In recent years, the demand for high-durability parts are rising too much. These challenges are difficult to overcome without an innovative framework based on an accurate database. The problems of high stress generated due to the hard friction and severe crystal dislocation during the forming process need to be solved. High frictional forces between the contact surfaces while forming lead to high sticking between the parts. In this work, forming process of the large sheet metal has been explored based on some parameters like material properties, stress generation, and their effects on the product quality. For this purpose, square sheet metal of 721*721*5 mm is considered, and the product formation through many forming steps was carried out. This work includes adopting some design steps, modeling, and analysis to control some parameters and minimize the generation stresses. The finite element software (ABAQUS/CAE) has been adopted for analyzing this process. In this simulation, the forming process evolution in different steps has been analyzed, and the influence of the effective parameters was performed. As a result, it's found that the generation stresses are highly concentrated near the fillet zones and proportional to the pressure, and depend on the nature of contact and friction. Simulation results also revealed that the uniform pressure during forming will leads to minimizing the friction and stress generation (5 %) and this will improve the product quality. Also, it's possible to identify and facilitate many difficulties and evaluate the possibilities before further investing in tooling. It's concluded that any accurate process like this must depend on some sequence steps like design, modeling, and simulation. Moreover, folding the large surface area needs accurate and adjustable types of equipment

Keywords: sheet metal forming, square cup, stress concentration, analysis process, ABAQUS simulation

UDC 621
DOI: 10.15587/1729-4061.2022.263452

IMPLEMENTATION OF FINITE ELEMENT ANALYSIS FOR SOLVING THE CONSTRAINTS IN FORMING PROCESS OF LARGE STEEL PARTS

Kamil Jawad Kadhim
Corresponding author
Assistant Professor*
E-mail: inm.kml@atu.edu.iq
Jabbar A. Jaber
PhD, Professor*
Hadi Raheem Ibrihim
PhD, Lecturer*
*Department of Mechanical
Al-Mussaib Technical Institute – Babylon
Al-Furat Al-Awsat Technical University
Najaf Main str., 6, Kofa, Iraq, 54003

Received date 21.06.2022
Accepted date 16.08.2022
Published date 31.08.2022

How to Cite: *Kadhim, K. J., Jaber, J. A., Ibrihim, H. R. (2022). Implementation of finite element analysis for solving the constraints in forming process of large steel parts. Eastern-European Journal of Enterprise Technologies, 4 (1 (118)), 64–71. doi: https://doi.org/10.15587/1729-4061.2022.263452*

1. Introduction

Products with a large surface area have many constraints and limitation problems during forming the required profile. It is easy to observe these defects in many industrial applications, like automobile parts, and domestic and laboratory equipment. According to many studies in similar cases, it found that the problem is not concerned with forming process itself, but it concerns with the stress concentration behavior. Therefore, research on the development of understanding of this process is relevant to some important factors such as total energy consumed with time, strain hardening, and stress concentration.

It is always advised to apply the finite element method for this approach to enhance the result accuracy. The modeling and simulation process is normally used to avoid many problems before the occurrence, and before tooling investment. The contact problems between the parts during assembly can easily be avoided, and the process parameters such as pressure, forces, and stresses will be estimated. The task of modeling a large product with a large number of meshing elements is a difficult task, and this will leads to a significant increase in calculation costs accordingly.

Therefore, studies that are devoted to this direction are dealing with the problem under consideration to estimate better results and conclusions. The physical properties and behavior of forming metal inside the die during the process are not easy to predict and always depend on trial and errors solution. However, this will needs a large number of iterations, which increase the cost rapidly.

2. Literature review and problem statement

The forming process is always used in producing finished and semi-finished parts with different shapes and geometries. One of the important methods to reduce the production time and cost costs in sheet metal forming is optimizing the main parameters of this process. Paper [1] study the methods of optimizing sheet metal forming from different views and estimating that the distortion behavior of the sheet metal is the first step to finding the optimal values for these parameters. In work [2], the simulation and analysis are implemented for reducing some defects such as tears, bad formation, and wrinkles. This economic process is significant, and simulation results can offer the best understanding of the influencing parameters on the product properties, and it can be a substitute for intense experiments.

The paper [3] provided an innovative of sheet-bulk metal forming (SBMF) process. It found that the hardness distribution and homogeneity of the forming parts are improved. Furthermore, it's also found that the failures can be prevented and the volume filling can significantly increase by up to 97 %. This forming method is performed depending on numerical feasibility. It's a robust method to estimate alternate bending and compressive stresses. The paper [4] works on reducing spring back, economical realization, and press loads are estimated that the numerical analysis of some complex geometry is considered efficient compensation for some expensive experiments.

The challenges of high accuracy and mass production parts with good mechanical properties and fulfilling environmental requirements need to adopt a new trajectory in the recent industry. These requirements of the above challenges need to concentrate the concerns on the development of the metal forming sector through adopting an accurate procedure [5]. In work [6], the finite element method as computational modeling is used in each forming stage of analysis of the progressive deformation to make the design efficient, and faster, and decrease the trial and error time. However, there are many upsetting problems related to the high surface area need to be solved.

In a process like forming, the thickness of the blank metal will remain constant without any change, since there's no extra increase in the surface area, [7]. Sequential simulation is used in the manufacturing process to enhance the prediction capabilities and validation purposes through physically comparing with experimental findings. Virtual prototyping is used to predict the output findings of the overall forming process [8]. Some numerical analyses used adaptive meshing to enable tracking and calculating of the large qualitative comparison and plastic deformations. These advantages help the researchers and manufacturing centers to emphasize their findings and prediction with less error percentage [9].

As the complexity of the part increases, the study of the influencing parameters becomes necessary. Determining and estimating some effects parameters like reduction ratio, effective stress, and punch force by FEM in a forming process before tooling investment is very essential.

In work [10], the developed ANN used with FEM simulation used in forming process to minimize the process development effort in terms of time and cost. In paper [11], the finite element is adopted to analyze and estimate the main factors, which particularly affect pressure and stresses distribution. Work [12] found that there are several important keys that influence the manufacturing process such as the risk of material failure, shape fulfillments, and variation. Resources like computational and computer-aided engineering is an efficient and useful tool since it allows rapid computation, prediction, and evaluation of the capabilities of both qualities and quantities process trajectory [13]. During the forming of the large sheet metal, and due to local plastic instability of the sheet material, partial wrinkling will occur from the in-plane to out-of-plane direction. To constrain the wrinkling in the box-shaped part it is the device to apply enough constraints on the sheet metal to avoid this problem [14].

While many works of literature went far in explaining of forming process, they avoid or do not give more interest to analyzing the forming of large surface parts in detail. Some parameters were achieved in several works of literature, but others do not. However, this analysis involves many constrain due to high interaction and the type of boundary conditions. Consequently; conducting this study to build the model is an important target in this field.

This type of forming requires a high accuracy design, because the nodal and elements displacement during the crystal dislocations over all the surface may be not uniform, and this will cause a high deformation rate. The importance of this modeling and simulation is to explore how to attain better mechanical properties of the forming parts.

3. The aim and objectives of the study

The aim of the study is to determine of problems and effects of the generation stresses, and pressure distribution during this severe forming process, and to explore the weak points resulting from the crystal dislocations over all the surface by building a finite element model for a cold-forming process for a large steel part.

The simulation findings could be used to determine and investigate the interaction between some different parameters during this dynamic process which can help the manufacturers to avoid and overcome many fabrication errors.

To achieve this aim, the following objectives are accomplished:

– to estimate and clarify the problems and effects of the generation stresses, and pressure distribution during this severe forming process, and predict the weak points before building the actual tools;

– to explore the general solutions for avoiding the problems of product sticking during ejection and removing step due to the high strain hardening occurrence in this type of big contact surface area.

4. Materials and methods of research

The starting point in this forming process is to define product and workpiece geometry with the other tools for the modeling procedure. In closed-die forming; it's very important to define all the starting geometrical properties like dimensions and material specifications. Fig. 1, 2 below illustrate the (2D), and (3D) geometry for the product understudy in this forming process.

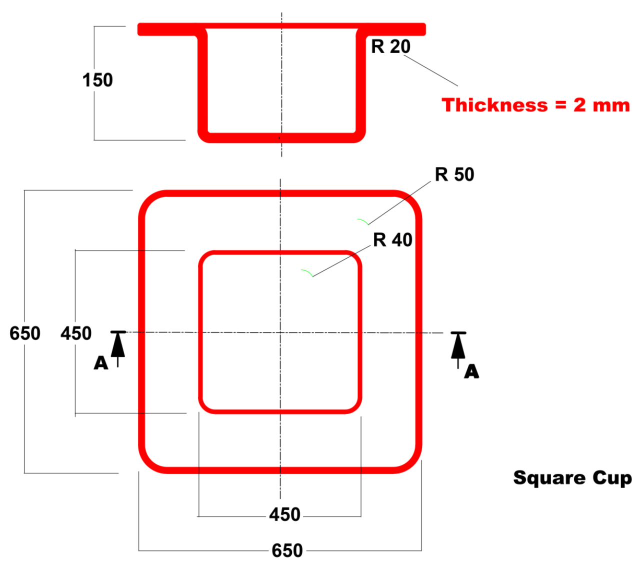


Fig. 1. The product in two dimensions

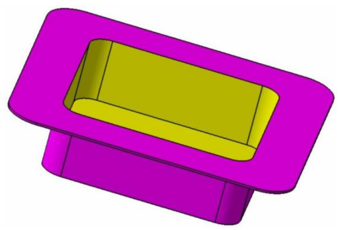


Fig. 2. The product in three dimensions

The blank sheet dimensions always must be calculated before the forming process. Formula (1) is used to calculate the dimensions of part development before forming depending on the final product dimension shown in Fig. 3:

$$L=\sqrt{d_1^2+4hd_1+0.5(d_2-d_1)^2}. \quad (1)$$

For $d_1=450$ mm, $d_2=650$ mm, $h=150$ mm (Fig. 3) $L=701$ mm.

And by adding the length values of four fillets, the final dimension will be 721 mm.

Consequently, the final dimensions of the blank sheet will be as shown in Fig. 4.

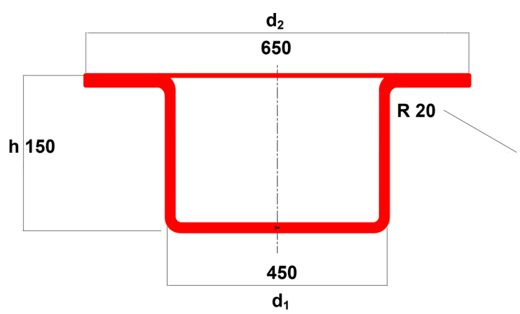


Fig. 3. Final product

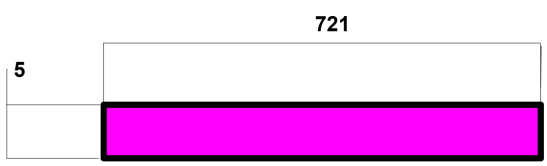


Fig. 4. The blank sheet dimensions

The operation sequence of this forming process is consists of three main steps. The first step is to align and justified the blank metal on the blank holder as shown in Fig. 5.

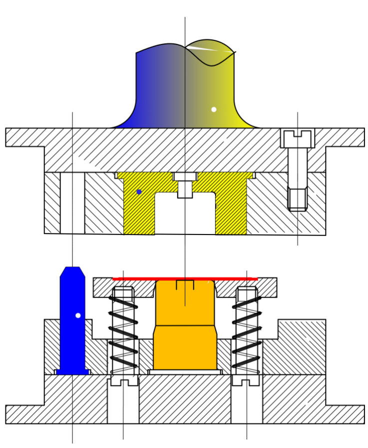


Fig. 5. Alignment of blank metal in the first step

The second step is considered a main forming step which includes pressing the sheet metal inside the die cavity to form the final shape according to the specific dimensions as shown in Fig. 6. High pressure will be required to push the metal inside the cavity due to high friction and heat generation during this process.

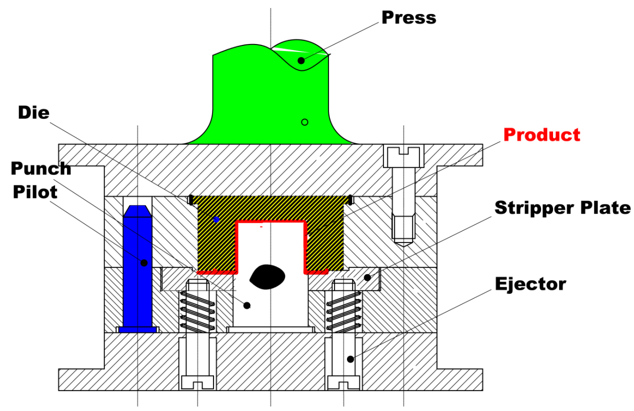


Fig. 6. Forming process

The final step in this forming process is the product ejection and removal outside the cavity. Based on the maximum shear theory; there is a huge amount of stress will be generated due to the crystal dislocation during this process.

The final step in this forming process is the product ejection and removal outside the cavity. Based on the maximum shear theory; there is a huge amount of stress will be generated due to the crystal dislocation during this process.

Also, high sticking forces between the product and punch need to be overcome during ejection to keep the part without deformation. Fig. 7 above illustrates the Part ejection during the final step.

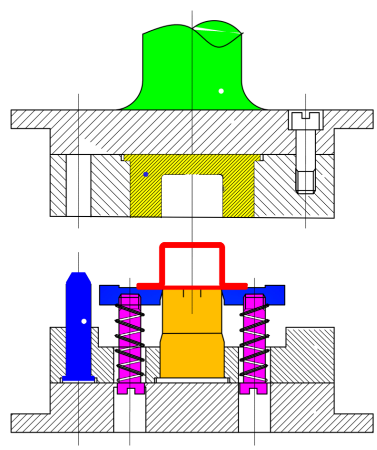


Fig. 7. Part ejection during the final step

The modeling technique of this deformation process is the best method for better understanding before any further manufacturing step. It always helps to avoid much error and decrease both cost and consumed time. The current model under study are consisting of some important parts like deformable sheet metal, rigid punch, rigid die, and stripper plate as shown in Fig. 8, *a*. Parts modeling is an essential process that helps to avoid and prevent any interference during the overall parts assembly. Furthermore, the following parts of the analysis and simulation process are required to convert and save this model in a format that can export to other analysis software. At this point, (CATIA/R16) software has been implemented for this purpose.

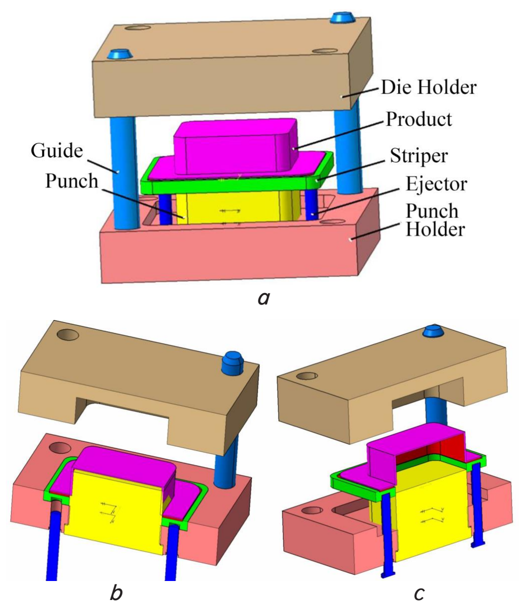


Fig. 8. Modeling the main parts:
a — during the forming step; *b* — during opening;
c — during ejection and removing the part

Fig. 8, *b* illustrates a model in the (3D) sectional process of the tools used during the forming process. Also, Fig. 8, *c* illustrates a model in the (3D) sectional process of the tools used during the process of ejection and removing the part. The properties of steel material used as sheet metal are illustrated and listed in Table 1 below.

For the purpose of carrying out a simulation and analysis of this deformation process; the Commercial software ABAQUS/(CAE) has been adopted. The main active parts in this process are sheet metal, which considers as a deformable part, and modeled as a Von-Mises plastic elastic material. The other two parts are punch and die which are both considered as rigid bodies that cannot deform as shown in Fig. 9.

Table 1

Mechanical properties of steel material

No.	Properties	Values
1	Yield Stress	378 MPa
2	Young's Modulus of Elasticity	196 GPa
3	Poisson's Ratio	0.33
4	Density	7340 kg/m^3
5	Expansion Coeff.	0.000120

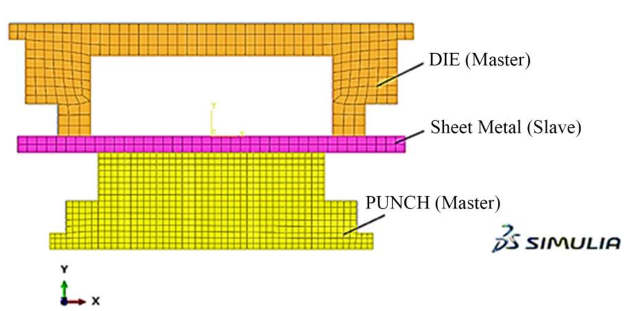


Fig. 9. The main active parts in the simulation process

The analysis process are consisting of some sequence steps. These sequence steps are starting from part creation, material assignment, interactions, applying load according to the boundary conditions, and submission for the analysis as shown in Fig. 10.

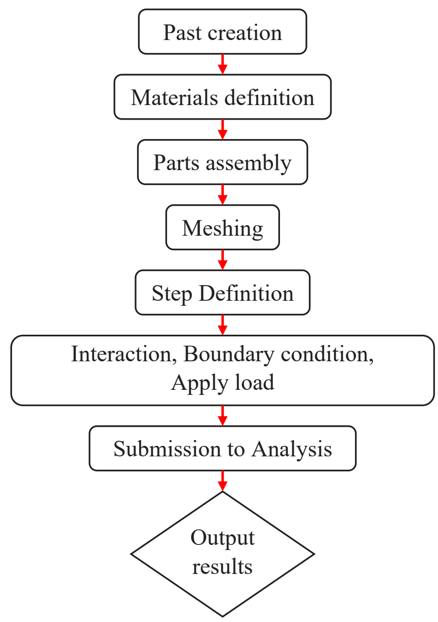


Fig. 10. Flowchart of Analysis process

The shaping Parameters of this deformation are strain rate and friction. The friction between the die and metal has a large extent influence on deformation. The friction coefficient is chosen to be 0.12.

This Multi-point forming process uses separated ideas to substitute the three-dimensional surface with the two-dimensional surface with highly adjustable which is fast, and reconfigurable.

In this process, the analysis is starting with placing and aligning the sheet metal in the exact position in order to ensure the pressure distribution uniformly.

The first simulation step is starting by applying the pressure on the die to move down toward the sheet metal till to be in touch with each other.

The boundary condition in this step includes constraining the punch from all directions to prevent any movement while the sheet metal is under forming to reach the final shape. Also, it includes applying a pressure load (1000 MPa) and concentrated force in a downward direction. Finally, and in the last step, the contact will release and ramp down automatically. Fig. 11 illustrates the steps of applying the load and boundary condition.

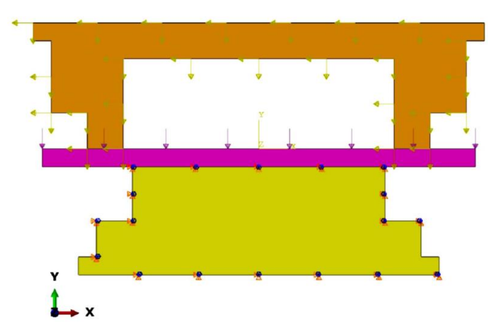


Fig. 11. Illustration of the steps of applying the load and boundary condition

The boundary conditions include applying constrain along the punch surface to prevent it from any movement, while the deformable part (Blank sheet) is without any constrain as shown in Fig. 12.

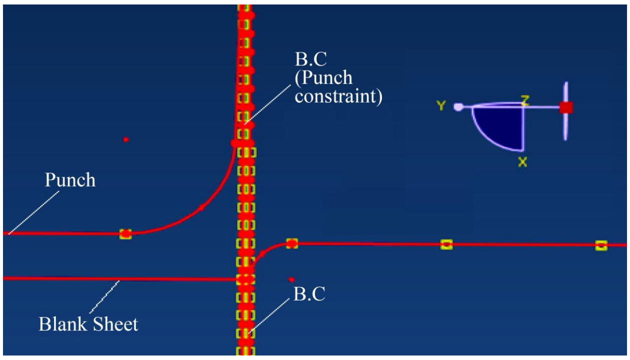


Fig. 12. The applied boundary conditions

Mesh refinement is very important in this simulation for accurate results implementation. Mesh control with (Quad-dominated) and medial axis mesh are used to refine this mesh. Standard element types (reduced integration) with user define are used in mesh control as shown in the foregoing Fig. 9.

5. Results of analysis and simulation of the forming process

5. 1. Stresses generation and distribution during the forming process

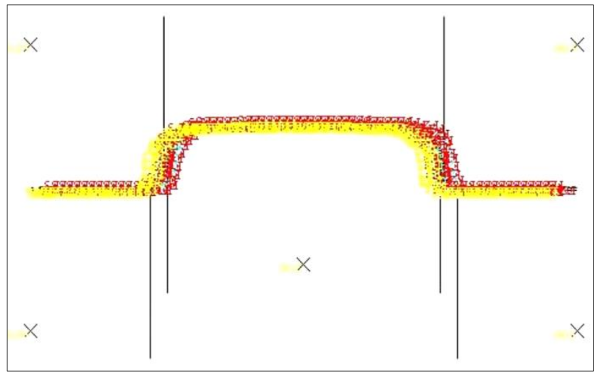
While this forming process includes many steps, each individual step has different interactions, load, and boundary conditions, hence the contact surfaces must define accurately according to the function of each step.

At this point, when the punch will be in contact with the sheet metal in the starting, there is a little deformation and the strain hardening is very small in this step. Subsequently, while the forming is in progress, and the punch pushes the sheet metal inside the die, there will be a high strain rate and stress generation due to severe dislocation in the material surface till the punch reach the final travel. The forming steps of the part are shown in Fig. 13, *a–c* below.

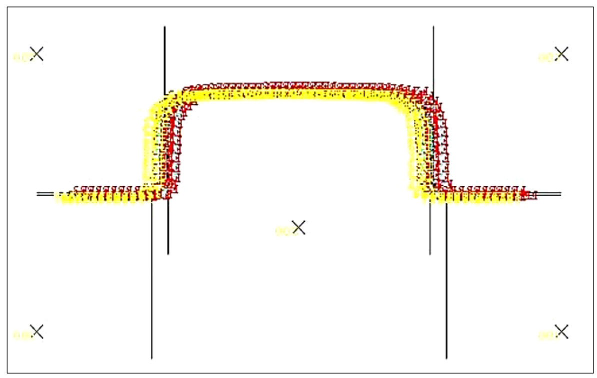
In the final incremental step, the displacement of all elements will stop, and all contact surfaces will be removed to prepare the product for the removal and ejection step. The final forming step is shown in Fig. 14.

The nature of deformation between the contact surfaces is nonlinear with elastic-plastic, so the high energy gained by nodal elements of the forming sheet metal will move downward due to the severe friction forces. Plastic strain will develop gradually to the maximum value in the last deformation step. The progress of deformation which starting when the punch starts moving gradually toward the workpiece and develop with increasing load till reaches the final limit. Finally, the applied load will be released and leaving the forming part with some of the spring back. Fig. 15, *a–c* illustrates the progress of the punch traveling toward the workpiece till the final limit.

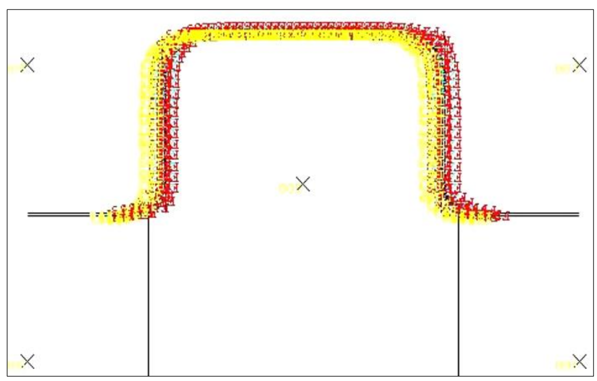
The contour plot of the final forming parts shows that the generation stresses distribution will be at the maximum values at the end of the center zone of the part, while it will be at the minimum value at the starting zone because the deformed nodes and elements will pass the elastic limit at this zone as shown in Fig. 16.



a



b



c

Fig. 13. Forming steps: *a* — First forming steps; *b* — Second forming step; *c* — Third forming step

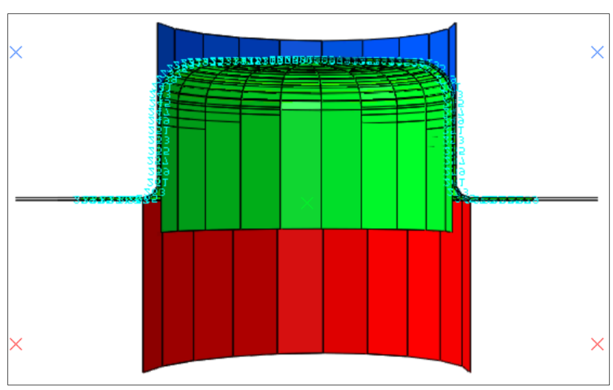


Fig. 14. The final forming step

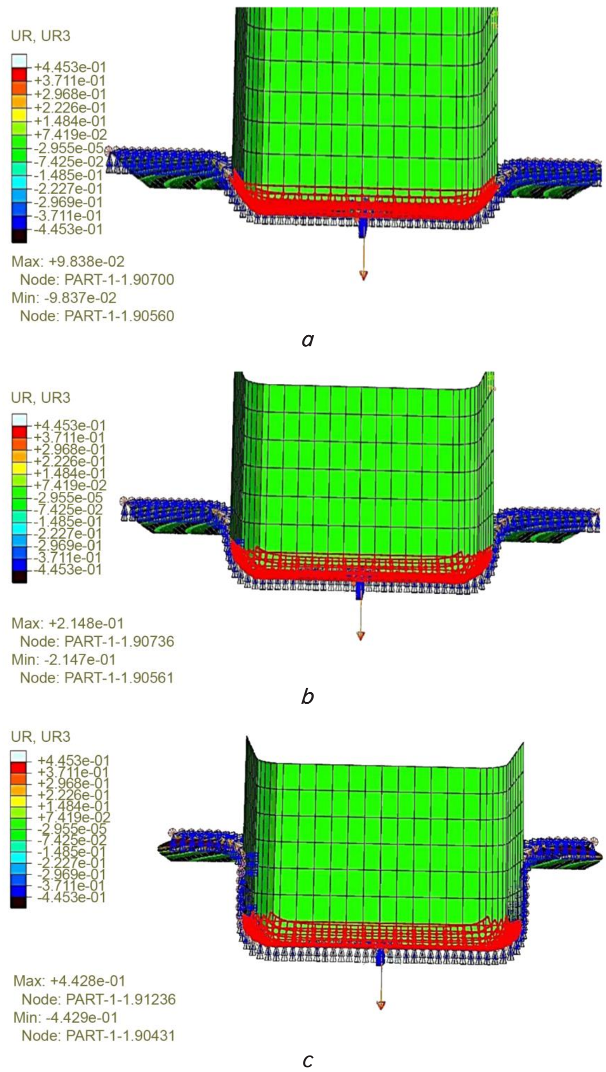


Fig. 15. Punch traveling: *a* — initial movement; *b* — second stage movement; *c* — the final displacement

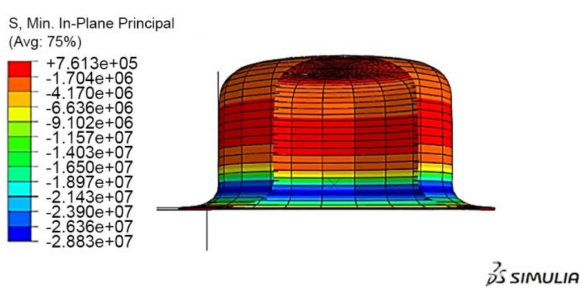


Fig. 16. The contour plot of stress distribution

However, the values of these stresses will little drop down after part removal due to material spring back property.

5. 2. Strain hardening and energy consumed during removal stage

The elastic strain of the deformed elements will be at maximum values at the starting of the process due to the nature of deformed metal which absorbs the first shock during the starting contact as shown in Fig. 17.

The major finding of this study was the significant relationship between strain and stress. During the part forming, the elements of sheet metal in touch with punch gain a high speed, and the strain is very small and moves down fast, but with process development, the strain will increase rapidly and the stress generation will reach the maximum values as shown in Fig. 18.

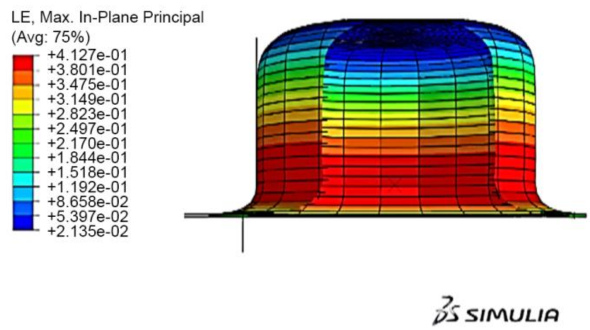


Fig. 17. The contour plot of elastic strain

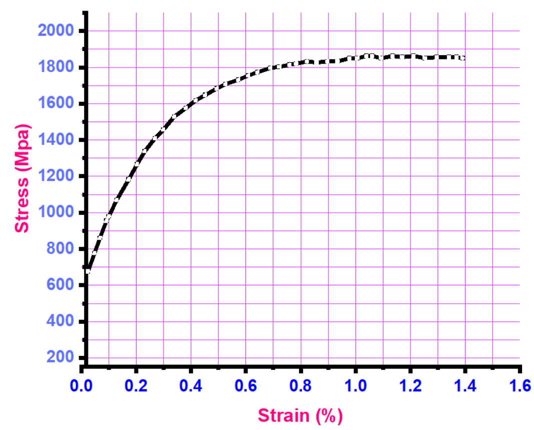


Fig. 18. Stress-Strain relationship

The other important result is regarding with relationship between the consumed energy with forming time.

The analysis plot from the output database visualization shown in Fig. 19 reflects the relationship between the consumed energy with time during the forming process.

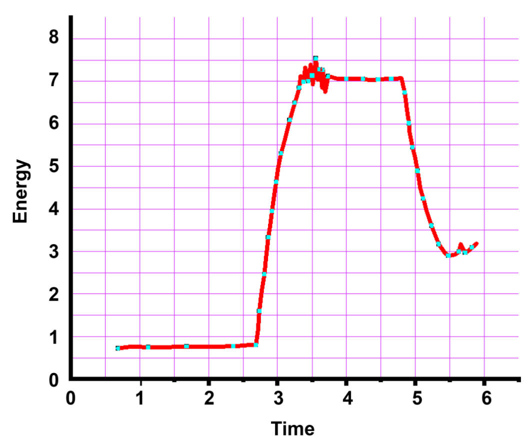


Fig. 19. Energy-Time relationship

The consumed energy by the overall pressing system will reach the maximum before exceeding the mid-way forming limit.

For comparison purposes, the results obtained by [3] approved that the hardness homogeneity distribution can be improved and the process failures can be prevented in a forming process (Functional Analysis). These findings are matching with the results obtained here. Also, the results from [14] (the multi-point forming method can effectively restrain the wrinkle of the sheet) led to a high indication of sobriety for this Information validity.

6. Discussion of results of analysis and simulation for forming process

The results obtained in this proposed method are intended to show the weak points in this forming type and focus on stresses and pressure distribution in order to suggest a homogeneous distribution by comparing with some of the existing research.

We have considered reducing all types of intensive loads along the contact surfaces to avoid product distortion.

To find out a suitable result, the findings of previous research have been stated for evaluation purposes. The forming process under study are consist of two materials with different mechanical and physical properties. These materials are defined as well and assigned to their sections based on specific criteria used in the software.

Due to the sliding interaction behavior between the parts (master and slave) as mentioned in Fig. 9, and the nature of constraints as in the boundary condition in Fig. 11 and Fig. 12 the displacement of the stochastic elements (deformation) can be eliminated to the minimum. Consequently, the high pressure and highly stressed zones overall the surface area is concentrated with their maximum values in the lower zone of the product as illustrated in Fig. 16.

The mesh method used in seeding and meshing the master parts (non-deformable parts) as in Fig. 9, and associated this mesh with relative geometries will ensure a uniform elements displacement during the last forming step. This will ensure a little variation in the cycle time of product ejection.

Consequently; the pressure and stress distribution on the contact surfaces and the relationship between these parameters has been determined. The strain of the forming profile is getting growth very slow in the beginning due to high dislocation and high-stress value in this stage as shown in Fig. 18.

Based on the results of this constructed simulation; the influence of stresses has been determined under different types of boundaries by considering that all rigid parts (non-deformable parts) as masters and the forming parts (deformable parts) are slaves. The deformation process with time will pass the critical stage after exceeding the mid-displacement of forming due to the stability and uniform stretch as shown in Fig. 19.

It's important to reveal that there are some limitations regarding the nature of the large surface area and the percentage of spring back. The nonlinearity and material orientation will impose some restrictions on calculating the parameters accurately.

The main limitations of this study are involving some boundary conditions related to some necessary restrictions of parts due to large forming areas. The disadvantage of handling this analysis method is the need for a high amount of data which always costs a long time due to running and repeating the simulation many times to find accurate results. Failure in determining the exact formulas and boundary conditions can affect the results significantly.

The development of such research types is related to overcoming and eliminating some of the shortcomings and restrictions which lead to raising the concentration and values of stresses by using suitable boundary conditions and interaction types between the contact parts. To eliminate the generation stresses which result in high and non-homogeneous pressure distribution, it's recommended to partition the simulated part into different zones to increase the element density which leads to the best accuracy.

7. Conclusions

1. This contribution presented and evaluated the forming manufacturing process through different approaches, and carried out according to the relevant geometrical properties of the part. Due to the large forming area, which leads to the generation of tangential compression stresses alongside the contact surfaces, it was found that there is some variation in the stress distribution around some fillets and corner zones. According to the contour plot of generated stresses, the percentage of this variation is approximately 3 %, and This percentage considers being acceptable variation due to the large surface area and cooling difference.

2. In this manufacturing process, load distribution should be uniform over all the surface areas to avoid sticking between the tools and product. Also, it's necessary to prevent any warpage in products during both forming and removing (ejecting) time. Some reliable and suggested solutions for preventing or minimizing the strain hardening occurrence are raised up which involve adjusting the value of the consumed and generated energy with time to control this factor. At this point, the required energy to overcome and release the product during the strain hardening stage and before spring back occurrence will increase rapidly due to high surface sticking.

It is important to adjust the applied load according to the stress-strain relation to ensure it is 5 % less than the ultimate value.

The presented setup can be expanded to predict and investigate the optimum forming parameters and used to evaluate some part designs with different material properties.

Conflict of interest

The authors declare that they have no conflict of interest in relation to this research, whether financial, personal, authorship or otherwise, that could affect the research and its results presented in this paper.

References

1. Kumar, S., Shejkar, S. K. (2021). A Review on Optimizing Sheet Metal Forming. International Journal of Engineering Research in Current Trends (IJERCT), 3 (4), 62-64. URL: https://www.ijerct.com/papers/03-04/a-review-on-optimizing-sheet-metal-forming.pdf
2. Baru, N. K., Teeuwen, T., Teller, M., Hojda, S., Braun, A., Hirt, G. (2021). On appropriate Finite Element discretization in simulation of gas-based hot sheet metal forming processes. IOP Conference Series: Materials Science and Engineering, 1157 (1), 012027. doi: https://doi.org/10.1088/1757-899x/1157/1/012027

3. Hetzel, A., Schulte, R., Vogel, M., Lechner, M., Besserer, H.-B., Maier, H. J. et. al. (2021). Functional Analysis of Components Manufactured by a Sheet-Bulk Metal Forming Process. Journal of Manufacturing and Materials Processing, 5 (2), 49. doi: https://doi.org/10.3390/jmmp5020049
4. Briesenick, D., Liewald, M., Riedmueller, K. R. (2021). New sheet metal forming process for springback reduction by continuous stress superposition. IOP Conference Series: Materials Science and Engineering, 1157(1), 012030. doi: https://doi.org/10.1088/1757-899x/1157/1/012030
5. Centeno, G., Silva, M. B. (2022). Tube and Sheet Metal Forming Processes and Applications. Metals, 12 (4), 553. doi: https://doi.org/10.3390/met12040553
6. Roque, C. M. O. L., Button, S. T. (2000). Application of the finite element method in cold forging processes. Journal of the Brazilian Society of Mechanical Sciences, 22 (2), 189–202. doi: https://doi.org/10.1590/s0100-73862000000200005
7. Chapter 8: Sheet metal forming processes. KSU - College of Engineering. Available at: https://faculty.ksu.edu.sa/sites/default/files/chapter_8_ie252-v2.pdf
8. Govik, A., Nilsson, L., Moshfegh, R. (2012). Finite element simulation of the manufacturing process chain of a sheet metal assembly. Journal of Materials Processing Technology, 212 (7), 1453–1462. doi: https://doi.org/10.1016/j.jmatprotec.2012.02.012
9. Cherouat, A., Borouchaki, H., Jie, Z. (2018). Simulation of Sheet Metal Forming Processes Using a Fully Rheological-Damage Constitutive Model Coupling and a Specific 3D Remeshing Method. Metals, 8 (12), 991. doi: https://doi.org/10.3390/met8120991
10. Hrudkina, N., Aliieva, L., Markov, O., Marchenko, I., Shapoval, A., Abhari, P., Kordenko, M. (2020). Predicting the shape formation of hollow parts with a flange in the process of combined radial-reverse extrusion. Eastern-European Journal of Enterprise Technologies, 4 (1 (106)), 55–62. doi: https://doi.org/10.15587/1729-4061.2020.203988
11. Maiorova, K., Vorobiov, I., Andrieiev, O., Lupkin, B., Sikulskiy, V. (2022). Forming the geometric accuracy and roughness of holes when drilling aircraft structures made from polymeric composite materials. Eastern-European Journal of Enterprise Technologies, 2 (1 (116)), 71–80. doi: https://doi.org/10.15587/1729-4061.2022.254555
12. Korzhyk, V., Khaskin, V., Grynyuk, A., Peleshenko, S., Kvasnytskyi, V., Fialko, N. et. al. (2022). Comparison of the features of the formation of joints of aluminum alloy 7075 (Al-Zn-Mg-Cu) by laser, microplasma, and laser-microplasma welding. Eastern-European Journal of Enterprise Technologies, 1 (12 (115)), 38–47. doi: https://doi.org/10.15587/1729-4061.2022.253378
13. Magid, H. M., Dabis, B. K., Abed alabas Siba, M. (2021). Analysis of the main factors affecting mass production in the plastic molding process by using the finite element method. Eastern-European Journal of Enterprise Technologies, 6 (1 (114)), 65–71. doi: https://doi.org/10.15587/1729-4061.2021.248375
14. Lin, J., Li, M., Qu, E., Liu, W. (2018). Research on the Process of Flexible Blank-holder in Multipoint Forming for Box-shaped Parts. IOP Conference Series: Materials Science and Engineering, 439, 042024. doi: https://doi.org/10.1088/1757-899x/439/4/042024