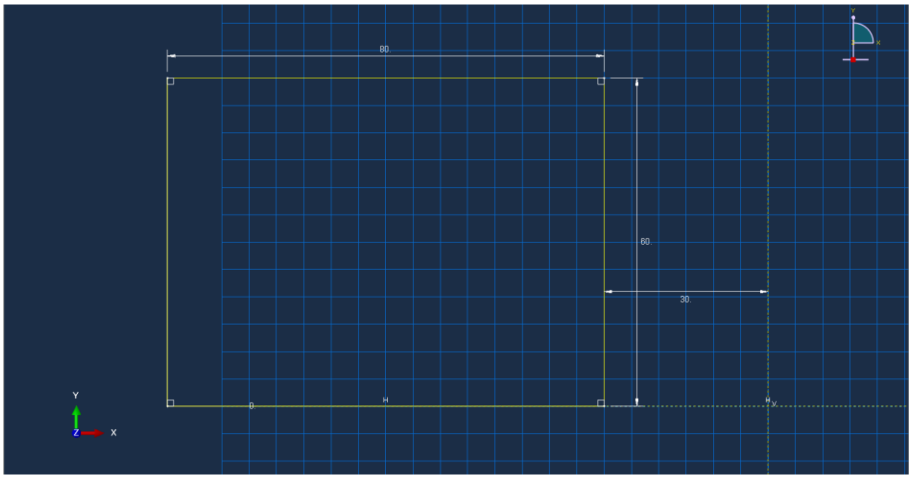


Fig. 4: Sketch of Blank Holder

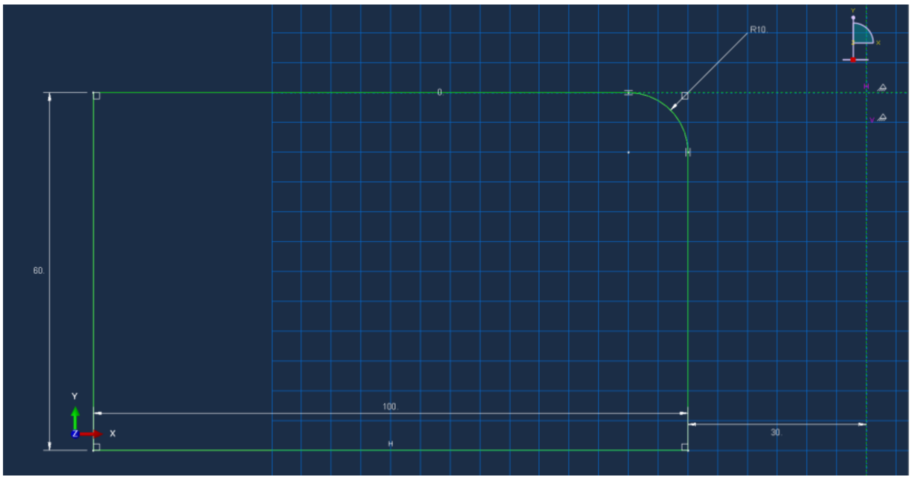


Fig. 5: Sketch of Die

The values for elastic and plastic properties are mentioned in table 1.

Table 1: Elastic & Plastic Behaviour

- **ELASTIC BEHAVIOUR**

YOUNG'S MODULUS	210 GPa
POISON'S RATIO	0.3

- **PLASTIC BEHAVIOUR**

YOUNG'S MODULUS	**STRAIN**
400	0
477.495949	0.002
495.408705	0.004
507.749529	0.006
517.461894	0.008
525.594322	0.01

532.65523	0.012
538.93396	0.014
544.612555	0.016
549.813778	0.018
554.624747	0.02
559.109756	0.022
563.317746	0.024
567.286929	0.026

3. MODELLING

The modelling has been done and the materials properties has been inserted with elastic and elasto-plastic material analysis and a mesh has been generated to give an optimum result.

Mesh has been generated for global size as 1. The mesh type considered as beam elements with B21 linear which is giving good results than quadratic type. Hence mesh has been generated with 144 elements. The mesh has to be assigned only for plate and none of the other bodies since other parts have been assigned rigid bodies.

4. ANALYSIS

All parts need to be assembled and need to provide a tolerance of the thickness of the plate so it does not overlap. In this model thickness of 0.6, 0.8 and 1.1 mm have been analyzed and similar tolerance has been provided between the plates and rigid bodies for optimum condition.

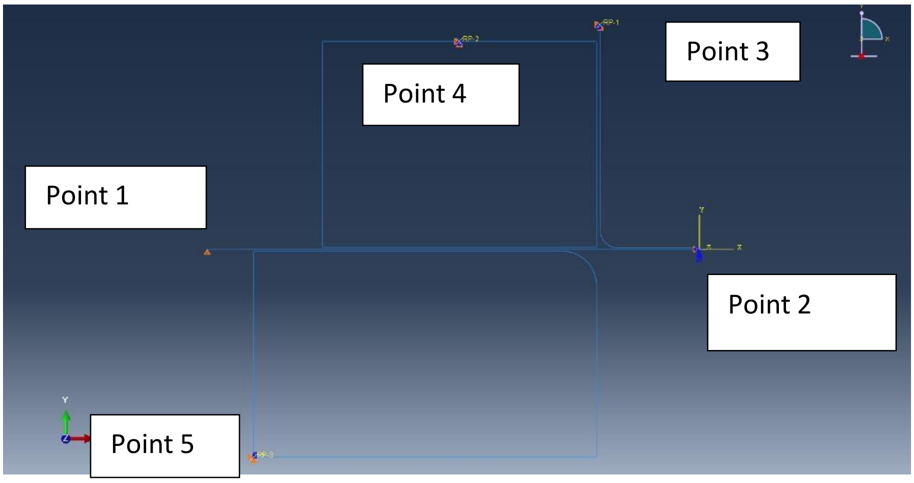


Fig. 6: Points depicting various points with boundary conditions

CASE STUDY 4

Reference points are needed to add the force and other various boundary conditions. The reference points are given rigid body constraint on the rigid body and interactions are provided between the surfaces of hard contact with friction co-efficient of 0.2. The various boundary conditions in U1, U2 and UR1 have been shown in the table 2 below.

Table 2: Boundary condition of various points based on the Fig. 6

Points	U1	U2	UR1	Comments
1	-	0	-	-
2	0	-	-	X-Symmetric
3	0	-60	0	Displacement
4	-	0	-	Load of 90kN is applied
5	0	0	0	Encastered

5. RESULTS:

5.1 FORCE ON THE PUNCH

We apply a load of 90kN on the Blank and find the various reaction forces and the variation with the various thickness of the plate with the same tolerance through a graphical representation which is observed in the Fig. 7.

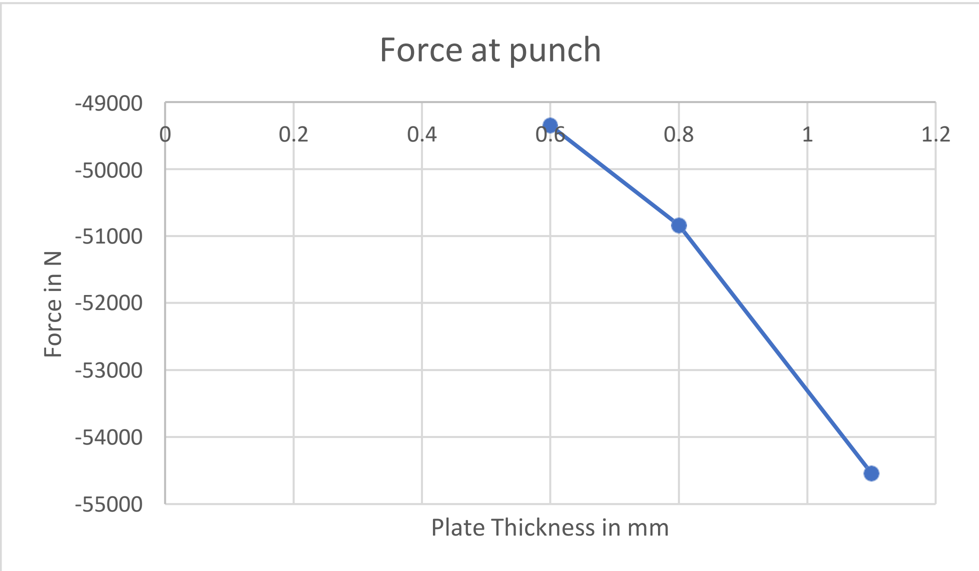


Fig. 7: Plot depicting the Force versus Plate thickness

From this we observe that as when plate thickness increases the reaction force at punch increases in magnitude.

5.2 MAXIMUM STRESS AND MAXIMUM PLASTIC STRAIN

Once we obtained variation with reactant force, we find the variation with maximum stress and plastic strain with respect to thickness in Fig. 11 and Fig. 12 respectively.

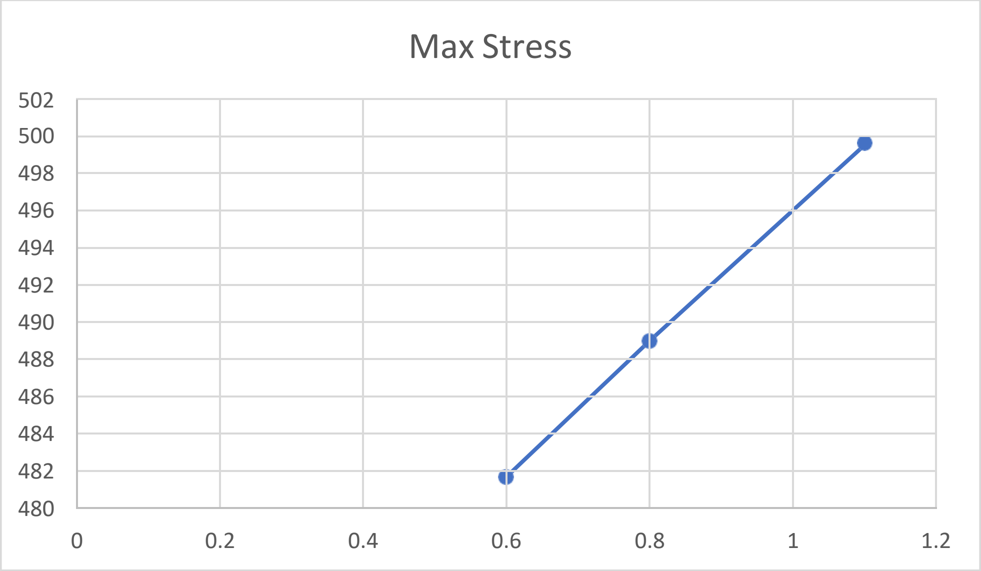


Fig. 11: Max stress with respect to thickness

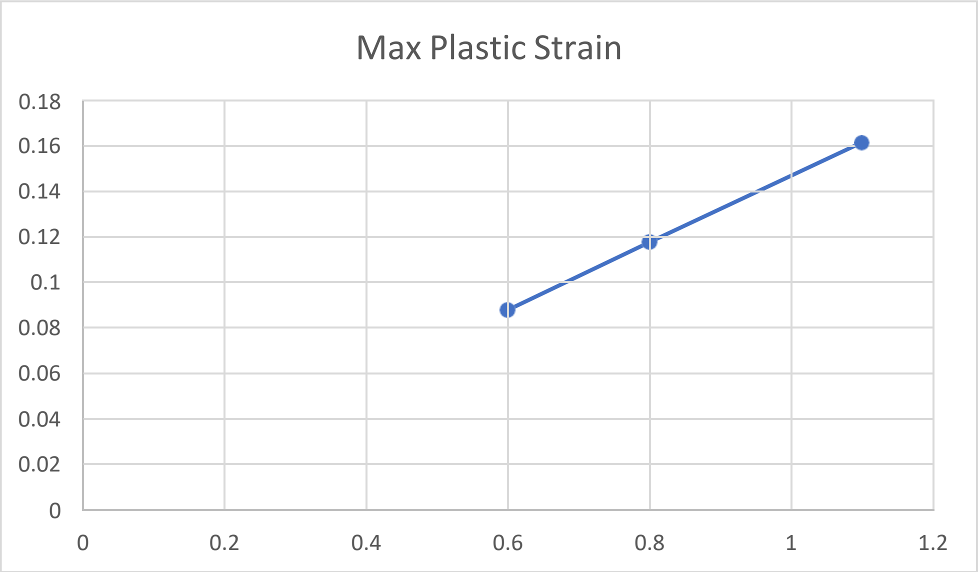


Fig. 12: Max Plastic strain with respect to thickness

The Fig. 13, 14 and 15 shows the contours of Max stress, max plastic strain and Force for 0.8 thickness. Fig. 16 shows the reaction force variation with stamping with time

CASE STUDY 4

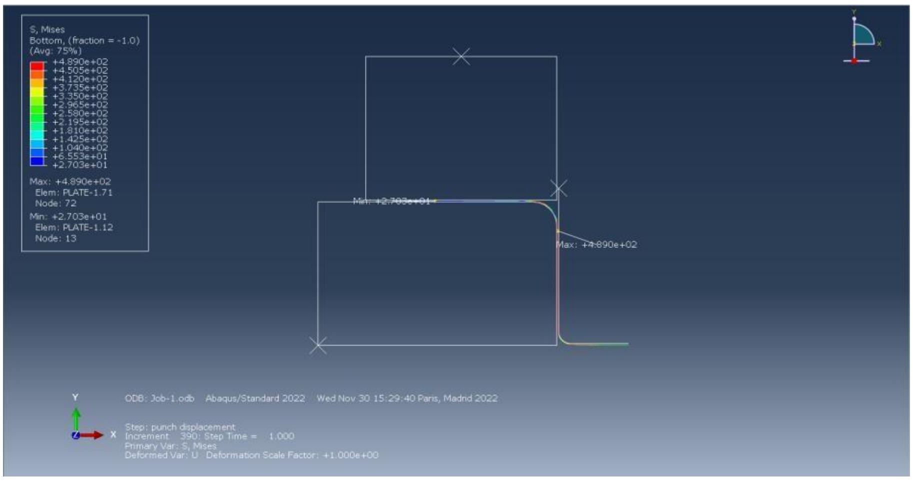


Fig. 13: Stress contour for 0.8 mm thickness

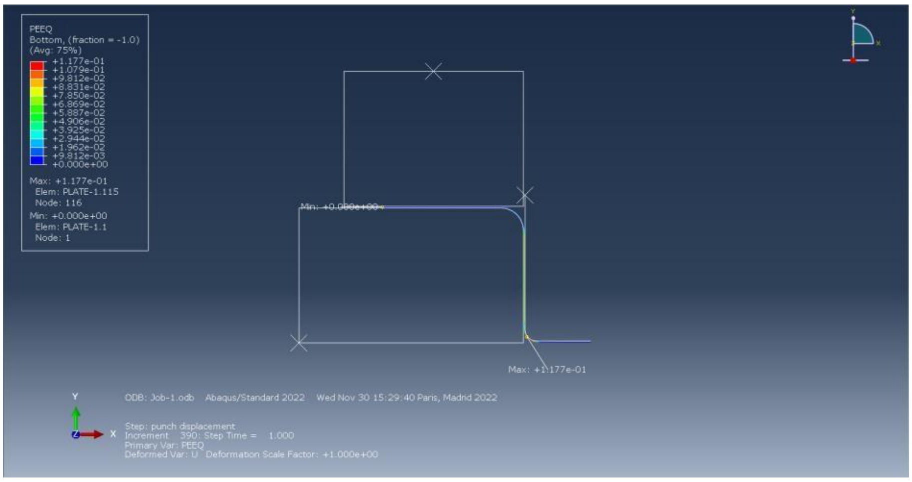


Fig. 14: Plastic Strain contour for 0.8 mm thickness

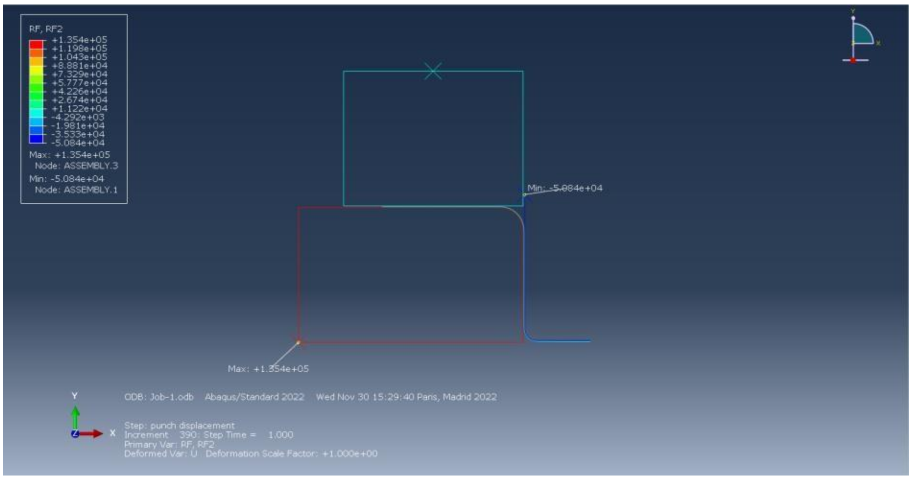


Fig. 15: Reaction Force contour for 0.8 mm thickness

CASE STUDY 4

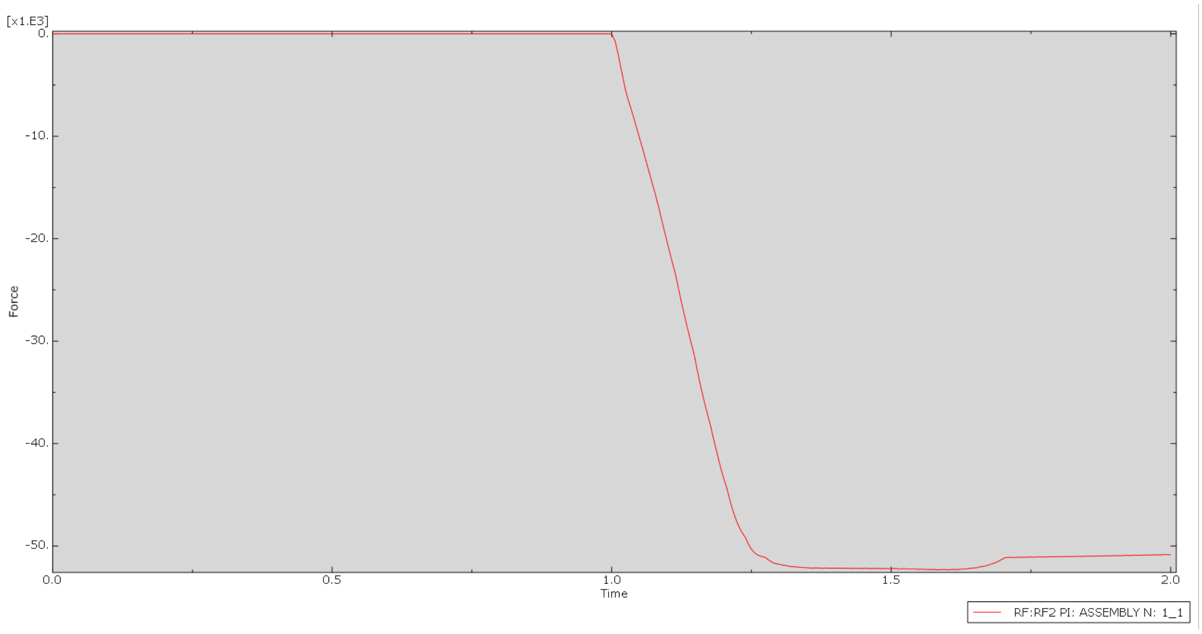


Fig. 16: Reaction Force variation with respect to time for 0.8 mm thickness

5.3 AUTOMATE CODE

Python Code highlighted below shows the variable used and a value assigning them to give results

```
## Punch part

### Radius Change

for radii in range(3, 11)
{
mdb.models['Model-1'].ConstrainedSketch(name='__profile__', sheetSize=200.0)
mdb.models['Model-1'].sketches['__profile__'].Line(point1=(0.0, 0.0), point2=(
    -25.0, 0.0))
mdb.models['Model-1'].sketches['__profile__'].HorizontalConstraint(
    addUndoState=False, entity=
    mdb.models['Model-1'].sketches['__profile__'].geometry[2])
mdb.models['Model-1'].sketches['__profile__'].Line(point1=(-25.0, 0.0), point2=
    (-25.0, 30.0))
mdb.models['Model-1'].sketches['__profile__'].VerticalConstraint(addUndoState=
    False, entity=mdb.models['Model-1'].sketches['__profile__'].geometry[3])
mdb.models['Model-1'].sketches['__profile__'].PerpendicularConstraint(
    addUndoState=False, entity1=
    mdb.models['Model-1'].sketches['__profile__'].geometry[2], entity2=
    mdb.models['Model-1'].sketches['__profile__'].geometry[3])
mdb.models['Model-1'].sketches['__profile__'].Line(point1=(-25.0, 30.0),
    point2=(0.0, 30.0))
mdb.models['Model-1'].sketches['__profile__'].HorizontalConstraint(
    addUndoState=False, entity=
    mdb.models['Model-1'].sketches['__profile__'].geometry[4])
mdb.models['Model-1'].sketches['__profile__'].PerpendicularConstraint(
```

CASE STUDY 4

```
    addUndoState=False, entity1=
    mdb.models['Model-1'].sketches['__profile__'].geometry[3], entity2=
    mdb.models['Model-1'].sketches['__profile__'].geometry[4])
mdb.models['Model-1'].sketches['__profile__'].ObliqueDimension(textPoint=(
    -15.1094169616699, 32.3786430358887), value=28.9, vertex1=
    mdb.models['Model-1'].sketches['__profile__'].vertices[2], vertex2=
    mdb.models['Model-1'].sketches['__profile__'].vertices[3])
mdb.models['Model-1'].sketches['__profile__'].undo()
mdb.models['Model-1'].sketches['__profile__'].ObliqueDimension(textPoint=(
    -36.7775917053223, 10.4368934631348), value=75.0, vertex1=
    mdb.models['Model-1'].sketches['__profile__'].vertices[1], vertex2=
    mdb.models['Model-1'].sketches['__profile__'].vertices[2])
mdb.models['Model-1'].sketches['__profile__'].undo()
mdb.models['Model-1'].sketches['__profile__'].FixedConstraint(entity=
    mdb.models['Model-1'].sketches['__profile__'].vertices[0])
mdb.models['Model-1'].sketches['__profile__'].ObliqueDimension(textPoint=(
    -40.0855865478516, 16.8353633880615), value=75.0, vertex1=
    mdb.models['Model-1'].sketches['__profile__'].vertices[1], vertex2=
    mdb.models['Model-1'].sketches['__profile__'].vertices[2])
mdb.models['Model-1'].sketches['__profile__'].ObliqueDimension(textPoint=(
    -15.9861679077148, -16.0959777832031), value=28.9, vertex1=
    mdb.models['Model-1'].sketches['__profile__'].vertices[0], vertex2=
    mdb.models['Model-1'].sketches['__profile__'].vertices[1])
mdb.models['Model-1'].sketches['__profile__'].ObliqueDimension(textPoint=(
    -10.4787063598633, 83.3585968017578), value=28.9, vertex1=
    mdb.models['Model-1'].sketches['__profile__'].vertices[2], vertex2=
    mdb.models['Model-1'].sketches['__profile__'].vertices[3])
mdb.models['Model-1'].sketches['__profile__'].FilletByRadius(curve1=
    mdb.models['Model-1'].sketches['__profile__'].geometry[2], curve2=
    mdb.models['Model-1'].sketches['__profile__'].geometry[3], nearPoint1=(
    -24.8139724731445, 0.420526504516602), nearPoint2=(-28.7446136474609,
    5.27205848693848), radius=radii)
mdb.models['Model-1'].Part(dimensionality=TWO_D_PLANAR, name='Punch', type=
    ANALYTIC_RIGID_SURFACE)
mdb.models['Model-1'].parts['Punch'].AnalyticRigidSurf2DPlanar(sketch=
    mdb.models['Model-1'].sketches['__profile__'])
del mdb.models['Model-1'].sketches['__profile__']
}

##BC

#### Punch Displacement U2

for Displace in range(-10, -60):
{
mdb.models['Model-1'].rootAssembly.regenerate()
mdb.models['Model-1'].StaticStep(name='Holderforce', previous='Initial')
mdb.models['Model-1'].StaticStep(name='Punchdisplacment', previous=
    'Holderforce')
mdb.models['Model-1'].rootAssembly.ReferencePoint(point=
    mdb.models['Model-1'].rootAssembly.instances['Punch-1'].vertices[3])
```

CASE STUDY 4

```
mdb.models['Model-1'].rootAssembly.ReferencePoint(point=
    mdb.models['Model-1'].rootAssembly.instances['Die-1'].vertices[3])
mdb.models['Model-1'].rootAssembly.ReferencePoint(point=
    mdb.models['Model-1'].rootAssembly.instances['Blankholder-1'].InterestingPoint(
    mdb.models['Model-1'].rootAssembly.instances['Blankholder-1'].edges[2],
    MIDDLE))
mdb.models['Model-1'].rootAssembly.Surface(name='Surf-7', side1Edges=
    mdb.models['Model-1'].rootAssembly.instances['Punch-1'].edges.getSequenceFromMask(
    ('[#f ]', ), ))
mdb.models['Model-1'].RigidBody(name='Constraint-1', refPointRegion=Region(
    referencePoints=(mdb.models['Model-1'].rootAssembly.referencePoints[16], ))
    , surfaceRegion=mdb.models['Model-1'].rootAssembly.surfaces['Surf-7'])
mdb.models['Model-1'].rootAssembly.Surface(name='Surf-8', side1Edges=
    mdb.models['Model-1'].rootAssembly.instances['Blankholder-1'].edges.getSequenceFromMask(
    ('[#f ]', ), ))
mdb.models['Model-1'].RigidBody(name='Constraint-2', refPointRegion=Region(
    referencePoints=(mdb.models['Model-1'].rootAssembly.referencePoints[18], ))
    , surfaceRegion=mdb.models['Model-1'].rootAssembly.surfaces['Surf-8'])
mdb.models['Model-1'].rootAssembly.Surface(name='Surf-9', side1Edges=
    mdb.models['Model-1'].rootAssembly.instances['Die-1'].edges.getSequenceFromMask(
    ('[#1f ]', ), ))
mdb.models['Model-1'].RigidBody(name='Constraint-3', refPointRegion=Region(
    referencePoints=(mdb.models['Model-1'].rootAssembly.referencePoints[17], ))
    , surfaceRegion=mdb.models['Model-1'].rootAssembly.surfaces['Surf-9'])
mdb.models['Model-1'].rootAssembly.Set(name='Set-4', referencePoints=(
    mdb.models['Model-1'].rootAssembly.referencePoints[16], ))
mdb.models['Model-1'].DisplacementBC(amplitude=UNSET, createStepName='Initial',
    distributionType=UNIFORM, fieldName='', localCsys=None, name='BC-1',
    region=mdb.models['Model-1'].rootAssembly.sets['Set-4'], u1=SET, u2=SET,
    ur3=SET)
mdb.models['Model-1'].rootAssembly.Set(name='Set-5', referencePoints=(
    mdb.models['Model-1'].rootAssembly.referencePoints[18], ))
mdb.models['Model-1'].DisplacementBC(amplitude=UNSET, createStepName='Initial',
    distributionType=UNIFORM, fieldName='', localCsys=None, name='BC-2',
    region=mdb.models['Model-1'].rootAssembly.sets['Set-5'], u1=SET, u2=SET,
    ur3=SET)
mdb.models['Model-1'].rootAssembly.Set(name='Set-6', referencePoints=(
    mdb.models['Model-1'].rootAssembly.referencePoints[17], ))
mdb.models['Model-1'].DisplacementBC(amplitude=UNSET, createStepName='Initial',
    distributionType=UNIFORM, fieldName='', localCsys=None, name='BC-3',
    region=mdb.models['Model-1'].rootAssembly.sets['Set-6'], u1=SET, u2=SET,
    ur3=SET)
mdb.models['Model-1'].rootAssembly.Set(name='Set-7', vertices=
    mdb.models['Model-1'].rootAssembly.instances['sheet-1'].vertices.getSequenceFromMask(
    ('[#2 ]', ), ))
mdb.models['Model-1'].DisplacementBC(amplitude=UNSET, createStepName='Initial',
    distributionType=UNIFORM, fieldName='', localCsys=None, name='BC-4',
    region=mdb.models['Model-1'].rootAssembly.sets['Set-7'], u1=UNSET, u2=SET,
    ur3=UNSET)
mdb.models['Model-1'].rootAssembly.Set(name='Set-8', vertices=
    mdb.models['Model-1'].rootAssembly.instances['sheet-1'].vertices.getSequenceFromMask(
```

CASE STUDY 4

```
    ('[#1 ]', ), ))
mdb.models['Model-1'].XsymmBC(createStepName='Initial', localCsys=None, name=
    'BC-5', region=mdb.models['Model-1'].rootAssembly.sets['Set-8'])
mdb.models['Model-1'].rootAssembly.translate(instanceList=('Die-1', ), vector=(
    0.0, 0.55, 0.0))
mdb.models['Model-1'].rootAssembly.translate(instanceList=('Blankholder-1', ),
    vector=(0.0, -0.55, 0.0))
mdb.models['Model-1'].rootAssembly.translate(instanceList=('Punch-1', ),
    vector=(0.0, 0.55, 0.0))
# Save by mastergr2 on 2022_12_09-17.06.04; build 2022 2021_09_15-19.57.30 176069
# Save by mastergr2 on 2022_12_09-17.06.40; build 2022 2021_09_15-19.57.30 176069
}

for Punch_load in range(-90000.0, -190000.0):
{
mdb.models['Model-1'].steps['Holderforce'].setValues(initialInc=0.1, maxNumInc=
    1000)
mdb.models['Model-1'].steps['Punchdisplacment'].setValues(initialInc=0.1,
    maxInc=0.1, maxNumInc=1000)
mdb.models['Model-1'].boundaryConditions['BC-1'].setValuesInStep(stepName=
    'Holderforce', u2=FREED)
mdb.models['Model-1'].boundaryConditions['BC-1'].setValuesInStep(stepName=
    'Holderforce', u2=0.0)
mdb.models['Model-1'].boundaryConditions['BC-1'].setValuesInStep(stepName=
    'Punchdisplacment', u2=-60.0)
mdb.models['Model-1'].boundaryConditions['BC-2'].setValuesInStep(stepName=
    'Holderforce', u2=FREED)
mdb.models['Model-1'].rootAssembly.Set(name='Set-9', referencePoints=(
    mdb.models['Model-1'].rootAssembly.referencePoints[18], ))
mdb.models['Model-1'].ConcentratedForce(cf2=Punch_displacement, createStepName=
    'Punchdisplacment', distributionType=UNIFORM, field='', localCsys=None,
    name='Load-1', region=mdb.models['Model-1'].rootAssembly.sets['Set-9'])
mdb.Job(atTime=None, contactPrint=OFF, description='', echoPrint=OFF,
    explicitPrecision=SINGLE, getMemoryFromAnalysis=True, historyPrint=OFF,
    memory=90, memoryUnits=PERCENTAGE, model='Model-1', modelPrint=OFF,
    multiprocessingMode=DEFAULT, name='1p1', nodalOutputPrecision=SINGLE,
    numCpus=1, numGPUs=0, numThreadsPerMpiProcess=1, queue=None, resultsFormat=
    ODB, scratch='', type=ANALYSIS, userSubroutine='', waitHours=0,
    waitMinutes=0)
mdb.jobs['1p1'].submit(consistencyChecking=OFF)
}
# Save by mastergr2 on 2022_12_09-17.10.30; build 2022 2021_09_15-19.57.30 176069
```

1. Punch Radius

We apply a for loop to change radius for the code and giving a variable part and assigning a loop to change values for each time.

Radius of fillet changes from 3 to 10 mm and the results have been plotted.

CASE STUDY 4

2. Block Holder Force

We apply a for loop to change radius for the code and giving a variable part and assigning a loop to change values for each time.

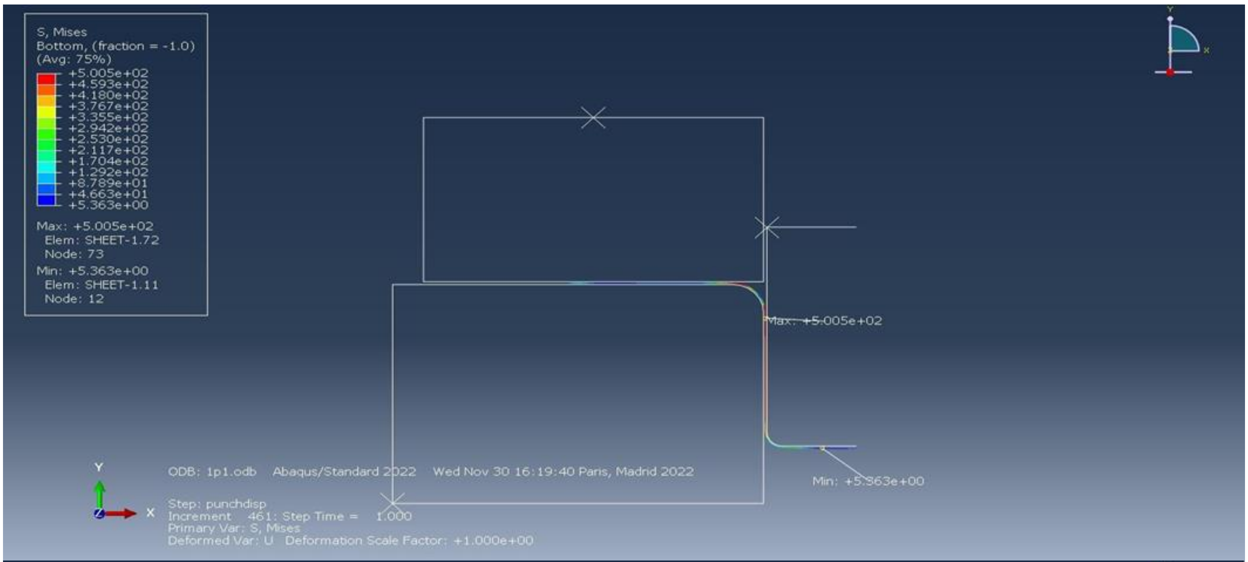
Radius of fillet changes from 3 to 10 mm and the results have been plotted.

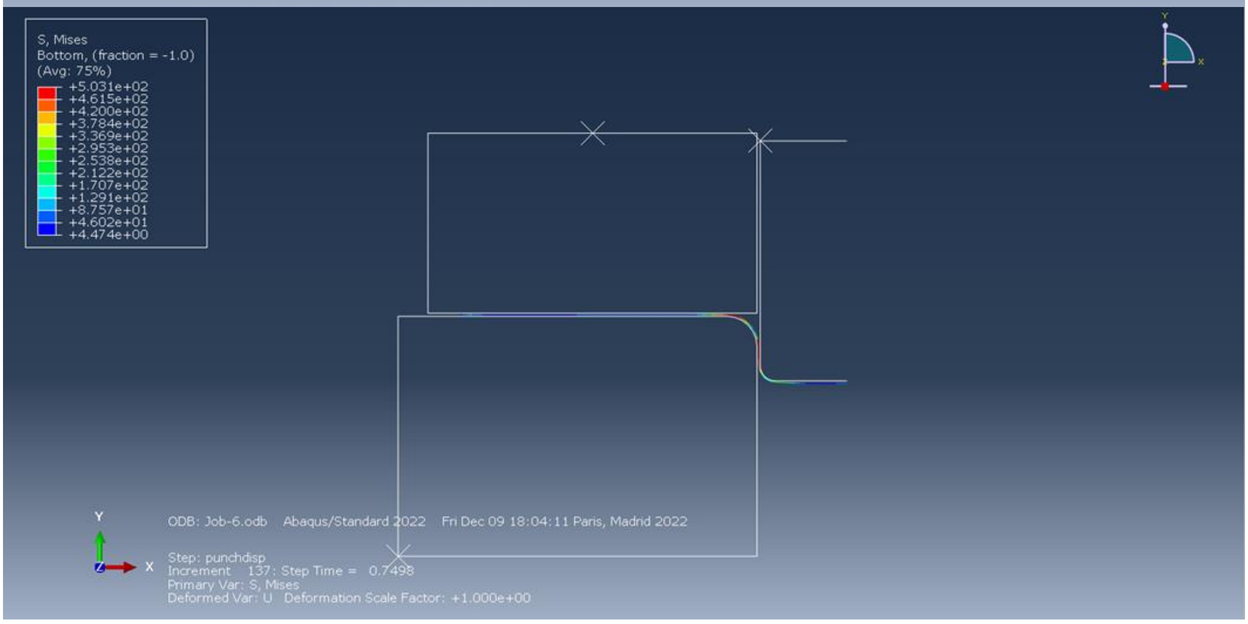
3. Punch Displacement

We apply a for loop to change radius for the code and giving a variable part and assigning a loop to change values for each time.

Radius of fillet changes from 3 to 10 mm and the results have been plotted.

For displacement change the values are changed and results have been displacement in images below.





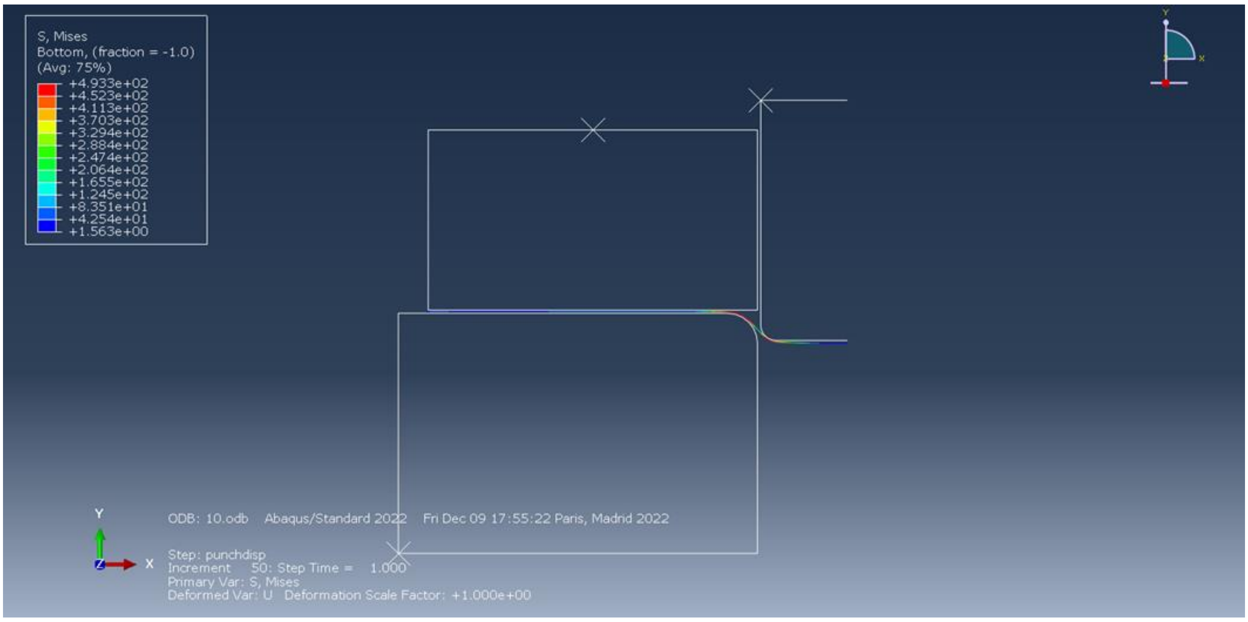


Fig. 17 Images showing Von_Mises stress for displacement 10, 30 and 60mm respectively

6. CONCLUSION

This simulation shows the variation of force increases with respect to thickness of the plate and also when time increase that is from initial position to the position where stamping is made, force is increased and the force comes back to original state. Also, the stress is max at place where the plate bends and maximum plastic strain where the end of the stamping attached to plate to leave some energy to the material as plasticity. Thickness place a very importat role while defining the stamping process. Hence also considering a beam element is also more sensible option to opt for good results. And also considering symmetric option reduces time and also similar results as entire part.

Using the code automation attempt has been made to change in radius, block force and change in displacement.