International Journal of Enhanced Research in Science, Technology & Engineering
ISSN: 2319-7463, Vol. 11 Issue 9, September-2022, Impact Factor: 7.957

A Study on Finite Element Method General Operating Process: A Review

Pardeep Kumar[1], Vineet Kumar[2]

[1]M.Tech Research Scholar , Mechanical Engineering Department, University Institute of Engineering & Technology, Maharshi Dayanand University, Rohtak, Haryana, India
[2] Professor, Mechanical Engineering Department, University Institute of Engineering & Technology, Maharshi Dayanand University, Rohtak, Haryana, India

ABSTRACT

Producing sheet metal has advanced to a point where it is now commonly employed. In the highly competitive manufacturing industry, reducing response time and costs, boosting efficiency, and enhancing the quality of things manufactured are essential for continued existence. Using finite element analysis as a simulation method, we can evaluate the performance of various components, equipment, and structures under a variety of different loading conditions. A categorization of regulating geometry and material nonlinearity has been established in connection with limited component reproduction of frame activities. In this work, we discuss bending sheets made of AISI1020 steel, 1060 aluminium alloy, and Duranickel alloy by using a bend test for a punch tool in the form of a V. The bending test was conducted on the sheets. The sheet metal strip undergoes permanent distortion due to the severe plastic strain applied to it. In bending sheet metal, one of the most significant challenges that might arise is the production of spring back as the material is being unloaded. In this line of study, the design of experiments and finite element analysis is used to examine sheet metal's bending characteristics. To explore the elasto plastic behaviour, parameterised numerical simulations are used. The study on the static mechanical behaviour of metal sheeting for various materials is done to establish its correlations. To carry out the procedure systematically, a FEM analysis of sheet metal three-point bending has to be established.

Keywords: Geometric, Nonlinearity, Material, Nonlinearity.

INTRODUCTION

The technique is known as "bending sheet metal" and is one of the methods of sheet steel forming utilised most of the time. Fabrication methods that include bending sheet steel are widely used in commercial settings to produce automobiles and aero planes. These methods include bending the steel into various shapes. These approaches frequently entail a system of trial and error for bending the sheet to the necessary attitude. About the bending operation, "the accuracy and success of the operation are determined not only by the working circumstances but also by the properties of the fabric" [1]. The spring-back impact is the most prevalent issue in U and V-shaped components, and it may result in large changes to the bend angle. This flaw is more noticeable in substances with greater power-to-modulus ratios, such as aluminium and excessive power metal. One of the operational reasons for bending conducted the most regularly in sheet metal forming processes is for engineering regions of bending. The great bulk of research on spring efficacy has been focused on sheet metal forming methods and systems in the automobile and aviation sectors [2]. Various automobile and aviation businesses have been founded expressly to do this study. The bending manner is characterised as the maximum deformation of the shape, and positive fault evaluations choose it in both the resistance and mechanical technique of defects. This was discovered by utilising lines in the unstable sequence of events that preceded this processing.

The methods of metal forming, also known as metal running, may be split into two main categories: bulk-forming and sheet steel forming. Some examples of techniques that come within the category of bulk-forming include forging, rolling, and extrusion. In this method, a sliver of plastic fabric is twisted into numerous useful forms [2]. Changing flat, thin sheet steel blanks into the requisite shapes is referred to as sheet metal forming. Press running is another name for this particular procedure. Forming sheet metal may be accomplished using various methods, including deep drawing, stretching, and bending, amongst others. It is normal practice in many sectors, including the automobile industry, the aerospace industry, and the appliance business, to produce a wide array of additives that range from simple to complex.

A sheet metal blank is made to endure pressure in preparation for the bending operation during the bending process. Because of this, the blank is pushed to bend at an angle, which ultimately results in the production of the intended shape. The painted object is first bent in a location that has some flexibility so that the process may begin. The thing that is being worked on goes through plastic deformation as the process continues forward, resulting in a change in the object's shape. As a result of the applied tension, the fabric is stretched farther than what is permitted by its yield energy but not further than what is permitted by its maximum tensile strength [3]. As a direct result of the bending process, the sheet metal will go through the sensations of compression and anxiety as a direct result of bending the sheet metal. At the same time, this causes the sheet's inner surface to be squeezed and lengthened. As a result, the sheet's outer surface is stretched out for a much longer time than the inner surface.

REVIEW OF LITERATURE

Khan Muhammad Idrees et al. (2022) The bending transition zone variation laws for V-shaped sheets and plates will be the primary focus of this research. According to initial ideas, the sheet may be broken down into two parts: the area undergoing deformation and the area not undergoing deformation. We often fail to recognise this fact, even though deformation cannot be reversed into non-deformation. In the V-shape bending simulation performed using aluminium alloy 7075-0, all transition sites of the work pieces have been determined by nodal displacement measurement. After that, I split a set of work pieces into five layers, and after they are all in the transition area, I measure the distortion values on each of those layers individually. Using these data, I was able to work out the fluctuation rules that govern the transition from the inner layer to the outer layer in terms of its width. An in-depth analysis was carried out to investigate the distinct width variations in the transition region. This investigation made use of a variety of beginning widths and punch radii.

Y. Parra-Rodríguez et al. (2021) "uniaxial tensile and V-die bending tests" are carried out on the material to investigate the DP600 steel's adaptability in several different directions concerning the rolling direction. In addition, microstructural research was carried out in each deformation zone so that the morphological changes in the microstructure grains could be shown. Anisotropy models were used to explore the plastic anisotropy shown by the material. The Hill-48 and the Barlat-89 were only two of the many types that fell into this category. When the rolling direction was turned 45 and 90 degrees, there was a noticeable increase in the amount of elastic recovery. Due to the unusual shape of the martensite, the rolling process creates zones of optimum placement inside the material. Hill-48 and Barlat-89 are two models that can offer a realistic description of the experimental yield surface and plastic anisotropy. Hill-48 was developed in 1948, while Barlat-89 was developed in 1989. In the year 1948, Hill-48 was created, and in the year 1989, Barlat-89 was produced. Compared to the experimental data, the Hill-48 model and the finite element methodology were both able to offer an accurate estimate of the elastic recovery that occurred during the V-die bending test. This was determined by examining how well each method estimated the recovery. This was made clear because both models were able to anticipate the experiment's outcomes precisely.

A finite element study is performed with the help of the ANSYS programme, which is then used in the real analysis performed by A. Dhilip et al. (2021). The purpose of this study is to make a comparison between the three distinct types of material in terms of the Maximum Principal Stress in (Mpa), Maximum Normal Stress in (Mpa), Equivalent Stress in (Mpa), and Maximum Principle Elastic Strain as inputs, and Ultimate Tensile Strength (UTS), and Yield as outputs, while bending the material. Specifically, the Maximum Principal Stress in (Mpa), Maximum Normal Stress in (Mpa), and Equivalent Stress in (Mp. This comparison will be made in terms of the stresses that are measured in megapascals (Mpa), as well as the corresponding stress that is also measured in megapascals. According to the research results, the maximum normal stress and maximum primary stress values for stainless steel are both lower than the values for annealed carbon steel 1020 in both cases. In addition, when compared to the other two materials, stainless steel requires the lowest amount of stress to begin deforming and the largest amount of stress to fail, while the other materials need the opposite requirement. In addition to this, its Hardness rating is far greater than that of the other two choices combined, making it the clear winner in this category. Because stainless steel 1020 annealed encounters lower maximum normal and main stresses than carbon steel 1020, one would think that the stainless steel's UTS would be lower; yet, the stainless steel's UTS is very high. This is due to the higher resistance to bending that stainless steel has compared to carbon steel.

According to Sourav Kumar Das et al. (2021), the automotive and aerospace sectors make substantial use of a process that includes manufacturing components using sheet metal. Specifically, the method involves punching holes into the sheet metal. Utilising a punch and die to form the appropriate contours and dimensions out of a sheet of flat material is the first step of the time-honoured manufacturing process known as metal shaping. This approach has been around for a considerable amount of time. This method dates back millennia and has a proven track record. In the current investigation, the energy-absorbing properties of copper sheet metal of varying sheet thicknesses were investigated with the use of a workstation application named.

FINITE ELEMENT METHOD

In a wide range of engineering applications, the FEM technique may solve numerical problems. The technique may handle any intricate shape or geometry independent of material or boundary limitations, as long as the method is broad enough to solve the issue. The generality of the finite element technique makes it appropriate for intricate engineering structures and systems whose governing equilibrium equations lack closed-form solutions. Consequently, parametric design studies may be carried out using this drawing tool, which allows designers to investigate a number of design scenarios (forms, materials, loads, and so on) to determine the ideal design[4].Stress analysis of intricate aircraft systems was pioneered by the aerospace industry utilising this technique. It was developed from aviation design's so-called matrix analysis approach. There is much support from researchers and practitioners for this method. The assumption that a body or structure may be partitioned into "finite components" serves as the conceptual basis of the finite element approach. There are merely a few joints, termed nodes or nodal points, that link the different components together to create the fundamental frame or framework.

The FEM is a numerical technique for addressing problems that may either be stated as functional minimisation or be characterised by partial differential equations. This approach was created in the 1960s. A representation of a domain of interest may be built using a collection of finite elements. When dealing with finite elements, approximation functions are determined based on the nodal values of a physical field sought. An ongoing issue with the physical world is turned into a discretised problem using the FEM and has unknown nodal values [5]. To discover a solution to a linear problem, it is essential to solving a set of equations using linear algebra. Utilising the nodal values of the finite elements makes it feasible to access the values inside them. There are two aspects of the FEM that need more thought and concentration, and they are as follows: 1) The piecewise approximation of physical fields on finite elements is capable of providing a high level of accuracy. This is the case even when the approximating functions being used are quite simple (increasing the number of elements, we can achieve any precision). 2) The localisation of the approximation is responsible for the sparseness of the equation systems involved in a discretised problem. Because of this, it is now much simpler to address problems that include a significant number of undetermined nodal values.

In finite element models, the degree of freedom may vary anywhere from tens of thousands to even hundreds of thousands, depending on the practical analysis being performed. It is impossible to make such meshes by hand because they need too much precision. A piece of software known as a mesh generator does its task by segmenting the solution domain into a sizeable number of more manageable subdomains called finite elements. There is more than one kind of mesh generator available to users today. This section will focus on discussing two types of tools: block mesh generators and triangulators [6]. Both of these types of tools are used to create two-dimensional meshes. Block mesh generators need initial gross partitioning to carry out their intended functions. The issue area has been broken up into several separate pieces that can be controlled with far less effort than before. Every single block has to conform to a certain type of predetermined shape.

When producing the mesh that is included inside the block, a mapping approach is used in most instances. In the vast majority of instances, Triangulators will generate an irregular mesh inside arbitrary domains. In mesh creation, Voronoi polygons and Delaunay triangulation are often used as building blocks. Converting a later triangular mesh into a mesh comprised of components of a quadrilateral form is possible. The Delaunay triangulation algorithm may be generalized to be used in three-dimensional settings.

METHOD OF DISCRETE ELEMENTS AND ITS GENERAL OPERATING PROCESS

To analyse with the help of finite elements, a model or element must first be sectioned off into more manageable subcomponents that are also called finite elements and have restricted dimensions. To achieve the desired outcome, the initial model or structure must be recreated as an assembly of the variables above, each of which must be linked at a distinct nodal position or point. Because the actual change of subject variables such as displacement, stress, temperature, and pressure within the continuum is still a mystery, a simple function can approximate ground variables. The actual change of subject variables within the continuum is still a mystery [7]. The temperature and pressure of the ground are two examples of ground variables. The estimate functions, also known as interpolation models as a group, take as their input the field variables of the nodes in question. Adjustments to the field's nodal standards may be made if the corresponding equations for the field, which are often in the form of matrix equations, can be solved.

After the nodal values have been provided, the approximation properties of the field variable become accessible for use. The method known as finite elements is commonly used to present a step-by-step presentation of challenging ongoing problems.The following is one method that may be used when dealing with static structural applications:

The first step explains how the model was put together (Domain). During the beginning of the finite element approach, the structure of the output area is broken up into several more manageable sub-areas that are collectively referred to as elements.In the second step, the appropriate interpolation form was chosen. We assume that it is difficult to accurately predict the dislocation (field variable) explanation of a problematic structure under any given set of load circumstances to provide an estimate for the unknown answer [2]. The expected conclusion has to be straightforward and able to fulfil a variety of convergence conditions.The third step involves beginning work on the component stiffness (feature matrix) matrices and loads. It is possible to obtain the stiffness matrix of the element e, which is denoted by the symbol K (e), as well as its load vector, which is denoted by the symbol P (e), by employing equilibrium conditions in conjunction with an appropriate variation precept and applying this to the unspecified displacement model [8].The fourth step is to combine the individual expressions of the equations to bring about equilibrium in the system. The individual element rigidity matrices and load vectors need to be gathered properly, and the overall equilibrium equation has to be expressed in as much detail as is humanly feasible:

$$[K]\varphi = P$$

[K] refers to a set of stiffness matrices, denotes the vector of nodal variations, and P represents the vector of nodal pressure across the whole form.

Finding the values of the nodal displacements by solving a system equation is the fifth step (subject variable). Modifications need to be made to the traditional balanced equations to reflect the model's boundary conditions in question adequately. Once the boundary conditions have been integrated, the equilibrium equations may be given.

$$[K]\varphi = P$$

The vector may be handled very quickly and with little fuss when working with linear issues. This is because the vector is linear. To solve nonlinear issues, it is necessary to do so in a series of stages, and each step needs the stiffness matrix [K] and the weight vector P to be modified. In addition to this, it is essential to accomplish the goal of finding the answer in phases. In the sixth step, calculations of strain and stress are carried out at the atomic level [9]. To calculate the detail lines and stresses based on the nodal displacements taken into account, the main equations of structural mechanics of stable mechanics may be used. The logical progression of the FEM's step-by-step method is bolstered by the same terminology used in the phases that came before it.

CONCLUSION

This study focuses on sheet and plate transition zone variation rules for V-shape bending. Given a bending sheet, there are two different deformation regions, one caused by punch loading and the other unaffected by punch loading; nonetheless, with our current understanding of bending sheets, we tend to disregard the notion that deformation areas cannot be turned into non-deformation areas. We believe there is no distortion in the non-deformation zone but severe distortion in the width direction in the deformation area. The transition zone between the deformation zone and the non-deformation zone is the focus of this research. Distortion occurs in the transition region, and its value does not remain constant or change with huge values but rather varies slowly and with extremely few values. In this research, I study the transition zone of bending sheets and plates in terms of location, interval, and distortion value using finite element analysis. The bending of thin sheets and plates is modelled using the ANSYS Finite Element software.

REFERENCES

[1]. Janis Reinold, Venkatesh Naidu Nerella, Viktor Mechtcherine, Günther Meschke, Extrusion process simulation and layer shape prediction during 3D-concrete-printing using the Particle Finite Element Method, Automation in Construction, Volume 136, 2022, 104173, ISSN 0926-5805.
[2]. Daniel Ramirez-Tamayo AyoubSoulam, A complex-variable finite element method-based inverse methodology to extract constitutive parameters using experimental data, International Journal of Solids and Structures 243 (2022) 111545.
[3]. Changqing Huang and Hongsheng Liu (2022). Investigation of current-assisted electromagnetic tensile forming for sheet metal, The International Journal of Advanced Manufacturing Technology volume 118, pages2221–2232.
[4]. Ashraff Ali, K. S., Suresh Kumar, S., Allen Jeffrey, J., Ravikumar, M. M., &Rajkumar, S. (2021). An insight into stress and strain analysis over on hexagonal aluminium sandwich honeycomb with various thickness glass fiber face sheets. Materials Today: Proceedings, 47, 493–499. https://doi.org/10.1016/j.matpr.2021.05.038

[5]. Gavrilescu, I.; Boazu, D.; Stan, F. "Estimating of Bending Force and Curvature of the Bending Plate in a Three-Roller Bending System Using Finite Element Simulation and Analytical Modeling". Materials 2021, 14, 1204. https://doi.org/10.3390/ ma14051204.
[6]. Y. Parra-Rodríguez, J. M. Arroyo-Osorio, R. Rodríguez-Baracaldo, "Experimental and computational analysis of springback in dual phase steels" Scientia et TechnicaAño XXVI, Vol. 26, No. 02, junio de 2021
[7]. A. Dhilip, A.Archana, M.Brindha, M.Kanmani, "Bending Stress Analysis In Sheet Metal Forming Process", Nat. Volatiles &Essent. Oils, 2021; 8(5): 3064-3075.
[8]. Sourav Kumar Das, Dr. BibhutiBhusanPani, "Explicit Dynamic Analysis and Design Optimization of Sheet Metal Forming", International Journal of Innovative Science and Research Technology , Volume 6, Issue 4, April – 2021.
[9]. Alhijazi, M., Zeeshan, Q., Qin, Z., Safaei, B., &Asmael, M. (2020). Finite Element Analysis of Natural Fibers Composites: A Review. Nanotechnology Reviews, 9(1), 853–875. https://doi.org/10.1515/ntrev-2020-0069
[10]. Stephen A Akinlabi, Olawale S Fatoba, Peter M Mashinini, "Effect of Bottoming on Material Property during Sheet Forming Process through Finite Element Method", Materials Science and Engineering, 328, 2018.
[11]. AmulBiradar, M.D Deshpande, "Finite Element Analysis of Springback of a Sheet Metal in Wipe Bending Process", International Journal of Science and Research (IJSR), Volume 3, Issue 7, July 2014.