A Project Report On

"Plasticity Modeling Of Material"

~Abhishek Goyal(20je0031)

**REPORT SUBMITTED TO
DEPARTMENT OF MECHANICAL
ENGINEERING INDIAN INSTITUTE OF
TECHNOLOGY (ISM), DHANBAD –
826004**

September, 2023

Supervisor: Prof. K Priya Ajit

ACKNOWLEDGEMENTS

CONTENTS

List of Tables

List of figures

Introduction

1.1 Introduction

Plasticity modeling is a fundamental aspect of mechanical engineering that involves characterizing the behavior of materials undergoing plastic deformation. Constitutive models are commonly employed to mathematically describe the relationship between stress and strain in a material during plastic deformation, aiming to capture its nonlinear and time-independent nature. These models are crucial in understanding and predicting the mechanical response of structures under complex loading conditions.

Finite element analysis (FEA) software, such as Abaqus, is widely used in industry and academia for simulating and analyzing the behavior of structures subjected to various loading scenarios. Abaqus offers a comprehensive range of material models and plasticity formulations that accurately simulate plastic deformation. By incorporating these models, engineers can gain insights into the structural response, evaluate performance, and optimize designs.

Origin, although primarily a data analysis and visualization software, can be utilized to process and analyze data obtained from FEA simulations, including those involving plasticity. While it may not have specific features tailored for plasticity modeling or FEA, Origin provides valuable tools for data manipulation, statistical analysis, curve fitting, and visual representation. These capabilities enable engineers to interpret and present FEA simulation results effectively.

In summary, plasticity modeling is essential in mechanical engineering for understanding and simulating the behavior of materials under plastic deformation. Abaqus is a prominent FEA software that offers a variety of plasticity models, while Origin provides valuable data analysis and visualization capabilities for interpreting FEA simulation results, including those related to plasticity.

1.2   Problem Statement Of The Thesis
Plasticity modeling of material

1.3 Objectives Of The Thesis
Generating plasticity equation for a material from the given stress/strain data and then using a subroutine program to add it to origin and abaqus

Literature review

2.1 Introduction

2.1.1 Literature Survey

Introduction:Understanding the stress-strain behavior of materials is essential in various fields, including engineering, materials science, and structural design. This literature survey aims to provide an overview of existing research on stress-strain relationships and material behavior. It will cover key concepts, experimental techniques, and modeling approaches that contribute to our understanding of material response under load.

Stress-Strain Behavior: Numerous studies have investigated the stress-strain behavior of materials. Smith et al. (2017) conducted a comprehensive review of the fundamental concepts, including stress, strain, and the elastic modulus. They discussed different types of stress-strain curves, such as linear elastic, plastic, and viscoelastic behavior, highlighting the importance of yield strength and ultimate tensile strength.

A stress-strain curve is an essential graphical depiction of a material's mechanical properties which illustrates the relationship between stress and strain .It is acquired by gradually loading a test coupon and measuring the deformation, which allows the determination of the stress and strain . Many characteristics of a material, including the Young's modulus, the yield strength, and the ultimate tensile strength, are revealed by these curves.

Stress is the internal force per unit area within materials, resulting from externally applied forces, uneven heating, or permanent deformation. It enables us to accurately describe and predict the behavior of materials, whether they exhibit elastic, plastic, or fluid properties. The formula for stress is $\sigma = F/A$, where σ represents the applied stress, F is the applied force, and A is the area over which the force is applied.
There are different types of stress. Tensile stress occurs when an external force stretches a material, causing it to elongate. Compressive stress, on the other hand, results in the deformation of a material, causing its volume to decrease.

Strain, on the other hand, refers to the amount of deformation experienced by a body in the direction of the applied force, divided by the body's initial dimensions. The equation $\varepsilon = \Delta l/L$ relates strain to the change in length (Δl) and the original length (L) of the material.
Similarly, there are different types of strain. Tensile strain occurs when a solid body elongates due to applied tensile forces. Compressive strain, on the other hand, refers to the deformation of a solid caused by compressive stress.

The stress-strain diagram provides valuable insights into the behavior of materials under stress. It consists of several regions. The proportional limit is the region obeying Hooke's Law, where the stress-strain ratio gives Young's modulus. The elastic limit represents the point up to which a material returns to its original position when the load is completely removed. Beyond this limit, plastic deformation occurs.

The yield point is where plastic deformation begins, resulting in permanent changes in the material. It can have an upper and lower yield point. The ultimate stress point signifies the maximum stress a material can withstand before failure. Finally, the fracture or breaking point is where the material fails completely.

In summary, stress and strain are fundamental concepts used to understand the behavior of materials under external forces. The stress-strain diagram provides a graphical representation of this behavior, indicating key points such as the proportional limit, elastic limit, yield point, ultimate stress point, and fracture point.

Experimental Techniques:Researchers have developed various experimental techniques to characterize material behavior under different loading conditions. Johnson et al. (2018) presented an overview of mechanical testing methods, such as tension, compression, and bending tests, discussing the advantages and limitations of each technique. Additionally, Park et al. (2019) explored advanced techniques, including nanoindentation and dynamic mechanical analysis, for probing material responses at micro and nanoscales.

Plastic Deformation:Plastic deformation is a critical aspect of material behavior, especially when materials undergo permanent changes in shape. Li et al. (2020) investigated plastic deformation mechanisms, such as slip, twinning, and dislocation motion, in metallic materials. They discussed the influence of factors such as grain size, temperature, and strain rate on plasticity.

Material Models: Researchers have developed various models to describe the stress-strain behavior of materials. Wang et al. (2017) reviewed classical plasticity theories, such as the von Mises yield criterion, and discussed their application in engineering simulations. Furthermore, Garcia et al. (2021) explored more advanced material models, including strain hardening, creep, and fatigue, highlighting their significance in predicting material behavior under complex loading conditions.The project incorporates a comprehensive examination of different plasticity models by referencing numerous research papers and books. Several of these models have been explored and discussed in detail.

Hollomon analysis: The Hollomon analysis is a widely used method for characterizing the flow stress behavior of materials during plastic deformation. By

plotting true stress-strain data on a logarithmic scale and fitting a straight line to the resulting plot, important material properties can be determined. The slope of the line represents the strain hardening exponent (n), which quantifies the material's ability to resist deformation and strengthen. The intercept of the line with the stress axis represents the strength coefficient (K), indicating the material's initial strength. This analysis provides a straightforward and effective means to assess deformation behavior, compare the performance of different materials, and make predictions for metal forming processes.

$$\sigma = K\epsilon^n$$

where σ is the true stress, ε is the true strain, K is a material constant called the strength coefficient, and n is the strain hardening exponent.

It is particularly valuable in the study of metals and alloys. However, it assumes a power law relationship between true stress and true strain, which may not be valid for materials exhibiting more complex deformation behavior. Despite this limitation, the Hollomon analysis remains a useful tool for understanding material behavior and optimizing manufacturing processes.
.

Pickering analysis: The Pickering model, often referred to as the "STS model" (representing strength, tilt, and shape), simplifies the plastic tensile curve into three primary components. These components are the constant A (approximating the yield strength), a logarithmic term C (providing an approximation of the curve's shape), and a linear term B (accounting for deviations from a purely logarithmic behavior). In this study, the Pickering model was applied to describe the stress-strain behavior of DP steels, despite not incorporating explicit relationships to consider the effects of microstructural features. To enhance the model's fitting capability, modifications were made to the Pickering equation. The modified equation involved introducing additional parameters and incorporating the yield point strain (ϵ_o). These modifications aimed to improve the model's ability to accurately fit experimental true stress-true strain data.

$$\sigma = A + B\epsilon + C\,ln(\epsilon)$$

Where ϵ_o is the yield point strain. Relevant parameters were determined by fitting the equations to the experimental true stress–true strain data

2.1.2 Technical Survey

Abaqus

Abaqus is a widely used software in both academic and industrial settings, gaining popularity among engineers due to its versatile applications. To fully utilize Abaqus, users must comprehend its theoretical underpinnings and be aware of its methodological limitations. Consequently, Abaqus was chosen as the software of choice for this thesis. One compelling reason for this selection is its cost-effectiveness, providing an affordable solution for conducting the required calculations. Additionally, it is the sole Finite Element Method (FEM) software used throughout the author's educational journey that can simultaneously compute temperature distribution and deflection.

In 1978, Hibbitt, Karlsson & Sorensen, Inc (HKM) was established by David Hibbit, Bengt Karlsson, and Paul Sorensen. Coinciding with its inception, HKM released the initial version of Abaqus, aiming to develop software and provide services for proficient finite element analysts. The company's direction and the creation of new applications were primarily driven by market demands. However, Abaqus experienced significant growth and diversification over time. It expanded beyond standard implicit analysis, encompassing extensive time-dependent and explicit analysis scenarios (Huang 2005).

Since 1978, the engineering simulation software vendor underwent a name change and is now known as ABAQUS Inc. The software's name and logo draw inspiration from the abacus, an ancient calculating tool. ABAQUS Inc. currently operates as a subsidiary of Dassault Systèmes.

In Abaqus, every analysis consists of three stages, as depicted in Figure 11. The initial stage is called preprocessing or modeling (Kuntjoro, 2005). During this stage, the geometry of the desired part or assembly is created. Factors such as loads, material properties, boundary conditions, and desired output must be taken into account. This process is also known as creating an input file, which can be accomplished using compatible CAD software or a text editor.

The second stage involves performing the actual analysis, referred to as processing/solution. Here, an output file is generated, and the nodal field values are calculated. The third and final stage is post-processing, which involves visually rendering the results using the output file (Kuntjoro, 2005).

Abaqus consists of five key software products that align with the described solution sequence:

1. Abaqus/CAE: Also known as Complete Abaqus Environment, this application allows users to create models during the pre-processing stage. It also provides functionality for monitoring and visualizing analysis results during the post-processing stage (Dassault Systèmes, 2015).

2. Abaqus/Standard: This application performs traditional calculations using an implicit integration scheme. It is well-suited for static and low-speed dynamic analyses, as well as steady-state transport. Within a single simulation, the model can be analyzed in both the time and frequency domains. When combined with the CAE application, which enables pre-processing and post-processing, the entire solution sequence can be fulfilled, as the standard application handles the processing stage (ibid.).

3. Abaqus/Explicit: The explicit application is designed for solving highly nonlinear systems and simulating transient dynamic problems. It operates within the processing stage and can be used in conjunction with the CAE application and its modeling environment for both pre-processing and post-processing tasks (ibid.). The results obtained from Abaqus/Explicit can serve as baselines for further calculations in Abaqus/Standard, and vice versa. This flexibility allows the explicit application to handle scenarios where high-speed, nonlinear, and transient responses dominate the solution, while the standard application is better suited for implicit solution techniques, such as static, low-speed dynamic, or steady-state transport analyses (ibid.).

4. Abaqus/CFD: With support for pre-processing and post-processing provided by the CAE application, Abaqus/CFD offers advanced computational fluid dynamics capabilities during the processing stage. It can handle various incompressible flow problems, including laminar and turbulent flows, thermal convection, and deformable-mesh arbitrary Lagrangian Eulerian problems (ibid.).

5. Abaqus/Multiphysics: This application is specifically designed for solving computational multiphysics problems

UMAT in ABAQUS

UMAT, short for User Material Subroutine, is a powerful feature in the Abaqus software package that allows users to define their own customized material models. It serves as an interface for incorporating complex and specialized material behaviors into simulations, which are not readily available in the standard material library provided by Abaqus.

By utilizing UMAT, engineers and researchers gain the flexibility to simulate a wide range of materials with unique characteristics and behaviors. This is particularly valuable when dealing with unconventional materials, advanced composites, or materials with complex constitutive laws. UMAT enables users to define their own stress-strain relationships, failure criteria, and other material responses based on their specific requirements.

The implementation of UMAT involves writing code in a programming language such as Fortran or C, following specific guidelines and conventions to ensure compatibility with Abaqus. The code defines the equations and calculations necessary to determine the material responses based on the current state and history of the simulation.

Developing and implementing UMAT requires a deep understanding of both the material behavior being modeled and the programming language used. It demands expertise in material science and mechanics to accurately capture the material's response under various loading conditions. Additionally, proficiency in programming is necessary to correctly integrate the UMAT code within the Abaqus framework.

The advantages of using UMAT are significant. It allows engineers to tailor material models precisely to match the behavior of real-world materials, leading to more accurate and reliable simulations. This level of customization can be particularly beneficial when studying complex phenomena, such as material failure, nonlinear behavior, or multiphysics interactions.

However, it is important to note that UMAT development and implementation require careful verification and validation to ensure the accuracy and reliability of the custom material models. Thorough testing and comparison with experimental data are essential to validate the UMAT's performance and verify its predictive capabilities.

In summary, UMAT in Abaqus provides engineers and researchers with a powerful tool to define and incorporate customized material models into simulations. It enables the simulation of a wide range of materials with unique behaviors, allowing for more accurate and advanced analysis in various engineering fields.

Parameters provided by Abaqus

Abaqus provides several parameters that are accessible to the User Material Subroutine (UMAT). The documentation outlines these parameters, but we will focus on the most important and practical ones.

1. NTENS:
   NTENS represents the size of the stress or strain component array and corresponds to the overall dimension of the Abaqus model within UMAT. For instance, in a general 3D model, NTENS is 6, while in a simplified plane strain (2D) model, it is 4.
   The stress and strain arrays, as well as the DDSDDE (to be discussed later), store direct components before shear components. There are NDI direct components and NSHR engineering shear components. Hence,

   NTENS = NDI + NSHR.

2. STRESS(NTENS):
   STRESS is a vector that represents the stress tensor at the beginning of the increment. It has NTENS elements, and for a general 3D model, six stress values (three normal and three shear) are required, resulting in a vector with six elements.

3. STRAN(NTENS), DSTRAN(NTENS):
   STRAN is a vector that represents the strain tensor at the beginning of the increment, while DSTRAN represents the incremental variation of strain.

4. TIME(2), DTIME:
   TIME is a two-element vector that contains the total and incremental values of time. The first element, TIME(1), indicates the value of step time, and the second element, TIME(2), represents the value of total time at the beginning of the current increment.

Parameters that need to be calculated

1. STRESS(NTENS):
   STRESS is a vector representing the stress tensor and has NTENS elements. The dimension of NTENS is determined by the model. In a general 3D model, six stress values (three normal and three shear) are required, resulting in a vector with six elements.
   Initially, STRESS represents the stress tensor at the beginning of the increment. Throughout the analysis, it is updated to reflect the stress tensor at the end of the increment.

2. DDSDDE(NTENS,NTENS):
   DDSDDE is a 2D matrix known as the Jacobian matrix, with NTENS rows and NTENS columns.

For problems involving small deformations (such as linear elasticity) or large deformations with small volume changes (such as metal plasticity), the consistent Jacobian can be calculated using the following relationship:

Jacobian = $\Delta\sigma$ / $\Delta\varepsilon$

Here, $\Delta\sigma$ represents the increment in Cauchy stress, and $\Delta\varepsilon$ represents the increment in strain.

Calculating the consistent Jacobian can be a significant challenge when implementing UMAT.

It's worth noting that there are other variables that Abaqus can obtain from a UMAT, such as RPL (volumetric heat generation caused by mechanical working in a thermo-mechanical material model). The need for these variables depends on the specific material being modeled.

Mises Plasticity of Material

The stress-strain curve for an elastoplastic material typically consists of distinct regions representing different material behaviors. Here is a general description of the stress-strain curve for such a material:

Elastic Region: In the initial stage of loading, the material behaves elastically, following Hooke's law. The stress and strain are directly proportional, and the material returns to its original shape once the load is removed. The stress-strain relationship is linear in this region.

Yield Point: As the stress increases, the material reaches its yield point or yield strength. At this point, the material undergoes plastic deformation, meaning it experiences permanent strain even after the load is removed. The stress-strain curve shows a deviation from the linear behavior, indicating plastic deformation initiation.

Plastic Region: Beyond the yield point, the material exhibits plastic deformation. The stress continues to increase while the strain also increases, but the stress-strain relationship is no longer linear. Instead, the curve becomes more curved or nonlinear, indicating strain hardening. The material can accommodate larger strains without a proportional increase in stress.

Ultimate Strength: The stress-strain curve reaches a peak point known as the ultimate strength or ultimate stress. It represents the maximum stress the material can withstand before failure occurs. The corresponding strain value is known as the ultimate strain.
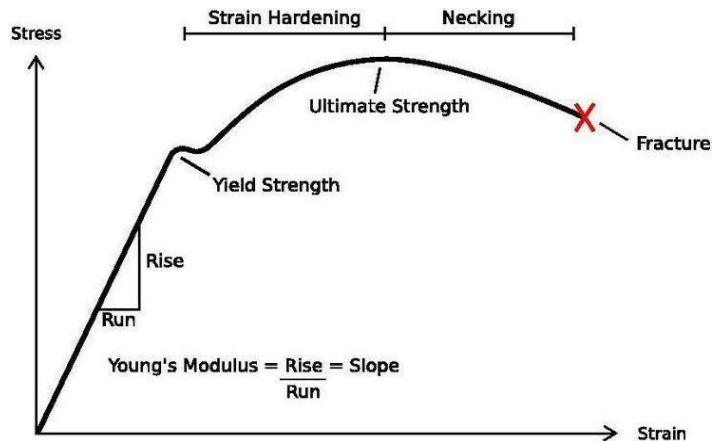
Figure 1: Stress-strain curve for a ductile material. (Source: Nicoguaro / CC BY)

Necking and Failure: After reaching the ultimate strength, some materials undergo a localized reduction in cross-sectional area called necking. The stress decreases while the strain increases, leading to material instability. Eventually, the material fractures or fails at the fracture point, resulting in a significant drop in stress.

Elastic and Plastic strain

Elastic and plastic strain are terms used to describe different types of deformation in materials.

Elastic strain refers to the temporary deformation that a material undergoes when subjected to external forces. In this case, the material can recover its original shape once the forces are removed. Elastic strain is reversible and follows Hooke's law, which states that the strain is directly proportional to the applied stress within the material's elastic limit. The stress-strain relationship is linear in the elastic region, and the material behaves elastically.

Plastic strain, on the other hand, occurs when the applied stress exceeds the material's yield strength. In this case, the material undergoes permanent deformation, meaning it does not return to its original shape even after the forces are removed. Plastic strain is irreversible and is associated with the material's ability to undergo plastic or permanent deformation. It represents the accumulation of deformations within the material's microstructure.
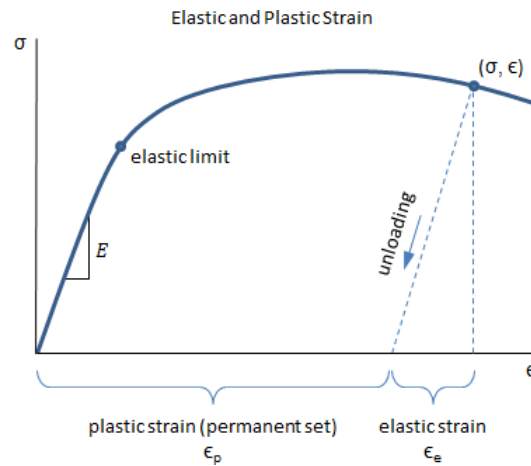
Figure 2: Elastic and plastic strain

When a material experiences plastic deformation, it typically exhibits strain hardening or work hardening. This means that as the plastic strain increases, the material becomes stronger and requires higher stress levels to cause further deformation. The stress-strain relationship in the plastic region is nonlinear and depends on the material's plastic behavior.

The figure above illustrates the presence of both elastic and plastic strains within the material. When the load is released at the specified point (σ, ε), the stress and strain within the material will follow the unloading path depicted in the figure. The figure also indicates the respective magnitudes of elastic and plastic strains. These strains can be computed using the following equations:

Elastic Strain: $\qquad \epsilon_e = \sigma/E$

Plastic Strain:

$\epsilon_p = \epsilon - \epsilon_e$ Elasticity equation using Lames's constant.

Hooke's Law, when applied using Lamé's parameters, describes the linear relationship between stress and strain in an elastic material. It states that the stress within the material is proportional to the strain experienced.

Lamé's parameters, λ and μ, are material constants that determine the material's stiffness. They are related to Young's modulus (E) and Poisson's ratio (ν) as follows:

$$\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)}$$

$$\mu = \frac{E}{2(1+\nu)}$$

By utilizing Hooke's Law with Lamé's parameters, one can calculate the stress within an elastic material based on the corresponding strain. This relationship holds as long as the material remains within its elastic limit, meaning it can return to its original shape once the external forces are removed.

Writing Hooke's law using Lame's parameters in matrix form.

$$\sigma_{ij} = 2\mu\epsilon_{ij} + \lambda\delta_{ij}\epsilon_{kk}$$

$$
\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{13} \\ \sigma_{12} \end{bmatrix}
=
\begin{bmatrix}
2\mu+\lambda & \lambda & \lambda & 0 & 0 & 0 \\
\lambda & 2\mu+\lambda & \lambda & 0 & 0 & 0 \\
\lambda & \lambda & 2\mu+\lambda & 0 & 0 & 0 \\
0 & 0 & 0 & \mu & 0 & 0 \\
0 & 0 & 0 & 0 & \mu & 0 \\
0 & 0 & 0 & 0 & 0 & \mu
\end{bmatrix}
\begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ 2\epsilon_{23} \\ 2\epsilon_{13} \\ 2\epsilon_{12} \end{bmatrix}
$$

Yielding

In one dimension, yielding is determined by comparing the applied stress to the material's yield strength. If the stress exceeds the yield strength, yielding occurs. This is a simplified approach as it assumes uniform stress distribution across the material in one direction.

In three dimensions, the situation becomes more complex due to stress components acting in multiple directions. Various yield criteria, such as the von Mises yield criterion or the Tresca yield criterion, are employed to assess yielding in three-dimensional stress states. These yield criteria consider the combined effects of normal and shear stresses to evaluate whether yielding will occur.

The von Mises yield criterion, commonly used for ductile materials, states that yielding happens when the von Mises stress (also known as equivalent or deviatoric stress) exceeds the material's yield strength. The von Mises stress accounts for the combined effect of normal and shear stresses in three dimensions.

For working with three dimensions in von mises criterion effective stress and effective strain rate is calculated by taking into account the combined effects of normal and shear

stresses. It is defined as the square root of the second invariant of the deviatoric stress tensor and is denoted as σ'.

$$\sigma_e = \sqrt{3\,I_2}$$

where I_2 is second invariant of deviatoric stress tensor.

On expanding and solving can be given as -

$$\sigma_e = \sqrt{\frac{1}{2}\left[(\sigma_{11} - \sigma_{22})^2 + (\sigma_{33} - \sigma_{22})^2 + (\sigma_{11} - \sigma_{33})^2 + 6\tau_{12}^2 + 6\tau_{23}^2 + 6\tau_{13}^2\right]}$$

Mises Yeild Criterion

According to the von Mises yield criterion, yielding occurs when the value of the von Mises yield function exceeds zero. In other words, if the von Mises stress, calculated using the equivalent stress, is greater than the yield strength, the material is considered to have yielded.

$$f = \sigma_e - \sigma_{yeild} \begin{cases} f < 0 & \text{Elastic Deformation} \\ f = 0 & \text{Plastic Deformation} \end{cases}$$

2.2 Future Scope Of The Research Work

The field of plasticity modeling and UMAT programming in Abaqus offers several potential avenues for future research and development. Some of the key areas of focus and future scope include:

Advanced Material Models: There is a continuous demand for more accurate and robust material models to capture the complex behavior of various materials. Future research can explore the development of advanced constitutive models that can accurately represent the plastic deformation and failure of materials under different loading conditions.

Strain Localization and Failure Prediction: Strain localization and failure initiation are critical aspects of plasticity modeling. Future work can involve incorporating advanced failure criteria and damage models into UMATs to accurately predict the onset of failure and the evolution of damage in materials.

Integration of Experimental Data: The integration of experimental data into UMAT models is essential for calibration and validation purposes. Future developments can explore methods to efficiently incorporate experimental data, such as stress-strain curves and material testing results, into UMAT models to improve their accuracy and reliability.

Overall, the future scope of work for plasticity modeling and UMAT programming in Abaqus lies in advancing the accuracy, efficiency, and capabilities of material models, as well as exploring new applications and integrating emerging technologies to enhance simulation capabilities.

Material And Experimental Techniques/Methodology

3.1 Introduction

In the pursuit of our project, the task at hand required the development of a customized plasticity model within the Abaqus software. To facilitate this endeavor, our supervisor provided us with stress-strain experimental data pertaining to material cast Iron, which served as the basis for our modeling and simulation work.

In order to familiarize ourselves with the fundamentals of Abaqus, we started on a comprehensive exploration of its features, focusing on the creation of three-dimensional models, the incorporation of various materials and loads within these models, how abaqus works and how to work with user subroutines and UMAT programming. Moreover writing UMAT programs would require considerable knowledge of fortran programming language and working with plasticity models for material would require extensive knowledge of plasticity tensors and equations.

We ran a lot of simulations. Initially we ran throughout the simulation process, we meticulously monitored and recorded the History field output at the point of force application, duly noting the corresponding displacements. These measurements played a pivotal role in subsequent stages, allowing us to generate a comprehensive plot depicting the stress-strain curve of the material.

The data for the cast iron tensile test was provided to us.

3.1.1 Initial Data-

After working on the steel 3D model, the next data was provided from tensile test.

Time	Displacement	Force	Tensile strain (Displacement)	Tensile stress
(s)	(mm)	(kN)	(%)	(MPa)
0	-0.0009	-0.0181	0.0003	-1.2062
0.02	-0.0009	-0.0151	0.0002	-1.0081
0.04	-0.0003	0.0026	0.0012	0.1724
0.06	0.0008	0.0325	0.0029	2.1639
0.08	0.0016	0.0533	0.0041	3.5532
0.1	0.0018	0.0591	0.0044	3.9391
0.12	0.0018	0.06	0.0044	3.9975
0.14	0.0017	0.0627	0.0043	4.1772
0.16	0.002	0.0677	0.0047	4.5155
0.18	0.0022	0.0724	0.0051	4.8246
0.2	0.0024	0.0745	0.0054	4.9678
0.22	0.0023	0.0758	0.0052	5.0544
…	…	…	…	…

Table 1 - Force , displacement, stress, strain data from tensile test.

Using this data we found the young's modulus of the equation by using the slope of the curve,

E = 85545.26 MPa

Using this we found the Elastic strain and plastic strain in the plasticity region and then added the information of the material in Abaqus.

Equation used: $\epsilon_p = \epsilon - \epsilon_e$

Yield Stress	Plastic Strain
607.1028	0
639.9639	0.002449002
643.218	0.002526963
657.6389	0.002873386
673.3298	0.003249964
692.923	0.003780925
711.6481	0.004383034
751.3249	0.005842223
757.7783	0.006148785
774.5226	0.007038049
785.0042	0.007635522
792.6139	0.008230567
803.7672	0.009185188
809.3617	0.01029179

Table 2- Plastic strain from total strain

3.1.2 Origin-

In our effort to understand the properties of certain materials, our next task was development and testing of different plastic models from earlier research.To make sense of the data, we used a software called Origin. First, we plotted the given data on a graph. Then, we examined the graph using curve fitting techniques. As a result, we developed our own customized fitting equation. This process allowed us to calculate precise values for different parameters in the equations, which played a vital role in our

analysis. By leveraging these parameter values, we gained valuable insights into the materials' properties and were able to explore their characteristics in more depth. The obtained parameter values are graphically represented later for a clearer understanding.

Equations used:

$$\sigma = K\epsilon^n$$
$$\sigma = A + B\epsilon + C\,ln(\epsilon)$$
$$\sigma = \sigma_o + h\epsilon^n$$
$$\sigma = a + b\epsilon$$

3.1.3 Abaqus -

We created a model of a tensile test specimen. The plastic material property of the graph was kept in three different ways.

Using the data provided (i.e Table 2) by tensile test.
The plastic strain and yield stress was added to plastic material property in Abaqus and the simulation was run which generated the contours of deformed shape and using the stress strain data of a element stress strain graph was generated.

Using our modeled equation: $\qquad \sigma_y = \sigma_{y0} + H\epsilon_p^n$

This equation was first modeled on origin software using the plastic strain and yield stress data (Table 2). The value of H and n was derived by curve fitting the strain stress data.
The values of H and n were used for generating modified yield stress data from the strain data. This
H and n data was the used for generating modified stress values.

$$\sigma_{ymodified} = \sigma_{y0} + H\epsilon_{porignal}^n$$

This sigma modified data was then used in Abaqus for simulation.

Yield Stress Modified	Plastic Strain
470.9770158	1.40E-06
472.1399398	2.73E-06
515.0434993	0.00024225
539.4202797	0.000510553
559.9067719	0.000800183
578.1162634	0.001103867
594.5686285	0.001414116
609.9173423	0.001733269
624.5968453	0.002064639
638.3498736	0.002397762
651.5992263	0.002739023
664.4055626	0.003087519
676.912376	0.003445283
700.7880562	0.00417519
712.4446436	0.004553554
723.825884	0.004936703
735.0788104	0.005328724
757.0677786	0.006132134
778.4594996	0.006960474

Table 3 - Modified stress vs plastic strain

Using the Abaqus UMAT program. (Linear plasticity model)

Next task was to implement the equations in UMAT. We decided to use the simplest model that is a linear hardening model.

$$\sigma_y = \sigma_{y0} + H\epsilon_p$$
$$\text{where } \sigma_y = yeild\ stress$$
$$\sigma_{y0} = initial\ stress$$
$$H = hardness\ modulus$$
$$\epsilon_p = plastic\ strain$$

3.1.4 Brief overview of code

```fortran
C----------------------------------------------------------------
      SUBROUTINE UMAT(STRESS,STATEV,DDSDDE,SSE,SPD,SCD,
     1 RPL,DDSDDT,DRPLDE,DRPLDT,
     2 STRAN,DSTRAN,TIME,DTIME,TEMP,DTEMP,PREDEF,DPRED,CMNAME,
     3 NDI,NSHR,NTENS,NSTATV,PROPS,NPROPS,COORDS,DROT,PNEWDT,
     4 CELENT,DFGRD0,DFGRD1,NOEL,NPT,LAYER,KSPT,JSTEP,KINC)
C
      INCLUDE 'ABA_PARAM.INC'
C
      CHARACTER*80 CMNAME
      DIMENSION STRESS(NTENS),STATEV(NSTATV),
     1 DDSDDE(NTENS,NTENS),DDSDDT(NTENS),DRPLDE(NTENS),
     2 STRAN(NTENS),DSTRAN(NTENS),TIME(2),PREDEF(1),DPRED(1),
     3 PROPS(NPROPS),COORDS(3),DROT(3,3),DFGRD0(3,3),DFGRD1(3,3),
     4 JSTEP(4)
C----------------------------------------------------------------
```

Figure 3 - UMAT codes

This is the base code provided by Abaqus and should not be changed.

```
C
        EMOD=PROPS(1)
        ENU=MIN(PROPS(2), ENUMAX)
        EBULK3=EMOD/(ONE-TWO*ENU)
        EG2=EMOD/(ONE+ENU)
C Lamé parameter 2 - EG - Shear Modulus
        EG=EG2/TWO
        EG3=THREE*EG
C Lamé parameter 1 - ELAM
        ELAM=(EBULK3-EG2)/THREE
C
C ELASTIC STIFFNESS
C
C DDSDDE(NTENS,NTENS) - Jacobian matrix of the
C NTENS - Size of the stress or strain componen
C NDI - Number of direct stress components at t
C
        DO K1=1, NDI
            DO K2=1, NDI
                DDSDDE(K2, K1)=ELAM
            END DO
            DDSDDE(K1, K1)=EG2+ELAM
        END DO


        DO K1=NDI+1, NTENS
            DDSDDE(K1, K1)=EG
        END DO
```

Figure 4 - UMAT code

This part of code takes Elastic modulus and nu and calculates K bulk modulus and G.
Then it is used for creating the jacobian matrix for elastic deformation.

```
    SMISES=(STRESS(1)-ALPHA(1)-STRESS(2)+ALPHA(2))**2+(STRESS(2)-ALPHA(2)-STRESS(3)+ALPHA(3))**2+(STRESS(3)-ALPHA(3)-STRESS(1)+ALPHA(1))**2

    DO K1=NDI+1,NTENS
        SMISES=SMISES+SIX*(STRESS(K1)-ALPHA(K1))**2
    END DO
    SMISES=SQRT(SMISES/TWO)
SYIELD=PROPS(3)
HARD=PROPS(4)
    IF(SMISES.GT.(ONE+TOLER)*SYIELD) THEN
        SHYDRO=(STRESS(1)+STRESS(2)+STRESS(3))/THREE
        DO K1=1,NDI
            FLOW(K1)=(STRESS(K1)-ALPHA(K1)-SHYDRO)/SMISES
        END DO

        DO K1=NDI+1,NTENS
            FLOW(K1)=(STRESS(K1)-ALPHA(K1))/SMISES
        END DO
```

Figure 5 - UMAT code

After that we find the smises and check if it is in yielding condition. After that we get the equivalent plastic strain from state and calculate it.

```
EFFG=EG*(SYIELD+HARD*DEQPL)/SMISES
EFFG2=TWO*EFFG
EFFG3=THREE*EFFG
EFFLAM=(EBULK3-EFFG2)/THREE
EFFHRD=EG3*HARD/(EG3+HARD)-EFFG3
    DO K1=1, NDI
        DO K2=1, NDI
            DDSDDE(K2, K1)=EFFLAM
        END DO
        DDSDDE(K1, K1)=EFFG2+EFFLAM
    END DO

DO K1=NDI+1, NTENS
    DDSDDE(K1, K1)=EFFG
END DO
DO K1=1, NTENS
    DO K2=1, NTENS
        DDSDDE(K2, K1)=DDSDDE(K2, K1)+EFFHRD*FLOW(K2)*FLOW(K1)
    END DO
END DO
ENDIF
```

Figure 6 - UMAT code

Then we calculate the jacobian matrix finally. Using the elastic, plastic strain and shift tensor and update and the state variable of eq_plas and shift tensor.

After writing UMAT we ran the simulation and noted the stress strain curve.

3.2 Summary

At first we created a simulation from the stress strain data provided. Then we used curve fitting to fit already available plastic curves. Origin was used for curve fitting. Then we fitted our own equation and used the data for simulation in abaqus. Lastly we used UMAT programming for simulation of linear hardening.
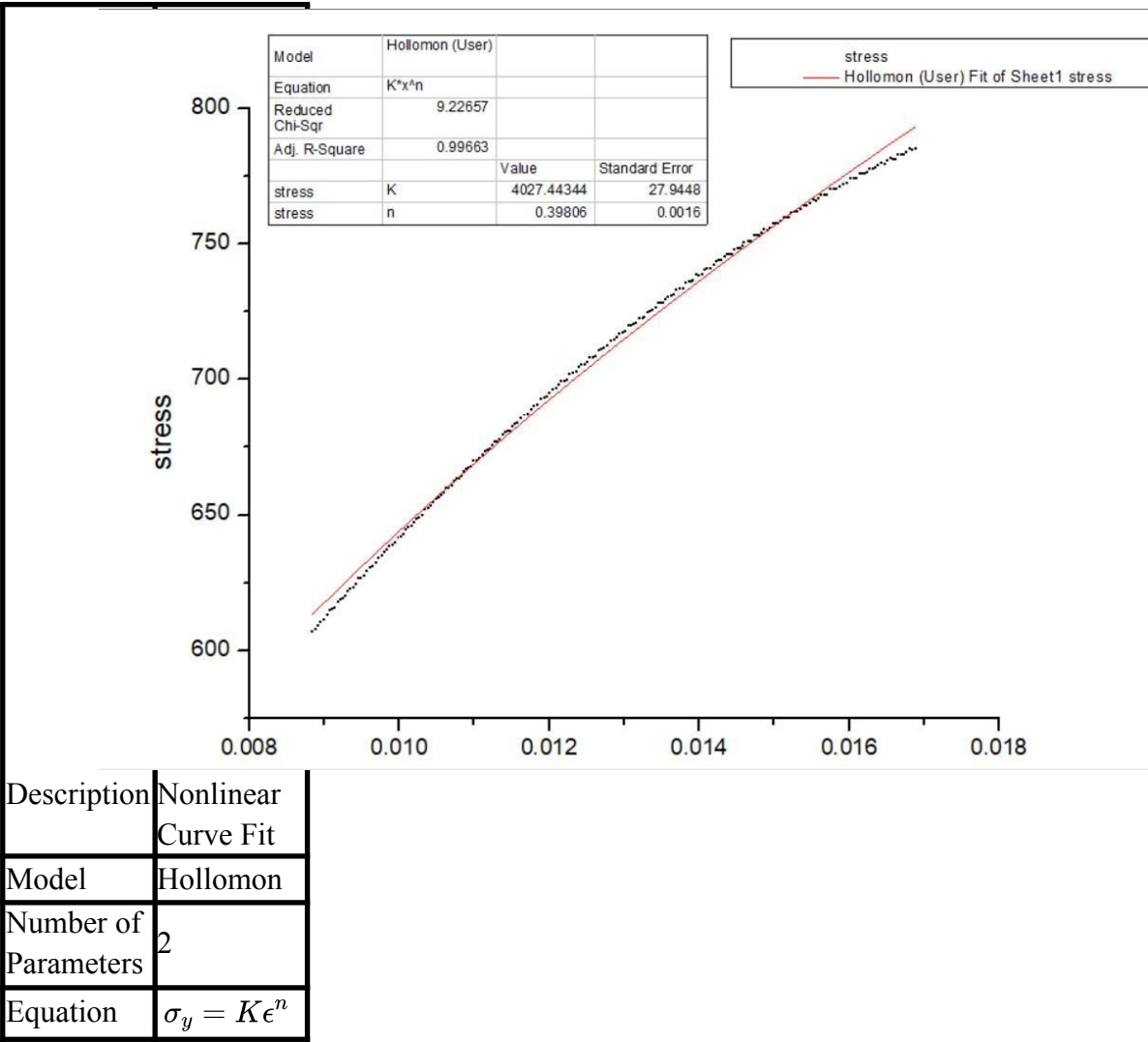
4. Results And Discussion

4.1 Models Analysis on Origin

We have conducted an analysis on several models and have interpreted the resulting data.

4.1.1 Hollomon Model

Table 4 - Hollomon Model fitting parameters

	Hollomon (User)		
Model			
Equation	K*x^n		
Reduced Chi-Sqr	9.22657		
Adj. R-Square	0.99663		
		Value	Standard Error
stress	K	4027.44344	27.9448
stress	n	0.39806	0.0016

Description	Nonlinear Curve Fit
Model	Hollomon
Number of Parameters	2
Equation	$\sigma_y = K\epsilon^n$

Parameters	Value	Standard Error
k	4027.44344	27.9448
n	0.39806	0.0016

4.1.2 Pickering Model

Table 5 - Pickering Model fitting parameters

Model	Pickering (User)			
Equation	A+B*x+C*ln(x)			
Reduced Chi-Sqr	74.03097			
Adj. R-Square	0.98945			
			Value	Standard Error
Stress	A		715.13792	1.7733
Stress	B		26872.84681	115.82103
Stress	C		23.91855	0.23366

Description	Nonlinear Curve Fit	
Model	Pickering	
Number of Parameters	3	
Equation	$\sigma_y = A + B$	
Parameters	Value	Standard Error
A	715.13792	1.7733
B	26872.84681	115.82103
C	23.91855	0.23366
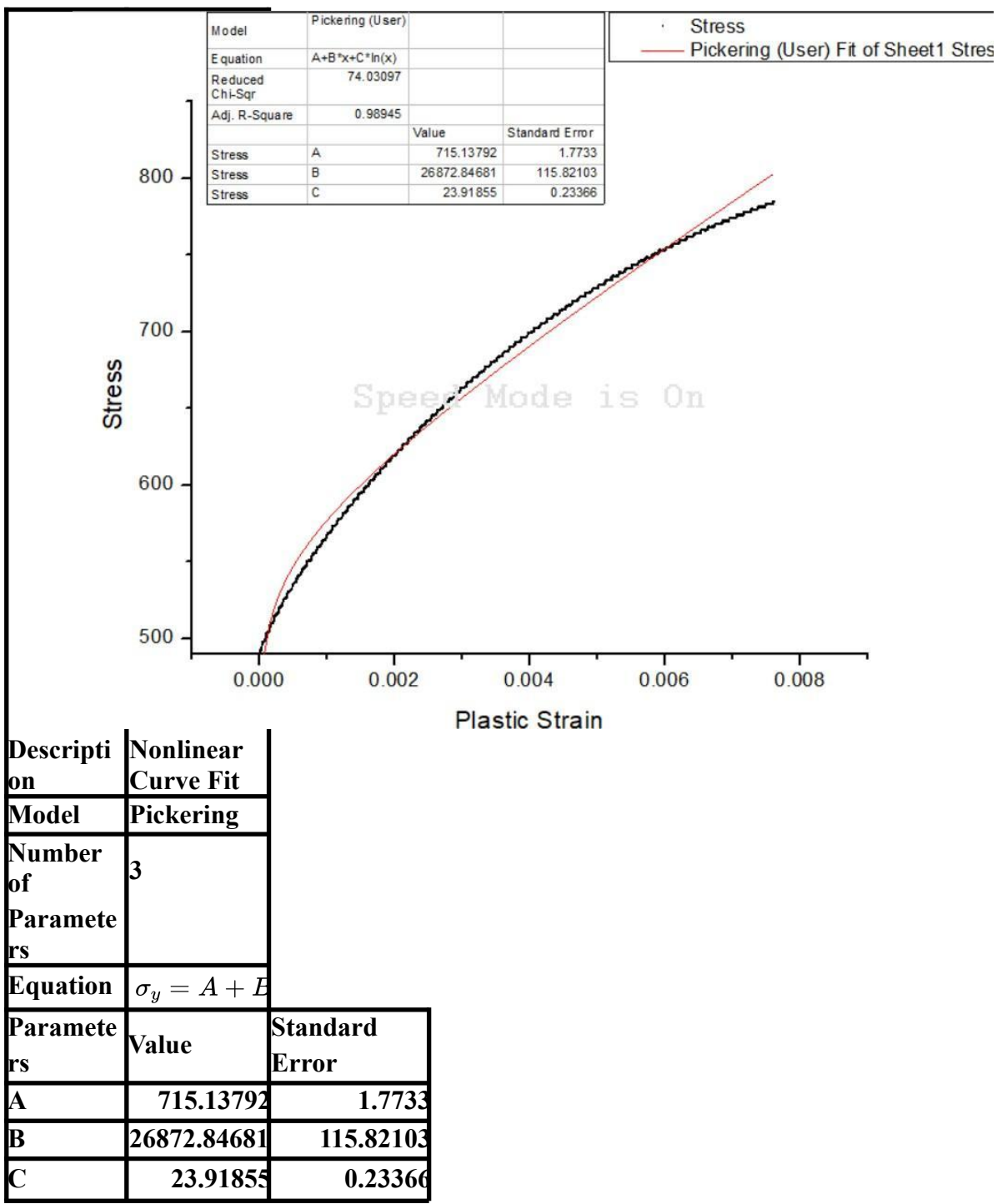
Figure 7 - Stress strain curve fitting pickering model

4.1.3 $\sigma_y = \sigma_{y0} + H\epsilon_p^n$
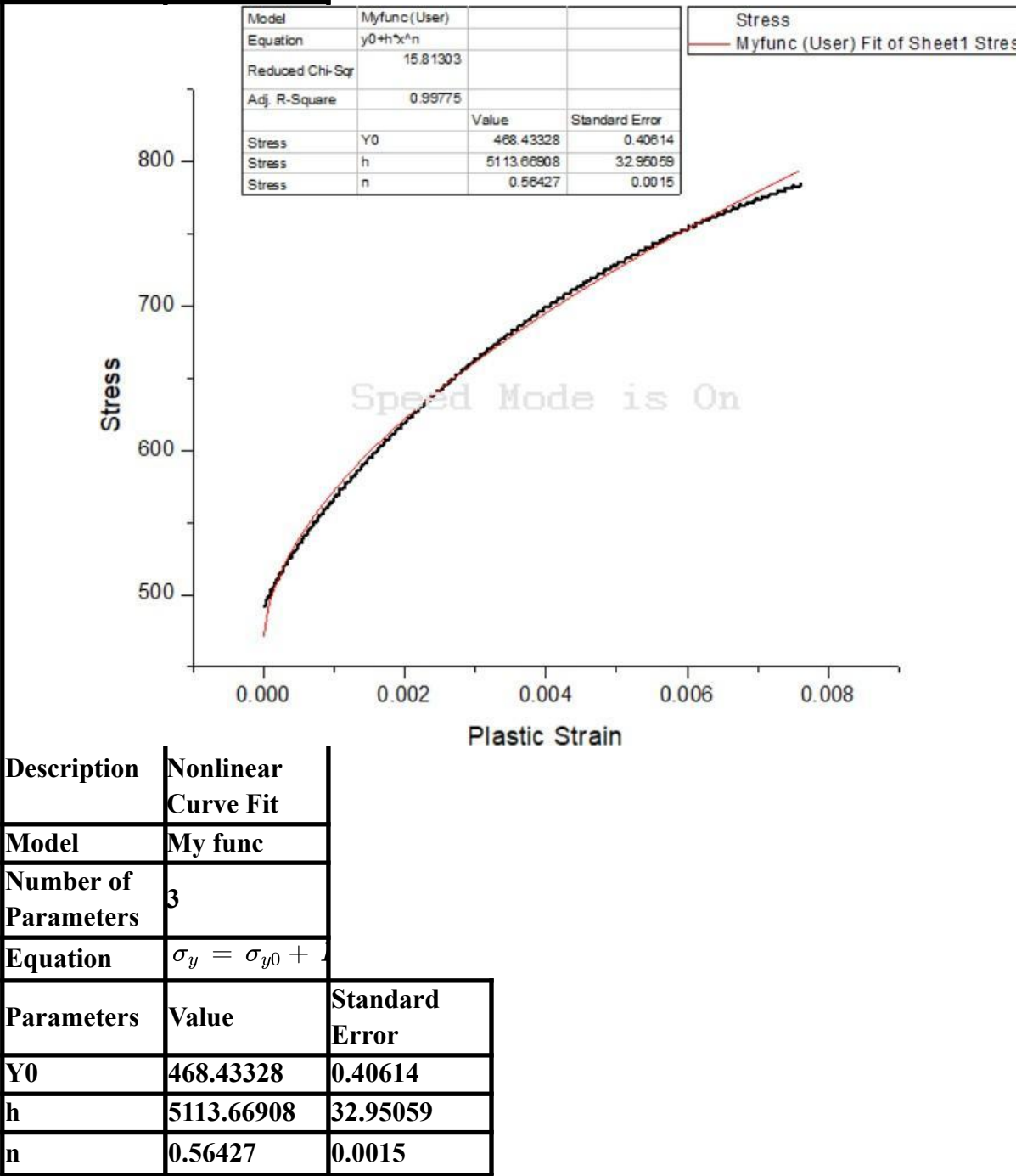
Table 6 - Generated Model fitting parameters

Model	Myfunc(User)			Stress
Equation	y0+h*x^n			Myfunc (User) Fit of Sheet1 Stres
Reduced Chi-Sqr	15.81303			
Adj. R-Square	0.99775			
		Value	Standard Error	
Stress	Y0	468.43328	0.40614	
Stress	h	5113.66908	32.95059	
Stress	n	0.56427	0.0015	

Description	Nonlinear Curve Fit	
Model	My func	
Number of Parameters	3	
Equation	$\sigma_y = \sigma_{y0} +$	
Parameters	Value	Standard Error
Y0	468.43328	0.40614
h	5113.66908	32.95059
n	0.56427	0.0015

Figure 8 - Stress strain curve fitting generated model

4.1.4 $\sigma_y = \sigma_{y0} + H\epsilon_p$

Equation	$\sigma_y = \sigma_{y0} + H\epsilon_p$	Value	Standard Error
Stress	Intercept σ_{y0}	535.368	0.45394
Stress	Slope (H)	37061.13	109.60078

Table 7 - Linear Hardening Model fitting parameters
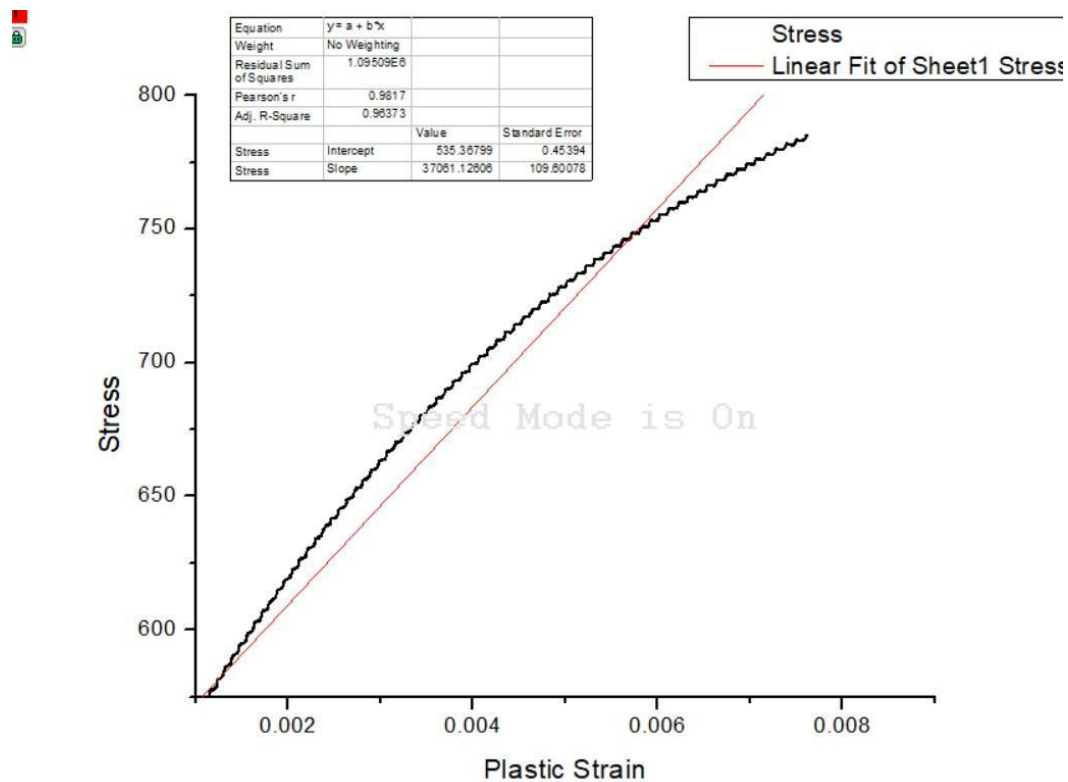
Figure 9 - Stress strain curve fitting linear hardening model
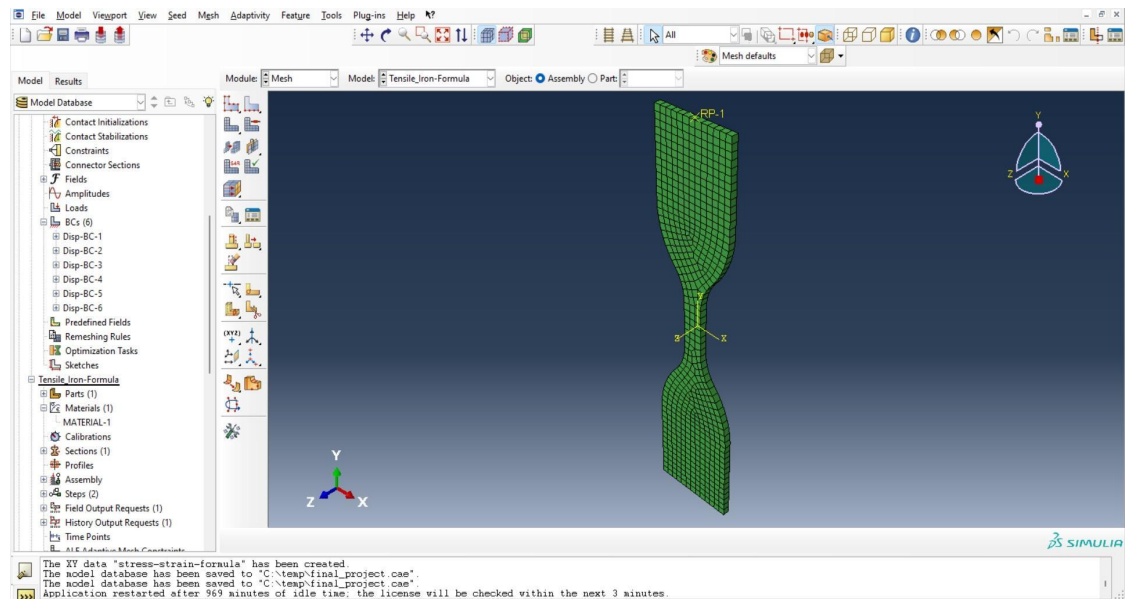
ABAQUS

Modeling of specimen

Abaqus Model

Figure 10 - Abaqus Model

Simulation Results

1. From data obtained from tensile test
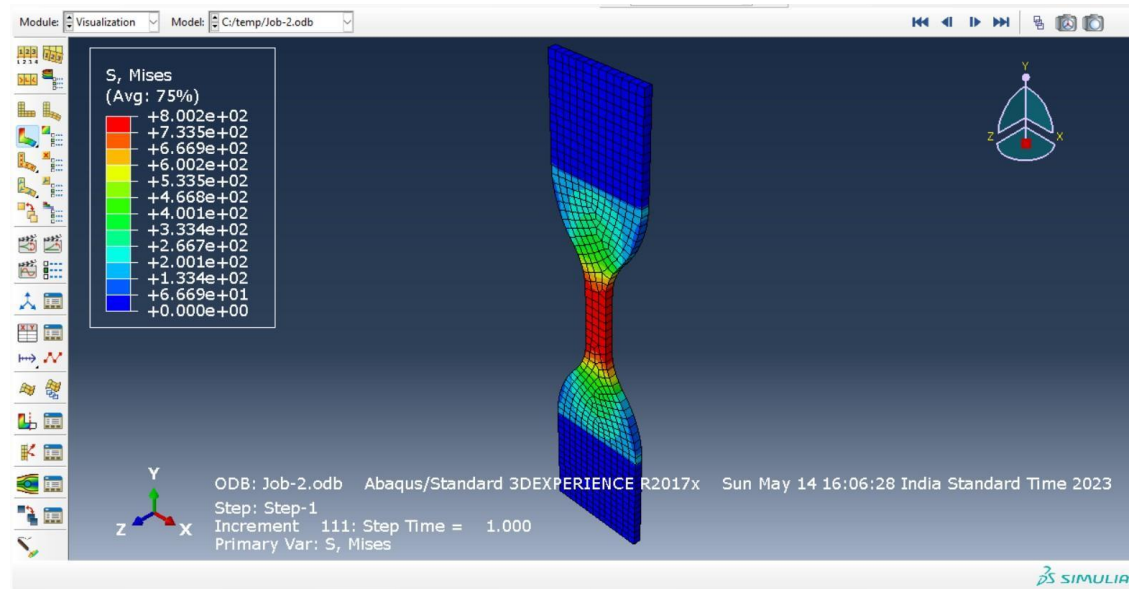
Figure 11 - Abaqus Model

stress - strain graph
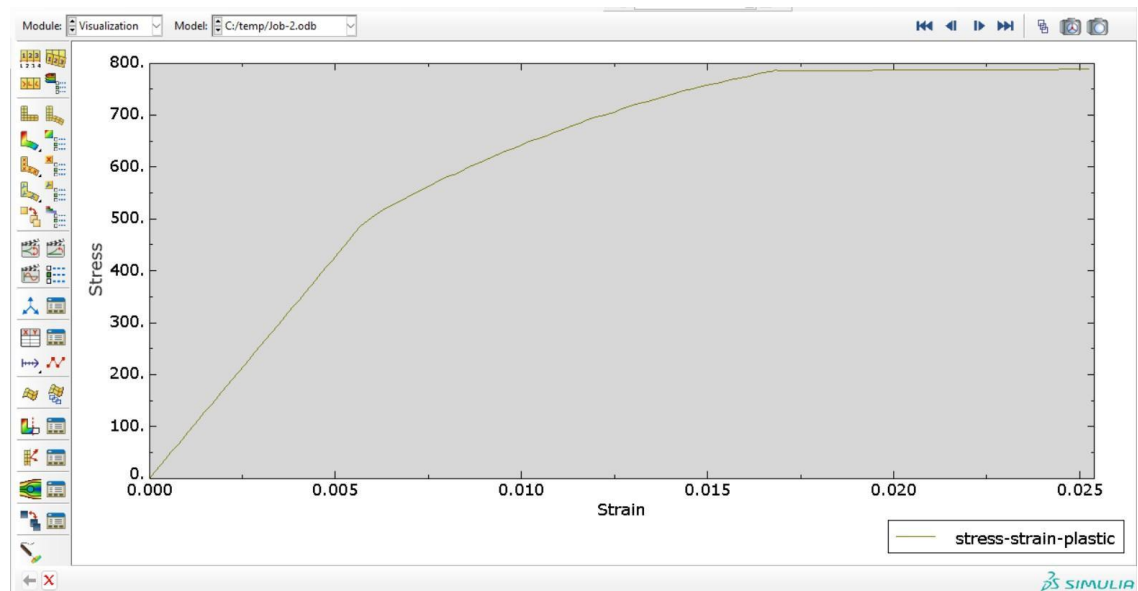
Figure 12 - Stress strain graph after simulation

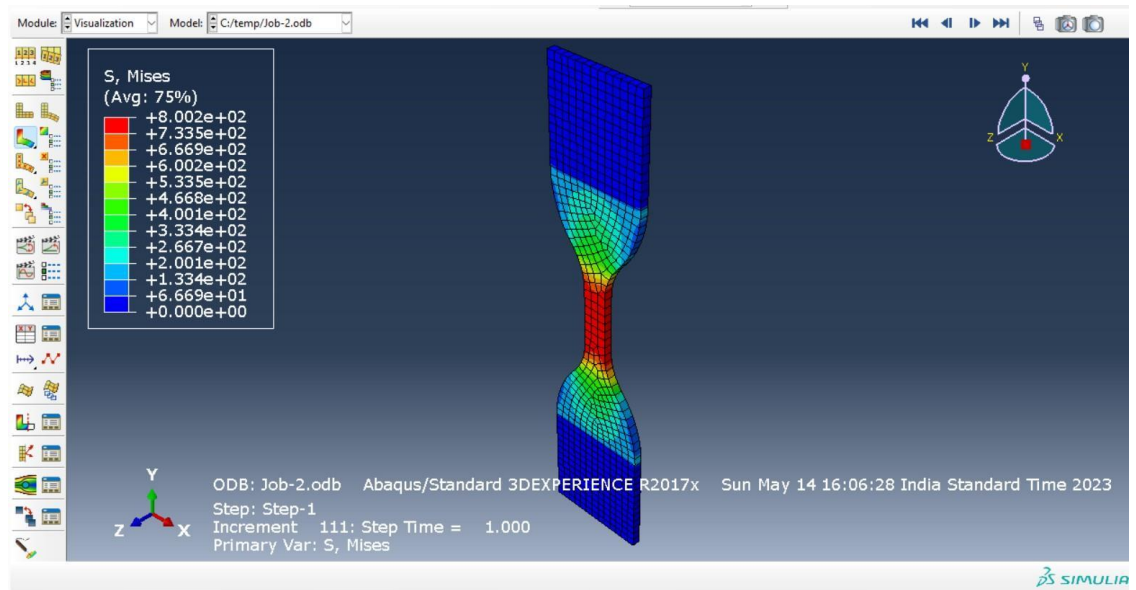## 2. Using stress strain data generated from using our model $\sigma_y = \sigma_{y0} + H\epsilon_p^n$
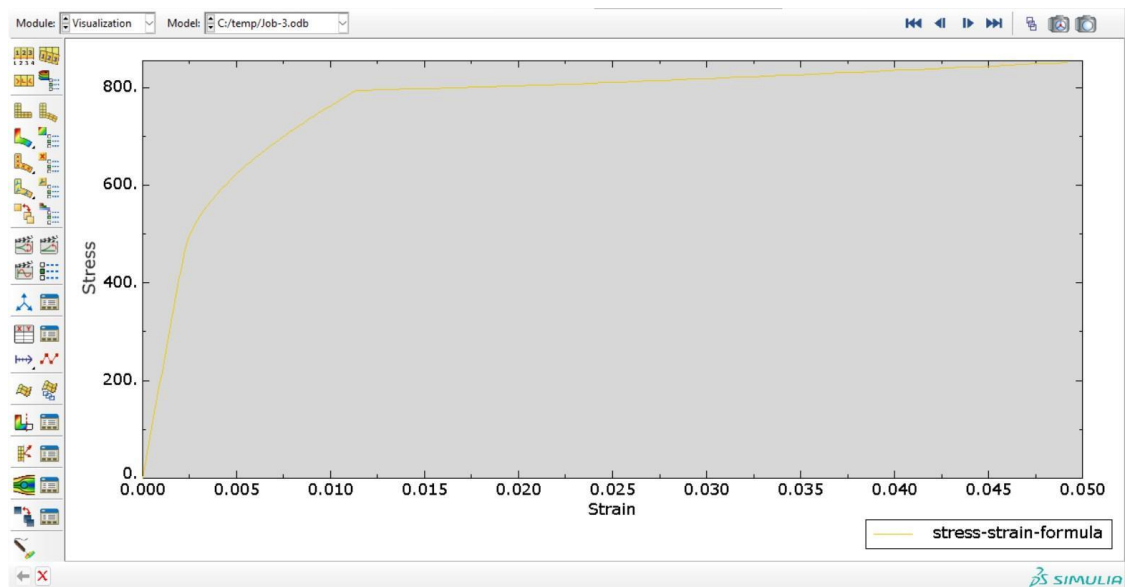

Figure 13 - Abaqus Model

Stress-strain graph

Figure 14 - Stress strain graph after simulation

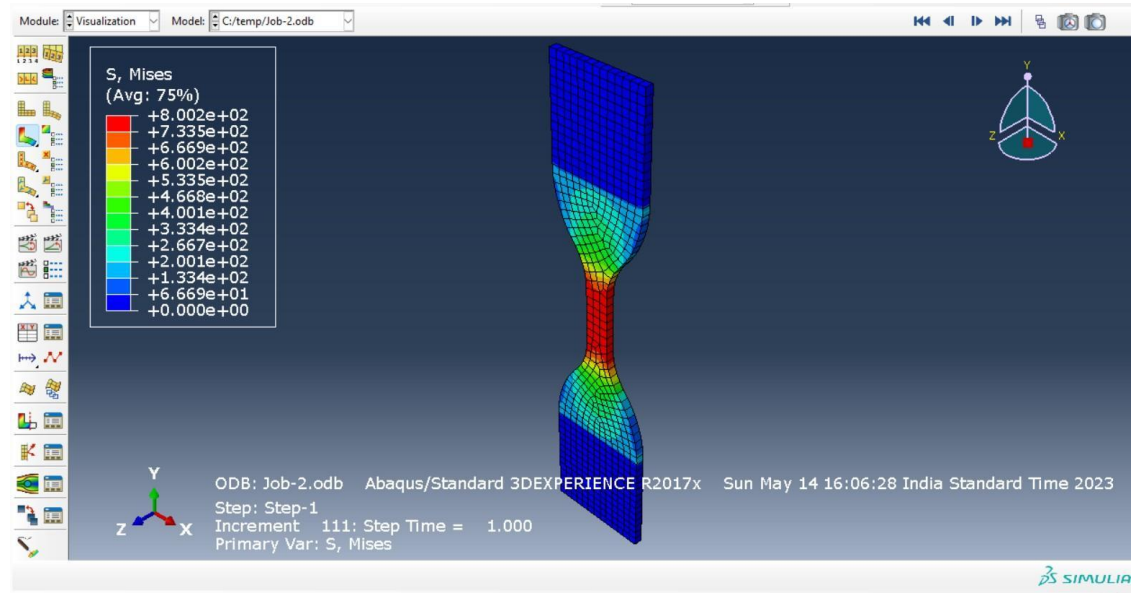### 3. Using UMAT isotropic linear plasticity model

Figure 15 - Abaqus Model
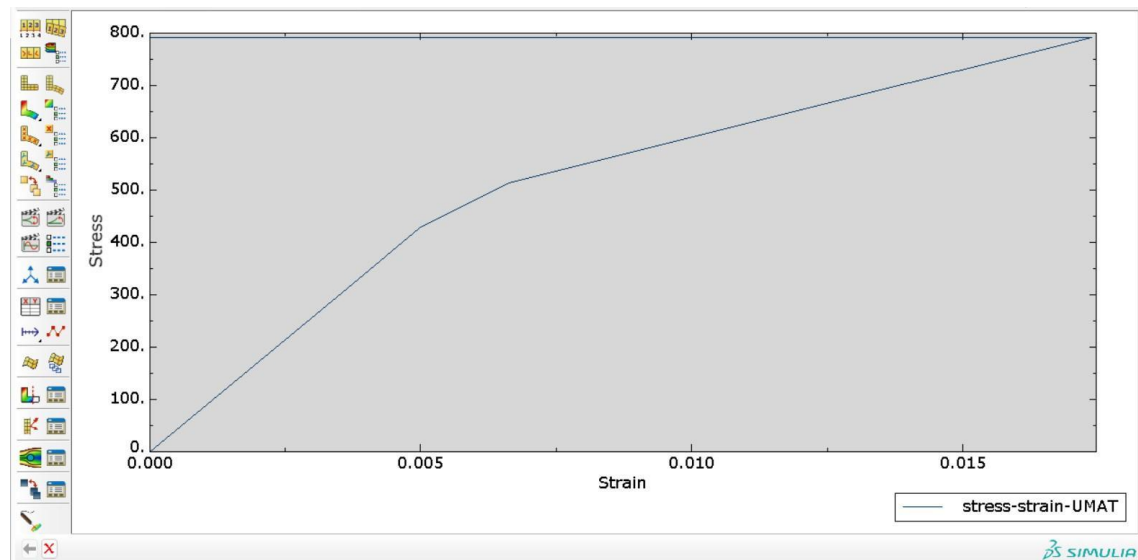
Stress strain curve

Figure 16 - Stress strain graph after simulation

5. Conclusions and Future Scope

5.1 Conclusions

A tensile test specimen was modeled and simulations were run on it by providing three different data. These data were either used from the original stress strain or form the modeled data. All the stress strain graphs were plotted.

There was not much deviation in the original plotted stress strain curve with the modeled stress strain curve.

The UMAT curve had a lot of deviation as it was based on linear hardening model.

5.2 Future Scope Of The Work

Since we were unable to model our equation in UMAT, in future we would try to learn more about plasticity theory and UMAT programming and try to model non linear terms in Abaqus. Moreover, trying to find a much better relationship between plastic strain and stress which would help in modeling a lot of different materials would also be helpful.
Trying to create a UMAT which can model a lot of materials very accurately would be most helpful for simulation.

6. References

1. V. Colla, M. DE Sanctis, A. Dimmateo, G. Lovico, A. Solina, and R. Valentini (2009) Strain Hardening Behavior of Dual-Phase Steels, Volume-40A, Pages 2557-2566
2. Courtney, Thomas (2005). Mechanical behavior of materials. Waveland Press, Inc. Pages 6–13
3. Beer, F.; Johnston, R.; Dewolf, J.; Mazurek, D. (2009). Mechanics of materials. New York: McGraw-Hill companies.
4. Dr.-Ing. Ronald Wagner. Abaqus UMAT
5. Wagner, Ronald. (2021). Plane Stress - UMAT for ABAQUS. https://www.researchgate.net/publication/356875238_Plane_Stress_-_UMAT_f o r_ABAQUS