UNIVERSITY OF THESSALY

SCHOOL OF ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING

Analysis of defects arising during extrusion process

by

PIPIGKAS KYRIAKOS

Submitted in partial fulfillment of the requirements for the degree of Diploma in Mechanical Engineering at the University of Thessaly

Volos, 2023

Approved by the Committee on Final Examination:

Advisor	Dr. Emmanouil Bouzakis, Associate Professor of Manufacturing Processes at University of Thessaly
Member	Dr. Aravas Nikolaos, Professor of Computational Mechanics of Structures at University of Thessaly
Member	Dr. Agoras Michalis, Assistant Professor of Nonlinear Composite Materials – Homogenization Theories at University of Thessaly

Date Approved: [Month dd, yyyy]

Acknowledgements

Before anything else, I would like to express my heartfelt appreciation to my supervisor Associate Professor Bouzakis Emmanouil for their invaluable advice and important insights throughout my thesis. Their commitment, support, and guidance have been essential to shaping my research.

I am extremely grateful to the academic members of the advisor committee Professor Aravas Nikolaos and Assistant Proffesor Agoras Michalis who have generously shared their knowledge and offered constructive input on my study. Additionally, I would like to acknowledge the department's stimulating academic environment, for which I am thankful.

Last but not least, I am appreciative to my family for their everlasting support, love, and encouragement. Their persistent support, faith in me, and sacrifices were the driving factor behind my academic achievement.

Analysis of defects arising during extrusion process

KYRIAKOS PIPIGKAS

Department of Mechanical Engineering, University of Thessaly

Supervisor: Dr Emmanouil Bouzakis
Associate Professor of Manufacturing Processes

Abstract

The thesis titled "Analysis of Defects Arising during Extrusion Process" is a comprehensive study of the extrusion process and the crucial defect of central burst. The first chapter provides a historical overview of the manufacturing process, including its evolution and significance.

The second chapter emphasizes on the mechanical behavior and the fundamentals of extrusion, includes cold extrusion warm and hot extrusion analysis, proceeds with the most significant parameters of the process and closes with mentioning the main defects.

The third chapter examines the use of Abaqus/Standard software for performing 3D extrusion analysis. This chapter presents the results of a detailed analysis using the software and helps the reader to further elaborate in the central burst defect.

The fourth chapter explores the central burst defect, a commonly observed extrusion defect. The chapter provides previous paper research of the defect and conducts an analysis using Abaqus/Explicit to understand its cause and impact.

The final chapter summarizes the results of the previous analysis and provides insights into further research that can be conducted to deepen the understanding of the central burst occurring in an extrusion process. The chapter concludes by emphasizing the importance of this research and its potential impact on the field of forming analysis.

Key words: Forming Analysis, Extrusion, Central Burst Defect, Abaqus/Standard, Abaqus/Explicit.

Table of Contents

Table of Figures

Chapter 1. Introduction

1.1. Motivation

Metal forming technology has a special position among all manufacturing processes because it enables the production of products with exceptional mechanical qualities and little material waste. In metal forming, the initial material has a relatively basic geometry and is deformed plastically in one or more procedures into a highly complicated shape. Tooling for metal shaping is often rather costly. Consequently, the technique is economically advantageous when a large number of components must be manufactured and when the requisite mechanical qualities of the completed product can only be acquired via a forming process.

Extrusion is the method of producing items with a predetermined cross-sectional profile by applying a force that causes the material to flow through a die. Its two primary benefits over other manufacturing methods are its capacity to make very complicated cross-sections and its ability to handle brittle materials, since the material is subjected to compressive and shear stresses. In addition, it produces a high surface quality and provides a great deal of design flexibility.

Central bursts, often known as chevrons, are the arrow-shaped internal flaws sometimes seen in cold extrusions or drawn wire. In cold extrusion, defects are often detected in the last, light phase of a multistep process. When central bursting occurs, a component's load-bearing capability is severely diminished.

Because to the danger of central burst, producers of important components, such as axle shafts and steering gear components, are obliged to install and maintain ultrasonic equipment for inspection of the produced components. While inspection may avoid the usage of components with central bursts, the rejection of extruded items increases manufacturing costs since both materials and labor are used to produce the rejected parts. Consequently, it is important to eliminate the possibility of central bursts.

1.2. Literature review

The majority of the millions of years that humans have occupied the planet have been spent foraging for sustenance with their bare hands. They also constructed primitive, simple buildings with their own hands, assuring some protection from the elements and the predators. Finally, they discovered how to start a fire, and developed basic hand tools that made survival somewhat simpler. These first tools included spears, stone axes, and hide scrapers. The first metal tools were the cast-iron ax, chisel, fishhook, and spike.

In general, manufacturing is the process of transforming raw materials into finished items. It includes the design of the product, the selection of raw materials, and the order of operational processes that will be used to make the product. The manufacturing sector is the foundation of every industrialized country. The degree of industrial activity in a nation is directly proportional to its economic health.

Manufacturing also includes operations in which the created product is utilized to generate further items. These products include enormous presses for forming sheet metal for automobile bodywork, equipment for producing bolts and nuts, and sewing machines for apparel production. Equally vital to the production process is the service and maintenance of equipment throughout its useful life. There are many different types of manufacturing processes, each with its own set of benefits and limitations. Some of the main categories of manufacturing processes include casting, machining, forming, joining, and finishing (Serope Kalpakjian, 2013, pp. 8-17) .

Pouring molten metal into a mold and allowing it to cool produces a solid object during casting. Machining is the removal process of material using cutting tools, resulting in a completed product. In contrast, metal forming operations include the application of force to a workpiece, resulting in a finished product with the required shape, Figure 1. Joining procedures include connecting two or more workpieces to create a single unit, while finishing operations entail treating a surface to enhance its look or performance.

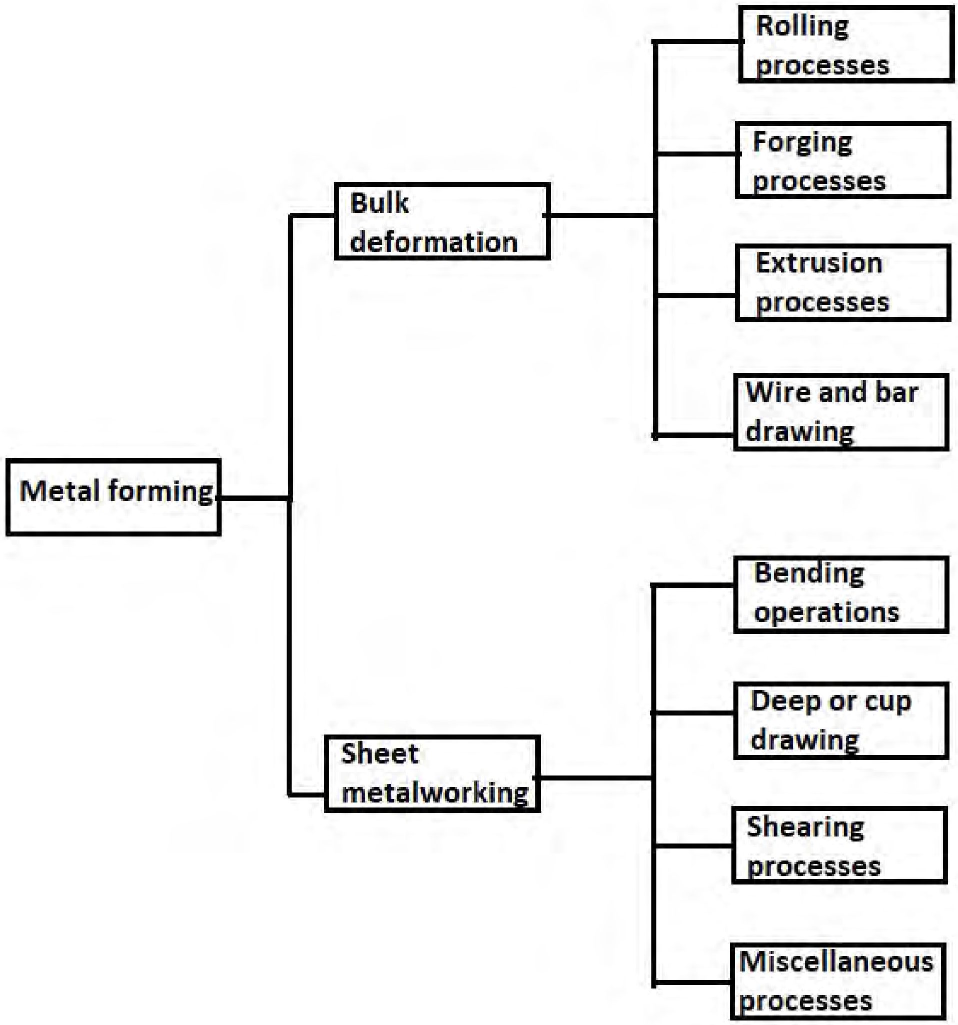

Figure 1: Metal forming workflow (Serope Kalpakjian, S. S. (2013a). Manufacturing Engineering & Technology (7th ed.))

Bulk deformation is the metal forming process in which a considerable change in shape happens due to plastic deformation in metallic components, while sheet metal forming is the metal forming activity in which the geometry of a sheet undergoes alteration when a force is applied. In bulk deformation, the area-to-volume ratio of work pieces is low, but in sheet metal forming, it is high.

Of all the shaping processes, metal forming is one of the most widely used techniques for creating metal parts and products. Metal forming involves the use of force to shape metal into a desired form, typically by using a mold or a die. Some of the main metal forming processes include forging, stamping, extrusion, and rolling.

Regarding all four of the above metal forming methods, manufacturers have to put excessive consideration on which technique will choose in order to successfully create the desired product. As important is the condition of the metal before the process, regarding the mechanical behavior, temperature and condition, it is also very crucial to consider the condition of the tools be used.

Concluding, metal forming is essential to the production of a wide variety of items for the industry and for everyday use. Different processes create different geometry and shapes, and as long the manufacturer considers the mechanical behaviors of the materials, keeps the error of tolerances at minimum and follows the regulations for safety in the materials, tools and human resources, then the products will be in their higher quality.

1.3. Diploma thesis organization

The rest of this thesis is divided into 4 sections occupying Chapters 2 - 6, respectively. Specifically, the organization of the thesis is as follows:

- Chapter 2 is an introduction to the extrusion process, including different types of extrusion and a discussion of basic equations used in the field.
- Chapter 3 presents a 3D analysis using Abaqus/Standard, providing a detailed examination of the extrusion process.
- Chapter 4 offers a historical view of studies on the central burst defect and presents a 2D approach in the central burst extrusion process analysis using Abaqus/Explicit.
- Chapter 5 discusses the results of the previous analysis and proposes a roadmap for further in-depth analysis of the central burst extrusion process.

Chapter 2. Understanding mechanical behaviors and defects in metal extrusion processes

2.1. Bulk deformation processes

Bulk deformation processes typically exhibit significant deformations and massive shape changes. However, the metal and the temperature at which the metal is formed influence the effect that strain rate has on flow stress. Also, the surface region to-volume of the work is somewhat small. The workpieces with this low area-to-volume ratio are referred to as bulk. For these processes, the shapes of the starting work are cylindrical billets and rectangular billets.

- **Rolling**- This is a compressive deformation process in which the thickness of a slab or plate is reduced by two opposing cylindrical tools called rolls. The rolls rotate so as to draw the work into the gap between them and squeeze it (Figure 2a) .
- **Forging**- The process of forging entails compressing a workpiece between two dies that are at right angles to one another in order to imprint the forms of the dies onto the product. While forging is often done hot working, there are various types of forging that are carried out in a cold environment (Figure 2b).
- **Extrusion**- By being pushed to flow through a die opening during this compression process, the work metal adopts the form of the opening to form the same cross-section. (Figure 2c).
- **Drawing**-In this forming process, the diameter of a wire or billet is reduced by pulling it through a die opening (Figure 2d).

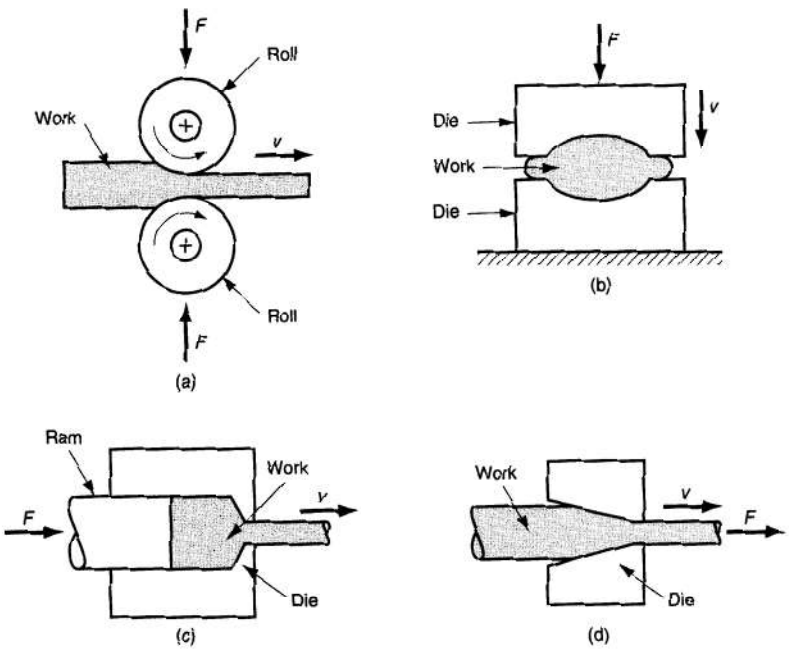

Figure 2: Basic bulk deformation processes (Serope Kalpakjian, S. S. (2013a). Manufacturing Engineering & Technology (7th ed.))

Forming and related operations on metal sheets, strips, and coils are all part of the process known as sheet metalworking. The starting metal has a high ratio of surface area to volume (Serope Kalpakjian, 2013, pp. 235-249). As a result, this ratio can be used to distinguish sheet metal processes from bulk deformation. Because the machines used in these operations are presses, the term "pressworking" is frequently used to refer to operations on sheet metal (other manufacturing processes also employ presses of various types).

2.2. Mechanical behavior

A force that is greater than the material's yield strength is required to plastically deform a metal. At the point when low measures of pressure are applied to a metal it will change its geometry somewhat, in correspondence to the power that is applied.

In essence, it will slightly compress, stretch, or bend. The amount will grow in proportion to the force that was applied. Additionally, when the force is released, the material will revert to its original geometry. Imagine stretching a rubber band, letting go, and then seeing it return to its original shape. Elastic deformation is the term for this.

A metal begins to undergo plastic deformation, rather than elastic deformation, once the stress on it exceeds a certain point as seen in the Figure 3 below. The material's geometric change during plastic deformation is no longer directly proportional to the stress, and the geometric change persists after the stress has been released, indicating that the material loses its shape.

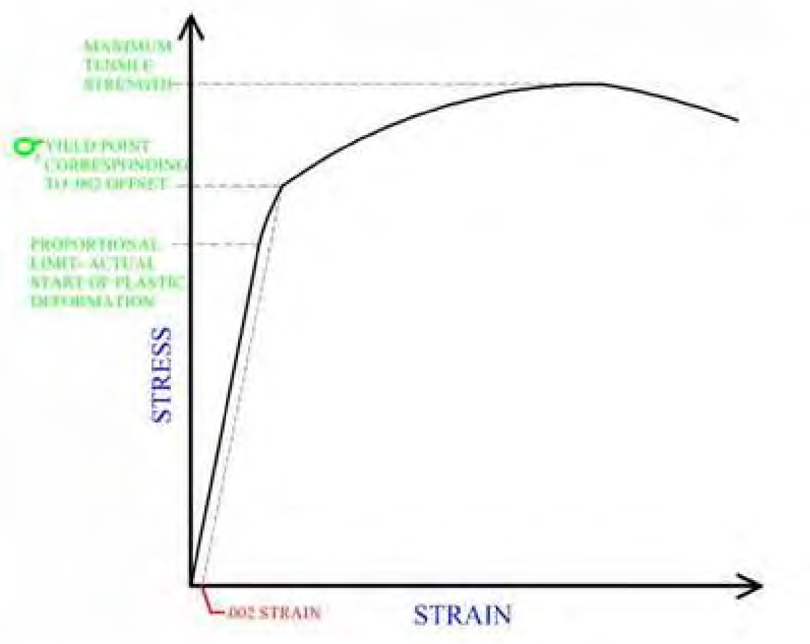

Figure 3: Stress-strain diagram for a typical metal (Metal Extrusion. (n.d.))

The proportional limit, which is difficult to precisely quantify, is the degree of stress given to a metal at which plastic deformation starts to happen. The yield point is commonly established using the 0.002 offset convention, which is assumed to represent the stress level at which plastic deformation (yielding) starts.

The stress-strain graph demonstrates that higher stress levels are required for a metal to continue deforming once it has reached its yield point and is plastically deforming. The degree that the metal is deformed plastically actually increases its strength. Work hardening or strain hardening are two names for this. As might be assumed, strain hardening plays a significant role in the metal forming processes. Despite the fact that it is often an obstacle that must be overcome, strain hardening is occasionally an essential stage in the production of stronger components.

During a metal forming process, it is crucial to recognize the force and power required to produce the desired deformation. A workpiece must be exposed to increasing stress in order to deform plastically, as seen by the stress-strain graph. Flow stress is the instantaneous force required at any step of the process to maintain the work material's yielding and flow. Strain may be considered as a function of flow stress. The value of flow stress may be utilised to analyse what is occurring at a given step of the metal-shaping process. The maximum flow stress is an important quantity in a variety of metal forming processes, as it indicates the force and power needs for the equipment required to execute the process. Determining the force required at the maximum strain of the material would enable one to determine the maximum flow stress.

Depending on the metal-forming process, the flow stress analysis may vary. For a process such as forging, the maximum flow stress value is of vital importance. For a process such as extrusion, in which the metal is continuously deformed and the various stages of deformation occur simultaneously, it is essential to analyse the mean flow stress value.

As seen in Figure 4, the rate of deformation is directly correlated with the amount of strain for all production-relevant metal-forming techniques. If the workpiece deforms more rapidly, the strain rate will increase. Both the particular procedure and the physical action of the equipment have a substantial effect on strain rate. The strain rate will influence the degree of flow stress. The relationship between strain rate and flow stress is dependent on the kind of metal and the temperature at which it is produced.

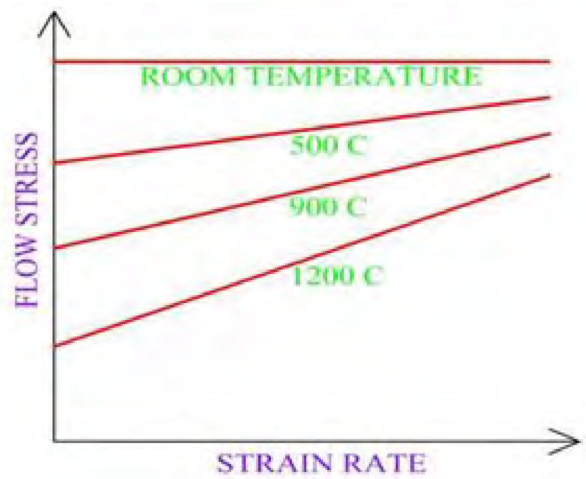

Figure 4: Flow stress vs strain rate for a typical metal at different temperatures (Metal Extrusion. (n.d.))

2.3. Extrusion process

Metal extrusion is a metal forming process in which a work piece, of a certain length and cross section, is forced to flow through a die of a smaller cross sectional area, thus forming the work to the new cross section (Metal Extrusion, n.d.). The length of the extruded part will vary, depending upon the amount of material in the work piece and the profile extruded. Numerous cross sections are manufactured by this method. The metal extrusion will have a consistent cross section over its whole length. Typically, work begins with a cylindrical billet, which is then shaped into a smaller-diameter round component, a hollowed tube, or another profile. Extrusion is more competent than other metal-forming techniques of achieving large geometrical changes and workpiece deformation. In the direction in which the work is extruded, the grain structure of the material is commonly extended, which is seen as advantageous.

Extrusion is often seen as a semi-continuous manufacturing process. The method is continuous since the same cross section will be created throughout its length. This length may be divided into several, independent sections. As the length of the extruded product is still restricted by the quantity of material in the workpiece, it is semi-continuous as opposed to completely continuous. At the conclusion of each cycle, it is necessary to reload the work item. Each cycle of a metal extrusion process produces a separate product, making it a discrete manufacturing approach. Like with other metal-forming processes, the research and improvement of this industrial process focuses heavily on forces and material flow patterns.

2.4. Advantages and limitations of cold extrusion, warm extrusion and hot extrusion

Hot forming or hot working refers to the process of shaping a metal over its recrystallization temperature. When it comes to enhancing the mechanical qualities of a material, hot working offers various benefits. Cast metal has voids and pores throughout. Hot working will press and rearrange the material to fill these voids. As molten metal solidifies, impurities tend to collect in huge numbers, resulting in the formation of solid inclusions. Because of these inclusions, the surrounding material gets weakened. During hot working, these inclusions break and distribute throughout the metal's mass. Typically, cast pieces have massive, uneven, column like grain structures. By heating a metal, its uneven structures are shattered and the bulk of material is recrystallized into a finer-grained structure. The mechanical qualities of the component, such as its ductility, strength, and impact resistance, are improved. Nevertheless, the bulk of metal extrusions in the manufacturing sector are performed on already hot-formed billets, so the material already has the mechanical benefits of hot forming. In addition to improving the physical qualities of the metal, hot working provides further benefits for industry applications. When a metal has attained its recrystallization temperature, it is easier to manipulate than when it is cold. As a metal's temperature increases, its tensile strength and ductility diminish, two properties that aid in its forming. Since strain hardening does not develop when metals are processed beyond their recrystallization temperature, considerable shape change is possible during hot working.

In addition to enhancing the metal's physical properties, hot working has other advantages for manufacturing processes. A metal that has reached its recrystallization temperature can be manipulated more easily than a cold metal. When the temperature rises, the metal's strength and ductility decrease, two characteristics that help the metal form.

As strain hardening does not occur when metals are handled beyond their recrystallization temperature, hot working allows for substantial shape change. The oxidation that occurs on the outermost layer of hot-worked metals is one of the key drawbacks of this process. As a result, an oxide scale layer forms on the workpiece's exterior surfaces. Scale can influence surface completion and precision of the part as well as expanding grating and wear at pass on metal points of interaction. When compared to a cold-forming manufacturing process, the disadvantages of a hot-forming process include heating to and maintaining high working temperatures, decreasing tolerances, and increasing die wear.

In the industrial process known as "warm forming," a metal billet or sheet is heated to a temperature between 650° and 900° degrees Celsius, still below its recrystallization point, and then shaped by applying pressure. It's a common compromise between the low cost and low precision of cold forming and the high precision and high cost of hot forming. Although hot forming is more precise than warm forming, it may lead to oxidation and other quality concerns if done at too high of temperatures.

Compared to hot extrusion, cold extrusion has the following advantages: no need of preheating, increased production, absence of oxidation or scale development on surfaces, enhanced surface texture, and capabilities to reinforce the component via strain hardening. Identical to other methods of hot forming, heat transfer between the workpiece and the cooler die surfaces poses a manufacturing challenge in hot extrusion. The temperature gradient can be reduced by preheating the extruding die to mitigate this issue. Additionally, lubricants aid in reducing heat transfer between the part and the mold. Isothermal extrusion, which is conceptually similar to isothermal forging, may be utilized with materials that are particularly challenging to extrude. In these occurrences, the mold is kept up with at, or somewhat beneath, the temperature of the work during the whole cycle.

Below the recrystallization threshold, between 400° and 800° degrees Celsius is commonly used for warm extrusion. It's ideal for materials that can't be cold extruded since it minimises the effort needed to distort the metal, enables better accuracy and tighter tolerances than cold extrusion, and avoids some of the quality difficulties that might come with hot extrusion.

In summary, each of these metal forming processes has its own unique advantages and limitations, and the choice of which one to use will depend on factors such as the desired shape and complexity of the component, the type of material being used, and the required production volume and speed.

2.5. Metal flow during extrusion process

A metal extrusion method involves forcing metal from a workpiece with a certain cross section to flow through a die with a smaller cross section, producing an extruded item. It's crucial to comprehend how the material moves when the component is being formed. It resembles fluid moving from one smaller channel into another in some ways. As it moves toward and through the die, the metal is deformed and forced to flow together. When the piece passes through the die, the outer layers deform more than the inner layers. In the outside parts that are further from the central axis, metal flow characteristics will be more turbulent, and there will be more material displacement. The material that is closest to the center will move through the mold more quickly, resulting in a higher velocity in comparison to the die. When employing square die, which are die with 90 degree angles, portions of the material near to the mold opening but next to the die may not move. These regions, also known as dead zones or dead metal zones, are identified by the stalling of metal flow. The material may not move in some areas that are near to the mold opening but far from the die. These regions, also known as dead zones or dead metal zones, are identified by a stagnation in the flow of metal. It should be noted that the material will undergo a certain amount of shearing at the dead zone interfaces and between layers.

2.6. Extrusion parameters

When making extruded sections, a metal extrusion die needs to have certain mechanical properties. The extrusion die must be sturdy and hard enough to withstand the high stresses caused by the manufacturing process without losing their dimensional accuracy (Figure 5). Additionally, they must resist wear, which is always a problem when extruding large quantities of metal. Die for hot extrusion need to be resistant to heat and able to keep their hardness and strength even at high temperatures. For metal extrusion molds, a common type of material is tool steel. Coating extruding dies can make them more resistant to wear. Since carbides are resistant to wear and can provide precise part dimensions, they are sometimes used as a mold material.

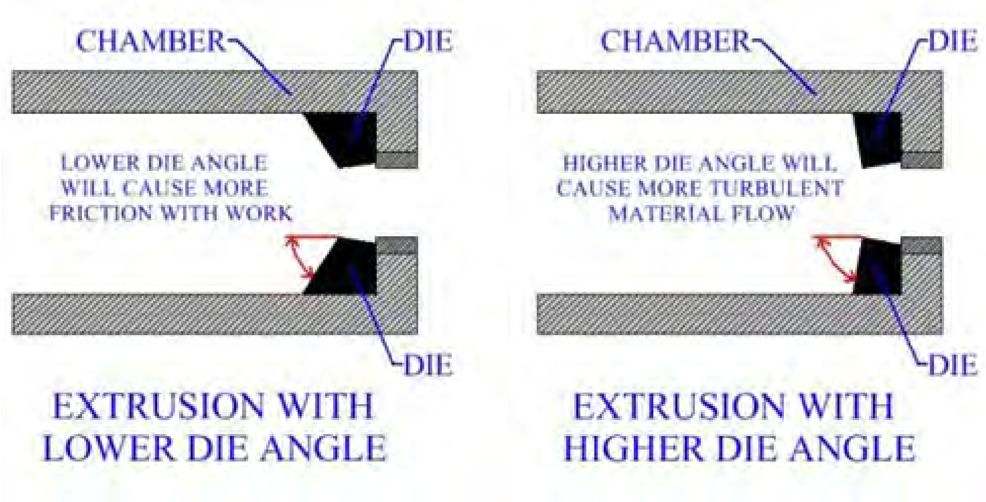

Figure 5: Effect of die angle during extrusion (Zeynep Parlar, 2015)

Because it has a significant impact on the flow of material, the angle of the extrusion die is an essential aspect of the manufacturing process. With different die angles, the force required to form a particular cross section, will vary. The work-die interface will experience more friction if the angle is lower. A factor known as friction raises the amount of force required to extrude a component. More material movement is caused by high die angles, especially in the outer regions away from the center. The metal flow experiences more turbulence as a result of the increased metal displacement. The amount of force required for the operation is also increased when the flow has more turbulence. A metal extrusion process must be designed with all factors taken into account.

The ideal die angle is one that lies somewhere in between the extremes of friction and turbulence—lower die angles have more extreme friction, while higher die angles have more extreme turbulence. For any metal extrusion process, the influence of other operational factors like temperature and lubrication makes it difficult to pinpoint the ideal die angle. Based on all of a given operation's considerations, the manufacturing engineer must try to provide the best angle.

The ideal die angle is one that lies somewhere in between the extremes of friction and turbulence—lower die angles have more extreme friction, while higher die angles have more extreme turbulence. For any metal extrusion process, the influence of other operational factors like temperature and lubrication makes it difficult to pinpoint the ideal die angle. Based on all of a given operation's considerations, the manufacturing engineer must try to provide the best angle.

The extrusion pressure rises as ram speed increases. Low ram speeds, on the other hand, cause the billet to cool, which causes flow stress to rise.

- The impact of a slow extrusion speed on the billet's cooling increases with the temperature of the billet.
- As a result, high-strength alloys necessitate high extrusion temperatures and high extrusion speeds.
- The temperature rise caused by deformation is also greater at high extrusion speeds.
- Selecting the best combination of extrusion speed & temperature is determined by trial & error for each alloy and billet size.
- For a given extrusion pressure the extrusion proportion which can be gotten increments with expanding temperature.
- High pressure allows for a large extrusion ratio at a given temperature.
- The temperature at which melting is about to take place is what determines the maximum billet temperature.
- The speed of the extrusion and the extrusion ratio determine the temperature rise during the extrusion.

During the extrusion of a part, lubrication is applied to the work-mold surfaces to facilitate metal flow. Parts are extruded in the manufacturing industry using a variety of special lubricants, including soaps, oils, graphite immersed in oil, and others. The tendency of some materials to stick to the tooling can present a challenge. A softer metal can be used as lubricant to prevent sticking. The work will be encased in the softer metal in this instance. When extruding tougher materials, molten glass is frequently used as an efficient lubricant in manufacturing practice, particularly during high-temperature processes.

The following are the requirements for a lubricant in hot extrusion:

- The amount of possible deformation increases as the billet's pre-heat temperature rises.
- It must be stable enough to withstand high working temperatures without breaking down.
- It must have a low shear strength. “Molten Glass" is the most frequently utilized lubricant for nickel-based alloys and steels.

The "Ugine-Sejournet Process" is a method in which molten glass is used as a lubricant. The steps are as follows:

- In an inert atmosphere, the billet is heated.

- Prior to entering the extrusion container, it is coated with glass powder.
- The glass coating reduces heat loss from the billet to the container wall and other tools by acting as a lubricant and a thermal insulator.
- Between the extruder and the die, the glass film is about 25 microns thick.

Temperature, ram speed, and the best lubricant interact as follows:

- Because of the low initial extrusion pressure, the lubricant is thick if the ram speed is too low. The glass reservoir soon runs dry as a result. The cost of lubricant goes up as a result.
- The glass film becomes too thin, and friction increases when the ram speed is too high.

Important lubrication requirements:

- The grease film should be finished and nonstop to find true success.
- Shear zones in metal are created by any gaps in the film, which eventually result in surface cracks.

Lubrication: (Glass)

- Needs to be kept at a high temperature and under a lot of pressure.
- Al, a low-strength alloy, doesn't need to be lubricated.
- The non-uniform deformation of metal results in a wide range of heat treatment responses.
- Effect of temperature, pressure & strain rate on the allowable working range or interdependence of extrusion speed & temperature. There will be a maximum amount of deformation on the workpiece at a given working pressure and temperature. As pre heat temperature of billet increases, the flow stress falls & therefore the amount of possible deformation increases.

2.7. Deformation behavior

1. Homogeneous Deformation: (Low friction coefficient)

Figure 6(a) indicates homogeneous deformation in direct extrusion Figure 6(d) shows homogeneous deformation in indirect extrusion (Serope Kalpakjian, 2013, pp. 360-380).

The following conditions are favorable for homogeneous deformation:

- Low container friction
- Well lubricated billet
- Hydrostatic extrusion conditions
- Indirect extrusion process.

2. Deformation with elevated friction coefficient between billet and container wall

Figure 6(b) below indicates increased container wall friction.

• This is indicated by severe distortion of grid pattern at the corners of the die due to a “dead zone.”

• The dead zone consists of stagnant metal which does not undergo any deformation.

• The grid elements at the center of the billet undergo pure elongation into the extruded rod.

• The grid elements near the sides of the billet undergo shear deformation

• The shear deformation requires additional energy called “redundant work”. This work is not related to metal working from billet to extruded product.

3. Deformation with very high friction coefficient

The condition of high friction at the container-to-billet interface is depicted in Figure 6(c) below.

• The center is where the metal flow is concentrated.

• High friction results in the creation of an internal shear plane.

• When a cold container cools the billet surface, this also happens. This is due to the fact that, at a low temperature, the flow stress at the sides of the billet increases in comparison to the flow stress in the center of the billet.

• Under such sticking conditions, a shear zone forms along which the metal separates internally.

• Under this circumstance, the extruded product consists of fresh, clean metal, and the billet's outer surface remains in the container.

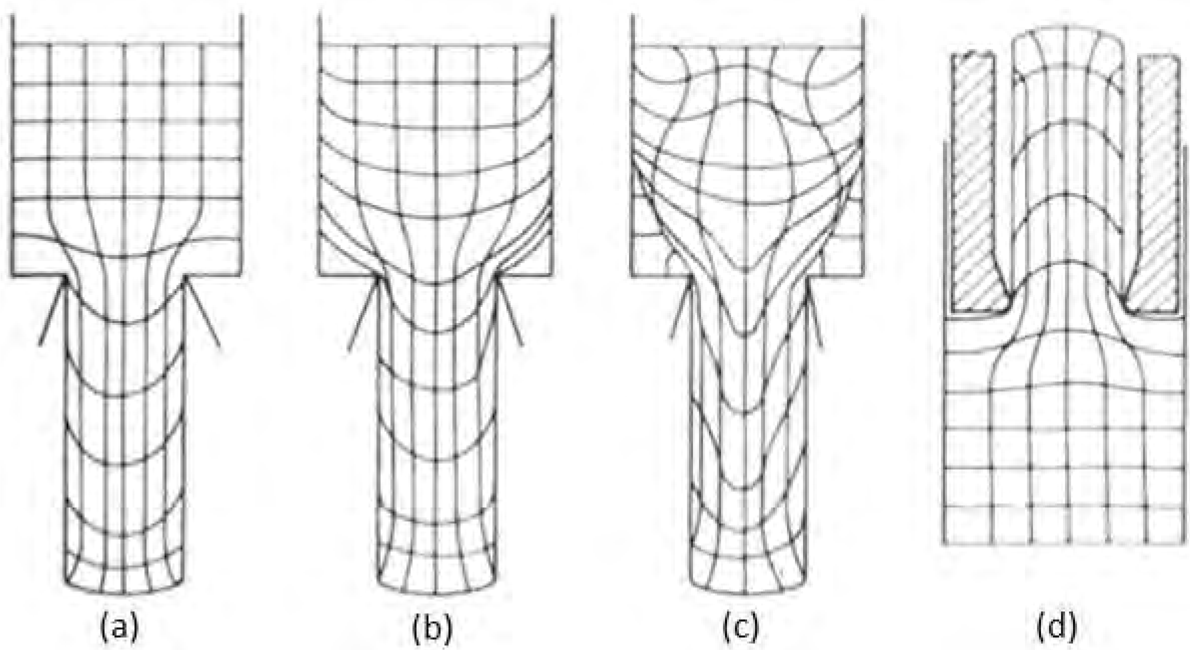

Figure 6: Different grid patterns during deformation in extrusion (elearningatria.wordpress.com)

2.8. Defects in extrusion

- Laminations of glass or oxide into the extrusion's interior Cause: Improper lubrication technique. Solution: use the best ram speed and provide the best lubrication on the billet's exterior.
- Extrusion defect: The final third of the extrusion may contain oxides and other impurities, making it unusable due to its poor mechanical properties. In the extruded product, this results in the formation of an "annular ring of oxide." Cause: Due to friction, the metal in the middle (thirds) of the billet moves faster than the periphery. As a result, it is first extruded. When there is friction between the container wall and the billet, this tendency of extrusion defects gets worse. Solution: The final third of the billet is left out without being extruded. However, this is financially impossible. A "follower block" is typically used instead. The billet is scalped or scraped by this block, which has a diameter that is slightly smaller than the containers, and the oxide layers in the container are left behind.
- Funnel or axial hole: It is an axial hole in the back finish of extrusion. Cause: During the extrusion of the final quarter of the billet, there is a rapid radial flow of metal. Solution: inclining the ram's face at an angle to the axis of the ram.
- Surface cracking: It appears as fir-tree cracking or a rough surface. Cause: i) Extrusion's longitudinal tensile stresses as it moves through the die. ii) A very high ram speed for the temperature that was specified. Solution: making use of the optimum ram speed, heating the container, and billet temperature.
- Center Burst, also known as "chevron cracking": low friction in the die's deformation zone and low extrusion ratios. Solution: improving the product by increasing the friction at the tool-billet interface.
- Variation in the properties and structure of the hot worked: This refers to the extruded product's non-uniform properties, which vary from front to back.

2.9. Main defects

Center cracking, **chevron cracking**, **arrowhead fracture**, and **center burst** (Figure 7, Figure 8a) are all terms used in the manufacturing industry to describe this kind of defect. Cracks develop along the central axis of the extruded section as a result of stresses breaking the material within the work piece as it is being extruded through the die. Because it occurs within the component's material, center cracking is a difficult defect to identify. A metal extrusion with a center burst is shown in Figure 7, the part has been cut in half to show the defect.

Understanding the flow of metal during profile extrusion is essential to comprehending the causes of center cracking. As was mentioned in the section on material flow, the metal extrusion's outer regions will experience more deformation, material displacement, and turbulent flow than the central regions. It is crucial to note the distinction between the metal movement in the outer regions and the central region. High stresses will develop within the material if the material displacement in the outer regions is significantly greater than the material displacement in the central region. Stresses will increase in proportion to the region-to-region variation in metal flow characteristics. Material breakage in the form of internal cracks will occur if the stress level becomes excessive.

The choice of die angle will play a major role in manufacturing practice in preventing center cracking. In a metal extrusion, high die angles will favor center cracking more than low die angles. The reason for this is that the work's outermost regions produce more flow turbulence. The likelihood of center cracking will generally decrease with higher fiction. Cracks will spread over inclusions, so the more inclusions there are in the work material, the more likely it is that this metal extrusion defect will occur. Center cracks are less likely to develop at lower extrusion ratios than at higher ones.

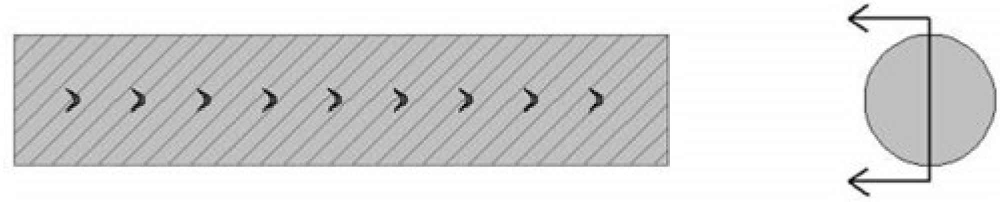

Figure 7: Center cracking in an extruded part (Metal Extrusion. (n.d.))

Piping, Figure 8b, also known as tailpipe or fishtailing, is a common defect that occurs when sections are produced through direct extrusion. Using a dummy block and properly preparing the work's surface can help avoid piping. The work material has piping at the end that is opposite the die. During the extrusion process, improper metal flow results in piping. At the work's conclusion, piping appears as a material-free void in the shape of a funnel. In any forming process, as previously stated, metal flow is an extremely significant factor to take into account. Creating a smooth metal flow during the extrusion process is the most effective method for resolving a piping issue. At the die-work interface, friction and thermal gradients must be observed and their cumulative effect on metal flow during the process must be determined. To ensure the smoothest possible material flow during the metal extrusion process, the manufacturing engineer must control the various process variables.

Surface Breakage, Figure 8c, on a metal extrusion, refers to damage to the component's surface. The majority of surface defects are cracks that, in varying degrees, extend from the surface into the component material. Typically, these cracks occur along the metal's grain boundaries. In the production of metal extrusions, excessive stresses on the material's surface are the primary cause of surface cracking defects. During the production of an extruded section, friction plays a significant role in controlling surface breakage. Surface cracking will thrive in an environment with more friction. An increased die angle and lubrication both have the potential to reduce friction.

While planning a metal extrusion producing process, it is vital to adjust all variables like grinding. Lower friction might make better circumstances at the surfaces between the work and pass on. However, center cracking may occur in a more turbulent outer flow if friction forces are too low. The hardness of the work stock material is another important factor in surface breakage defects. Another thing to think about is the speed at which the part is extruded. Conditions that are more conducive to material breakage will be created at higher extrusion speeds.

Surface cracking can be brought on by conditions in which the work material sticks to the extrusion die. In many different kinds of manufacturing operations, work adhering to tool surfaces can occasionally be a problem, particularly with certain materials. When the extrusion adheres to the die, pressure builds up behind the material frequently. The work comes loose with sufficient force, resulting in metal cracks. After a brief forward movement, the component once more adheres to the die. While the part is being extruded, this cycle continues to run itself. Around the part's peripheral, a series of cracks will appear at various intervals. This particular manufacturing defect is referred to as a "bamboo defect" because it resembles the appearance of a bamboo tree in some way.

A metal extrusion's surface cracking does not necessarily have to be caused by mechanical forces, thermal stresses frequently act as the primary cause. The extrusion's surfaces may rapidly lose heat if there are significant thermal gradients between the work and die interface. Thermal gradients at surfaces can be reduced with heated die and lubrication. Taking into account the operational variables, the work billet should be heated to the optimal temperature. The control of this kind of metal manufacturing process's thermal characteristics depends heavily on the speed of the extrusion and friction. Not only can faster extrusion speeds produce more heat but also more mechanical forces. The work material may crack as a result of the process's increased heat generation.

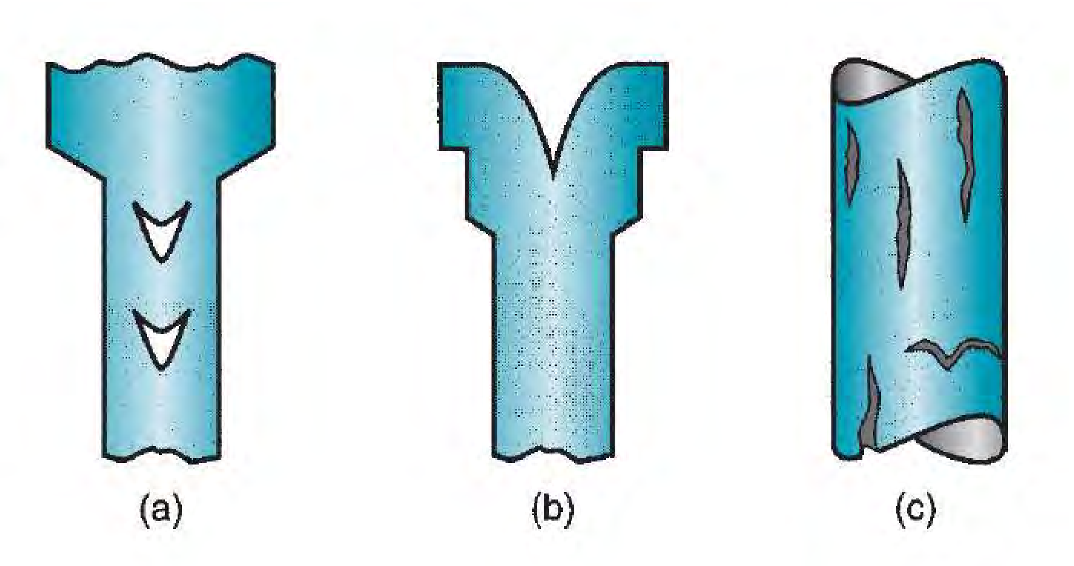

Figure 8: a) Central cracking, b) Piping, c) Surface breakage (Groover P. Mikell, Fundamentals of Modern Manufacturing: Materials, Processes, and Systems,2010)

Chapter 3. Three-dimensional analysis of extrusion process

Abaqus is a commercial finite element analysis software suite, with Standard, Explicit, and CAE constituting the software. Abaqus/Standard is a general-purpose solver that solves finite element analyses using a typical implicit integration technique. For highly nonlinear transient dynamic and quasi-static simulations, Abaqus/Explicit employs an explicit integration technique. Abaqus/CAE offers an integrated environment for modeling (preprocessing) and visualizing (postprocessing) analytical outputs.

The automobile, aerospace, and industrial product industries employ Abaqus. It is well-liked by academic and research institutes owing to its extensive material modeling capabilities and its adaptability. Abaqus also offers a variety of multi - physics features, including as coupled acoustic-structural, piezoelectric, and structural-pore capabilities, making it an ideal option for simulations at the production level when many fields must be connected.

Detailed simulations of the extrusion process may be carried out with the help of Abaqus CAE. It models the metal's behavior as it is passes through a die using the finite element technique, accounting for the billet's temperature-dependent behavior and its interactions with the die. Engineers may learn more about the stresses and strains inside the billet, the deformations that take place, and the temperatures that develop by utilizing it throughout the extrusion process.

Modeling the extrusion process in three dimensions in Abaqus CAE provides a thorough grasp of the procedure. The billet is simulated as solid homogenous in three dimensions, and the die as a set of surfaces that move and interact with the billet. In this case, factors like friction, heat transmission, and material deformation between the billet and the die are accounted for in the study. Stress contours, displacement plots, and temperature plots are just some of the ways the studys findings may be shown to aid in the extrusion process optimization and the avoidance of faults like central burst.

3.1. Part design and preparation for extrusion

To perform a simulation of the 3D extrusion process using Abaqus CAE, the first step is to launch the Abaqus CAE software (Abaqus/CAE User's Guide, n.d.). Once the software is running, a new model can be created, and the Part Module selected. In this module, it is possible to generate the billet and die required for the extrusion process. All units used in the simulation are in SI.

To create the billet, the Sketch tool can be used to draw a 3D solid revolution as in (Figure 9). The axis of revolution, start and end angles, and dimensions of the billet must be specified. In this simulation, the billet is a 3D deformable solid revolution with dimensions of 0.1 by 0.3 meters, a start angle of 0° degrees and an end angle of 180° degrees.

After creating the billet, the same Sketch tool can be used to create the die. The die will also be a 3D deformable solid revolution with the dimensions specified in the provided sketch. The start angle of the die can be specified as 0° degrees and the end angle as 180° degrees. Once both the billet and the die are created, the next step involves defining the material properties.

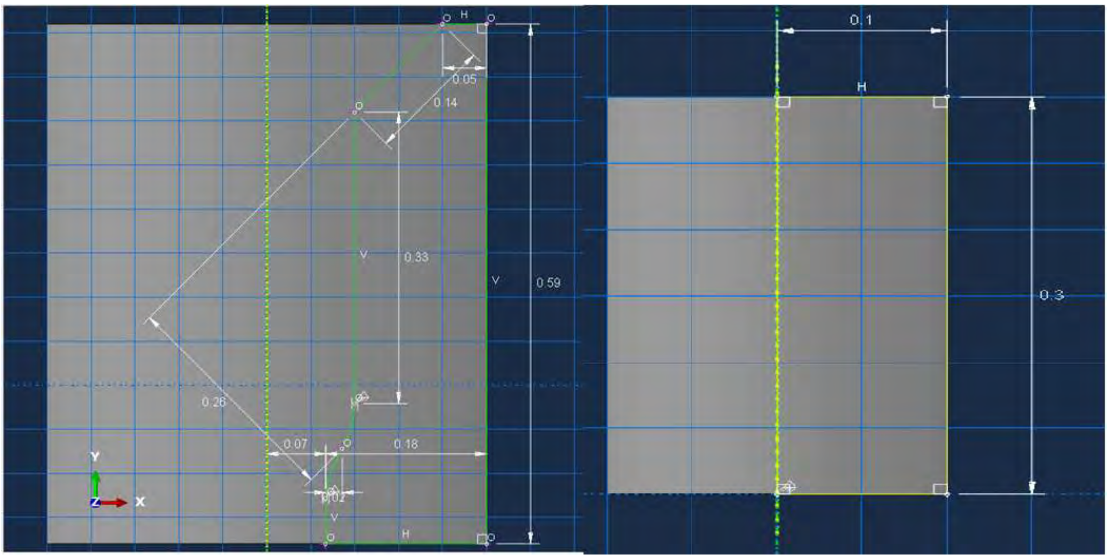

Figure 9: 3D extrusion billet sketch

3.2. Material properties

To declare the material properties for steel, one can navigate to the Material Module and follow these steps:

- Clicking on the "Create Material" button will allow a new material to be created.
- Steel" can be selected as the material type, and a name can be provided for the material.
- The material properties, such as Young's modulus, Poisson's ratio, and yield strength, can be input. These values can be obtained from the manufacturer's data or reference sources.
- Sections can be created for aluminium and steel, both of which are Solid-Homogenous.
- The aluminium section can be assigned to the billet, and the Steel section can be assigned to the die.

Figure 10 and the table of material properties below is exported from the Engineers Edge.

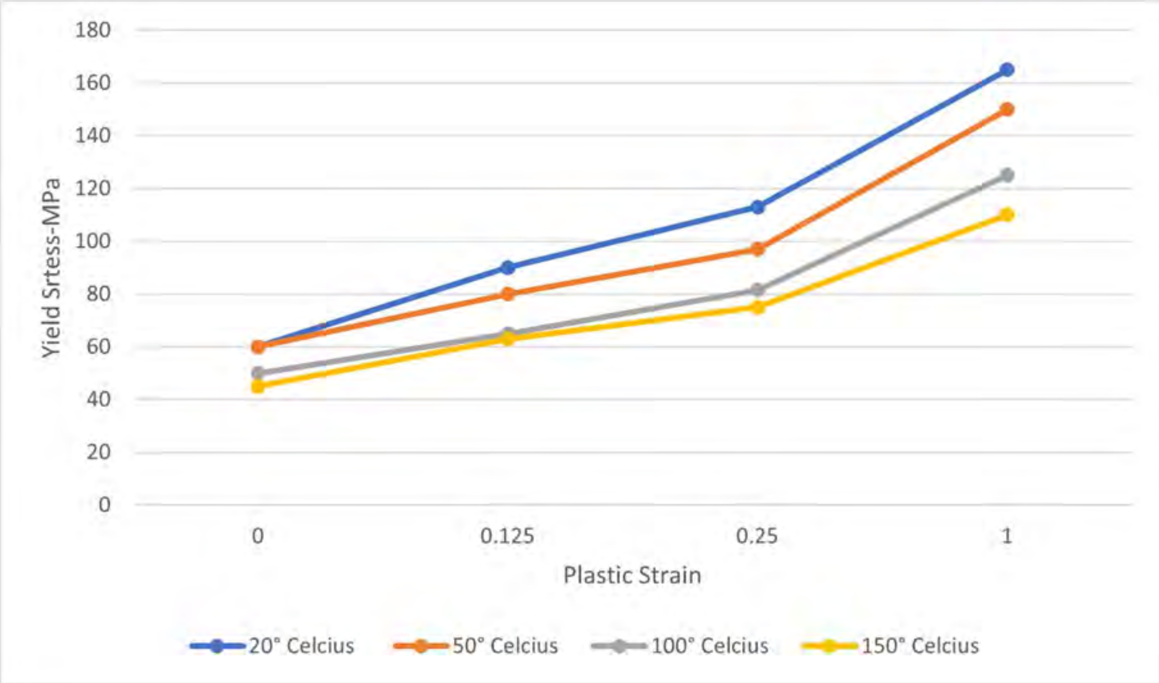

Figure 10: Plastic Stress and Strain

	Aluminium	Steel
Conductivity -(W/(mK))	204 (0° Celcius) 225 (300° Celcius)	50
Density-(kg/m^3)	2700	7800
Young Modulus- (Pa)	69e9	200e9
Poisson ratio	0.33	0.3
Inelastic Heat Fraction	0.7	0.9
Expansion coeffiecient- × 10^-6 /°C	23.1	12
Specific Heat- (J/kg*K)	880	460

Table 1: Material properties for commercial Aluminium and Steel

3.3 Assembly of extrusion components

In the Assembly module, the create instance button should be chosen and then both the billet and the die are selected. Also, in the instance, Dependent mesh is used and the reference point of the die is considered at the top of it as in Figure 11 below.

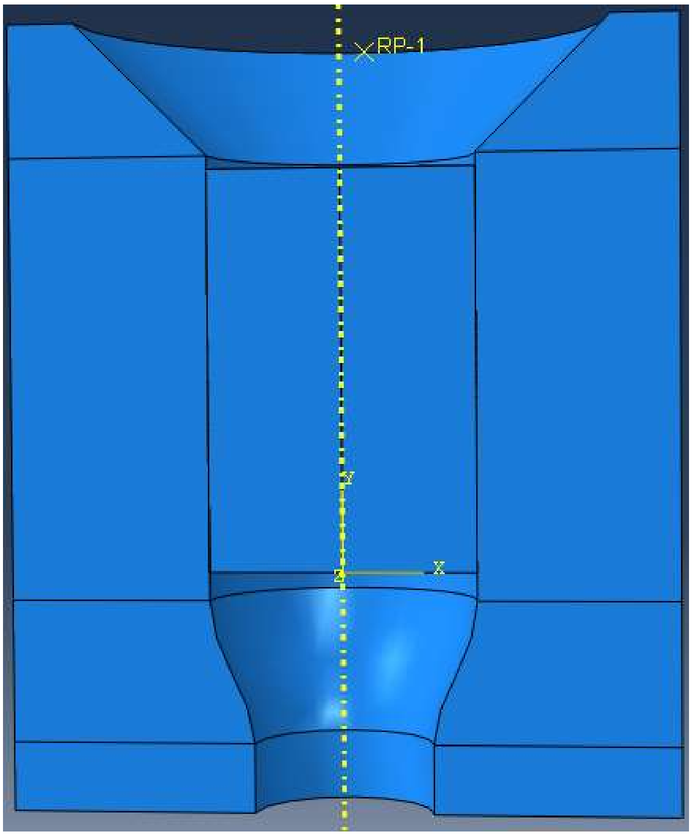

Figure 11: Assembly of the parts

3.4. Step definition and optimization

Coupled thermal-displacement analysis in Abaqus refers to a simulation in which temperature and displacement fields are concurrently solved and impact each other throughout the simulation. For instance, metalworking problems may involve substantial heating related to inelastic material deformation, which alters the material's characteristics.

Particularly, in the extrusion process, contact conditions arise in some issues in which the heat transferred between surfaces may be highly dependent on the distance between the surfaces or the pressure conveyed across the surfaces. Consideration is given to the thermal expansion or contraction of the material owing to temperature variations, which influences the displacement field. Similarly, the deformation of the material influences the temperature distribution inside the material.

At this stage, "Nlgeom" refers to the geometry used in the analysis. In a coupled temperature-displacement study, the behaviour of the material is dependent on both temperature and displacement, which may result in significant deformations. By selecting the "Nlgeom" option in the step module, the user indicates a desire to analyse the non-linear geometric behaviour of the material, taking into consideration the possibility of massive deformations.

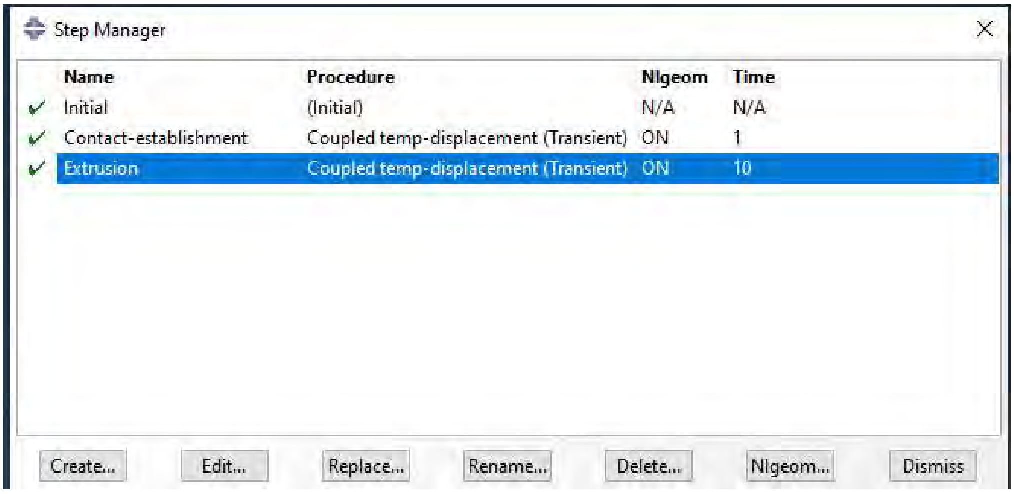

Figure 12: 3D extrusion step manager

In Abaqus CAE, the incrementation tab is used to manage the time stepping of the analysis. In the tab labelled Incrementation, the user may set the maximum number of increments, which affects the amount of time steps the analysis will take to achieve the final time and it can be calibrated by using the initial, minimum, and maximum values.

The starting value is the time step size used for the first increment. The minimum value is the shortest time step size and the maximum value is the biggest time step size that Abaqus will allow in the analysis. It is intended to avoid the time step size from getting too small, which would cause the analysis to be computationally costly while if it is too large it may lead to an incorrect solution.

The "include creep/swelling/viscoelastic behaviour" option in Abaqus/step CAE's module enables the user to add the effects of creep, swelling, and viscoelastic behaviour into the simulation. Creep is the persistent deformation of a material under stress over time, swelling is the expansion of a material caused by the absorption of liquids or gases and viscoelastic behaviour is the time-dependent, viscous and elastic behaviour of a material.

By including these characteristics, the simulation may offer a more realistic depiction of the material's real-world behaviour, reaction to applied loads throughout the extrusion process and forecast any flaws that may occur such as the central burst defect.

The extrusion process may be broken down into three parts, each of which needs unique Abaqus/CAE software configurations. The first phase is the initial, in which basic conditions for the extrusion process are established, as the billet's and die's starting temperature and displacement. The second phase is the formation of contact, which takes one second and its used so that the contact between the die and billet is established. The third step, that lasts ten second is where the billet is deformed as it is pushed through the die.

3.5. Interaction modelling and analysis

Tangential behaviour with penalty refers to the method used to define the contact behaviour between the billet and the die during the extrusion process (Figure 13, Figure 14). The name and activated steps can also be seen. The contact is defined using a penalty approach, which involves adding a penalty term to the energy equation of the simulation to enforce the contact. The penalty term ensures that the surfaces do not penetrate each other during the simulation, as if they were in physical contact.

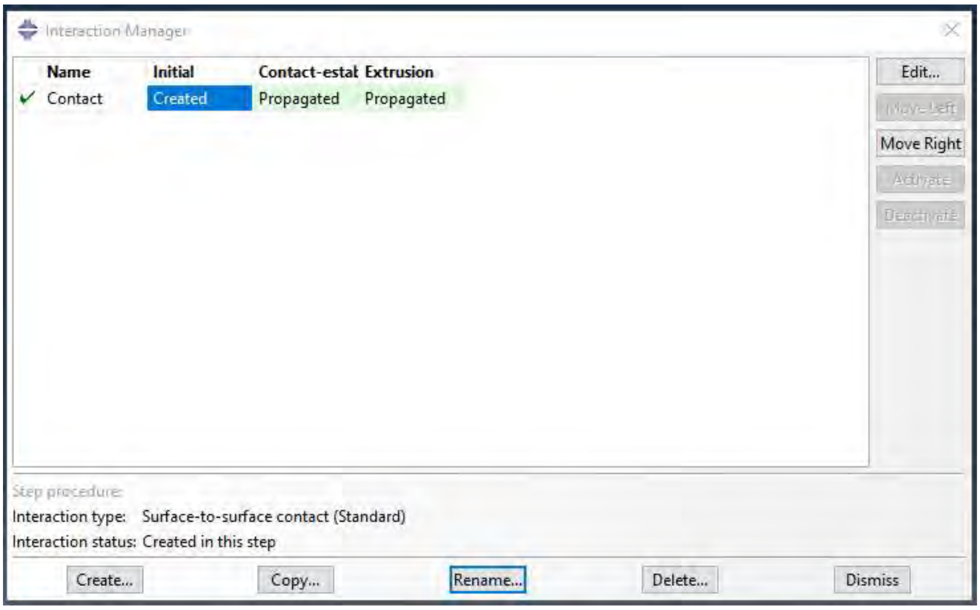

Figure 13: 3D extrusion interaction manager

In this instance, the friction coefficient of 0.1 is employed to model the frictional behavior between the billet and the die. The heat generation is set to its default setting, which indicates that the program will automatically calculate the amount of heat created during simulation based on the material parameters and other simulation factors. This method is beneficial for simulating the extrusion process because it properly captures the interactions between the billet and the die, hence shedding light on the system's behavior throughout the process.

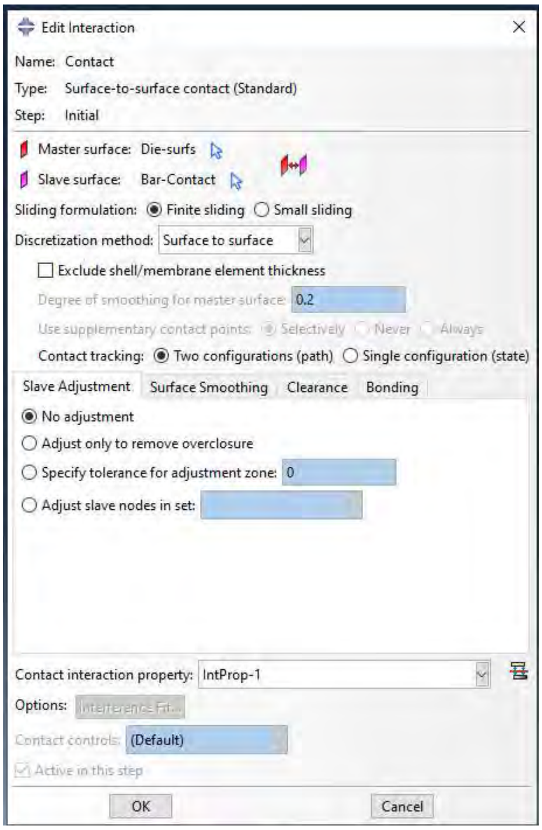

Figure 14: 3D Extrusion interaction properties

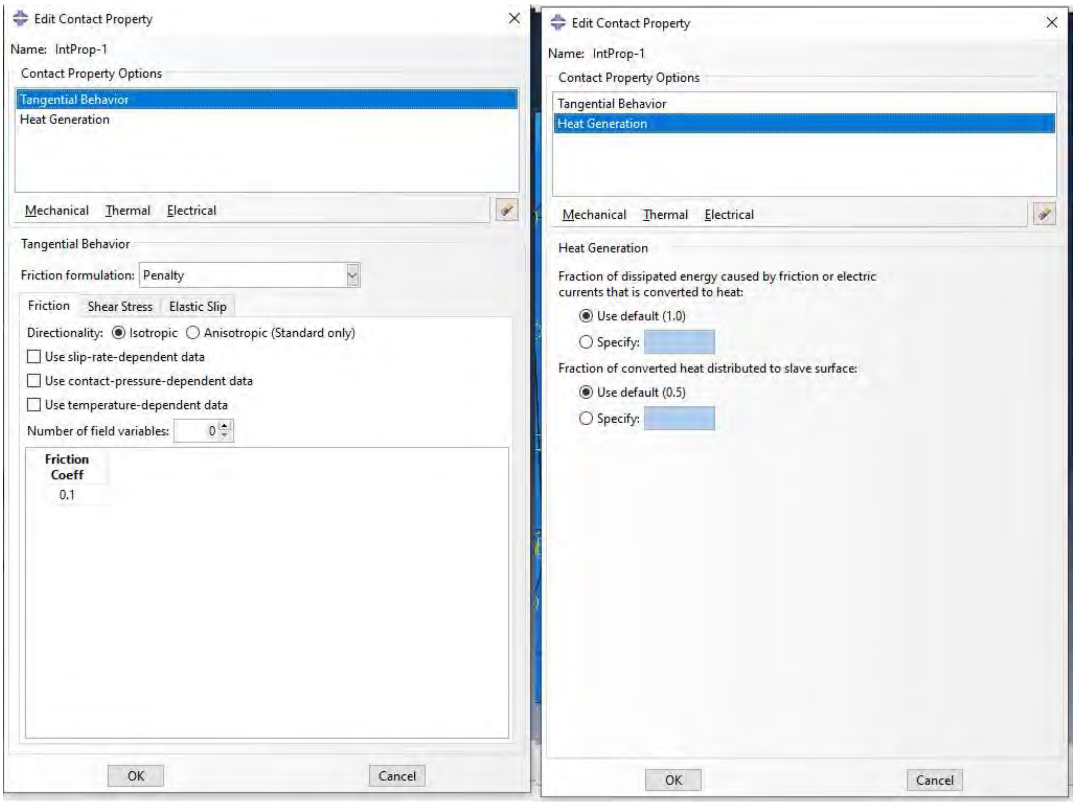

Figure 15: 3D Extrusion contact properties

3.6. Load and boundary conditions

Figure 16 depicts the load conditions in the extrusion process simulation are critical to accurately capture the behavior of the material during the process.

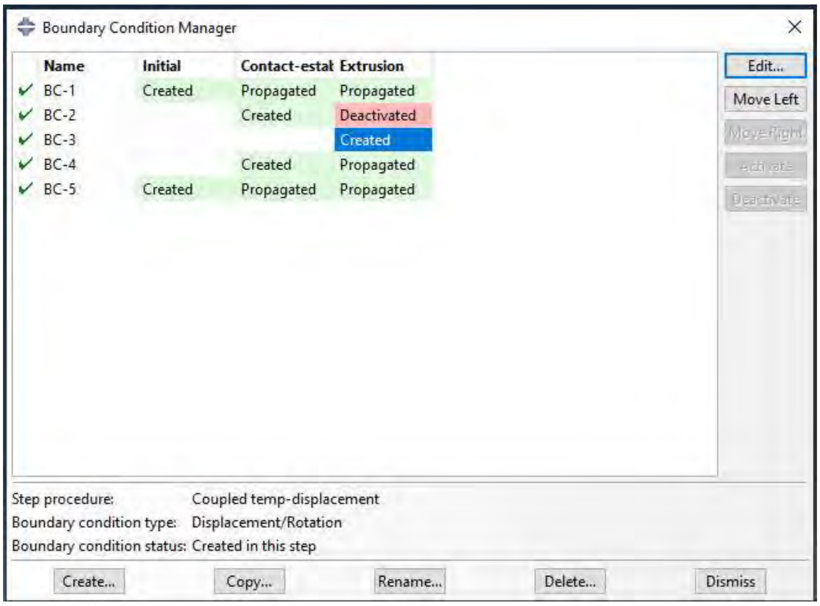

Figure 16: 3D extrusion boundary condition manager

The encastre constraint, as seen in Figure 17 upper left corner, applied to the initial load condition offers a fixed boundary condition to the die's reference point, prohibiting it from moving in any direction. This indicates that the reference point of the die will stay fixed during the simulation and it is applied at the beginning to ensure the stability of the model. At the contact-establishment stage, the second load condition, a displacement of 0.000125m along the Y axis at the top of the billet, represents the beginning movement of the billet into the die. This causes the billet to slide slightly inside the die, resulting in bringing the two surfaces into contact.

The Y-axis displacement reflects the vertical movement of the billet into the die, and the interaction module establishes contact between the two surfaces by using surface-to-surface contact with the die as the master surface and the billet as the slave surface. During the extrusion stage, the third load condition, the extrusion displacement of 0.25m along the Y axis at the top of the billet, represents the billet's path through the die and reflects the billet's final position after extrusion.

As part of the fourth load condition, a temperature of 20° degrees is applied to the reference point of the die, which is the contact-establishment step. The amount of the temperature load may be modified depending on the analysis's individual needs.

The final load condition, ZSYMM at the front of the billet (first step), is a boundary condition that guarantees the front plane symmetry of the billet, essentially constraining the front of the billet to maintain its original position. In the study of the 3D extrusion process, symmetry is crucial because it reduces the size of the model and the computing time without compromising the quality of the findings.

The specified temperature field with a magnitude of 20° impacting the whole billet is not a load condition, but rather a predefined temperature field applied in the first phase. This parameter is distinct from the other load conditions and is used to describe the billet's starting temperature before the extrusion process begins.

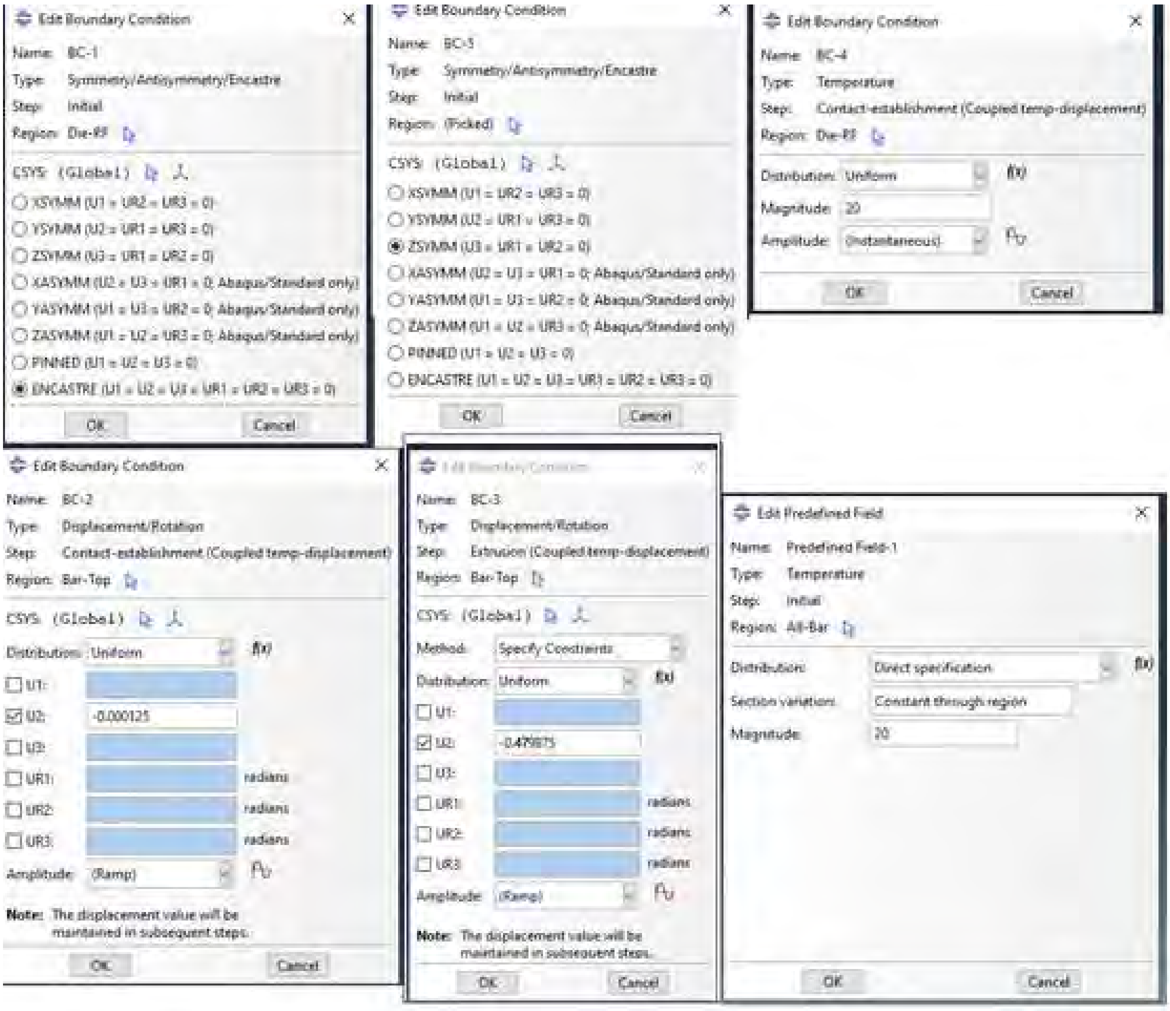

Figure 17: Boundary conditions settings

3.7. Mesh generation and optimization

Meshing refers to "discretization," which is the process of breaking down a 3D object or geometry into smaller, more manageable pieces. The mesh is the starting point for numerical analysis, which determines an object's behaviour by numerically solving equations that describe the physical events.

Abaqus includes a number of meshing techniques to help users generate meshes that have the required precision and computational efficiency for their simulations. For even more precise outcomes, mesh refinement may be used to focus on high-stress or locations of interest.

The numerical simulation of thermo-mechanical problems often uses a finite element of the C3D8RT type (Figure 19). It is a brick 8-node hexahedral element, the "R" indicates reduced integration and "T" stands for hourglass control. Reduced integration is used to lower the computational cost, as the stiffness matrix of this element type is computed with a smaller number of integration points and the hourglass control prevents unwanted deformations that occur due to hourglass modes.

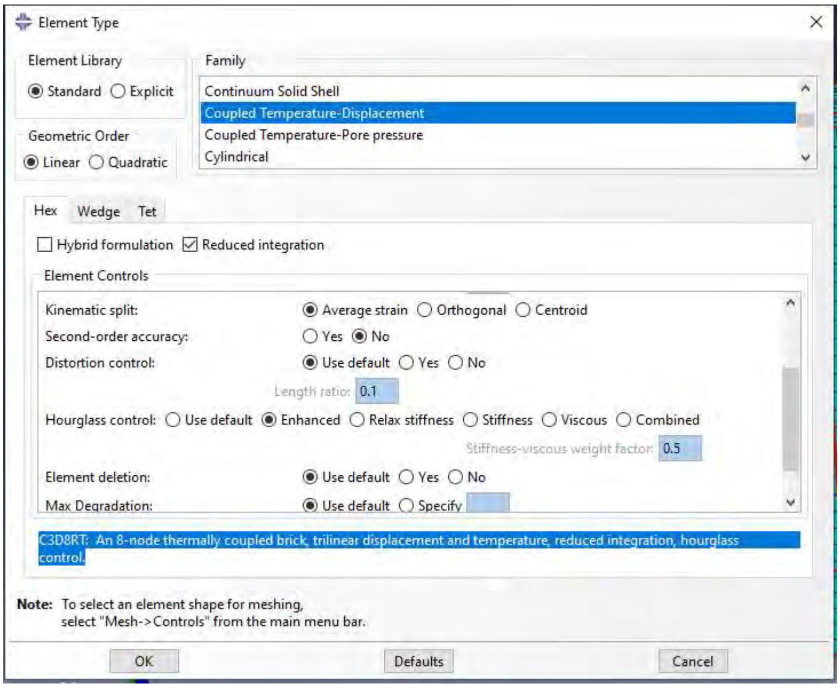

Figure 18: 3D extrusion element type window

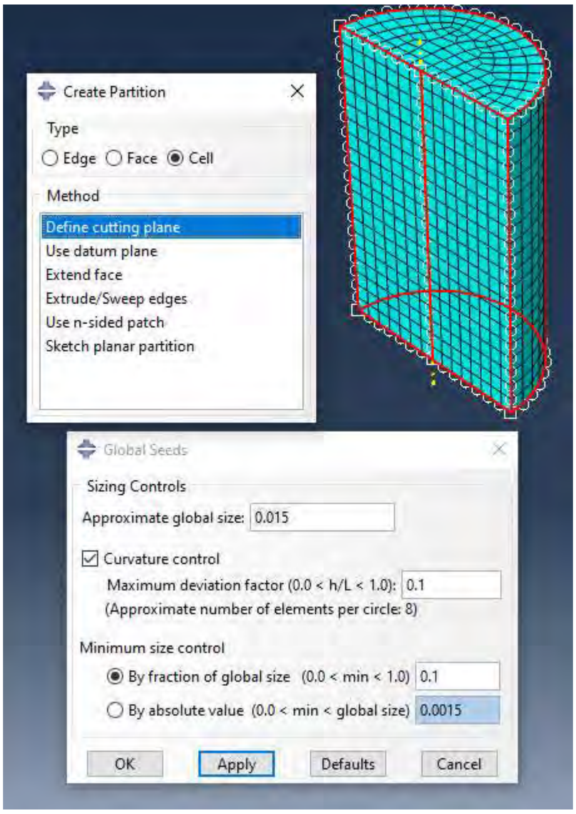

Figure 19: 3D extrusion mesh module

3.8. 3D extrusion with cooling step

Residual stress should be accounted for in the manufacturing process, as it may impair the product's performance and durability due to the fact that it remains in a material after exposure to external loads or temperature changes, such as mechanical stresses, thermal expansion, etc. Deformation of the material during extrusion, temperature variations, and changes in material properties are all possible sources of residual stresses, which increase the probability of fracture development and reduce the fatigue life of the material.

By simulating an extrusion process that includes a cooling step, it is possible to examine residual stresses in a finished product. This may be accomplished by directly measuring the residual stresses using techniques like as X-ray or neutron diffraction, or by assessing the temperature and strain fields during and after cooling. To reduce the amount of residual stress in the end product, the die design may be modified, the initial temperature of the components may be altered, and alternative lubrication techniques may be considered.

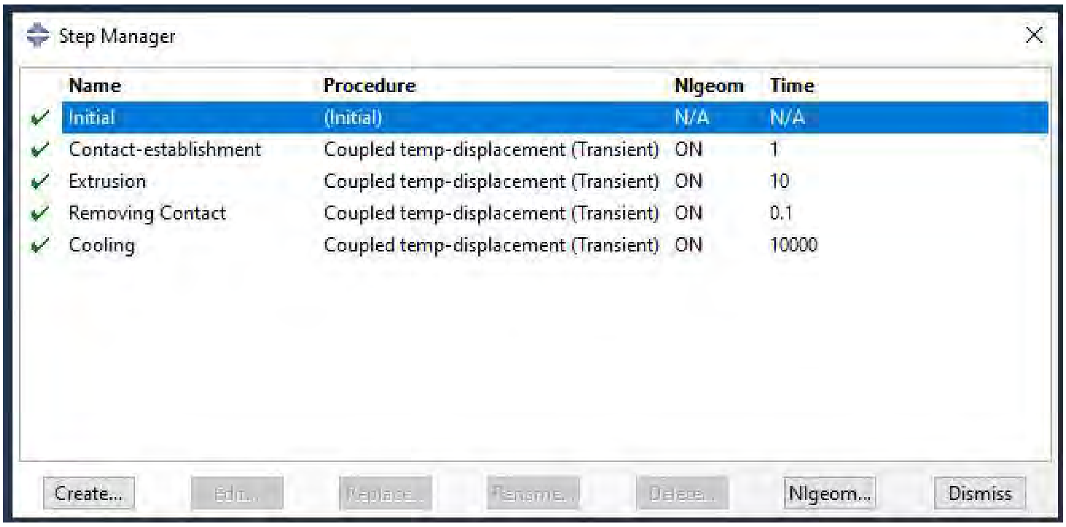

Figure 20: Extrusion process with cooling step

3.9. Results of 3D extrusion process

S, MISES

In Abaqus, the term "S Mises" refers to the Von Mises stress, which is a measure of the equivalent stress experienced by a material under complex loading conditions. The equivalent stress is calculated based on the maximum shear stress that a material can endure before it reaches its yield strength. The values derived from the equation (1) can be used to identify areas in a model that are experiencing high levels of stress, which may indicate potential failure locations and can be easily visualized by plots in Abaqus visualization.

Von Mises equation:

$$\sigma_v = \sqrt{(\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2 + 6({\sigma_{12}}^2 + {\sigma_{23}}^2 + {\sigma_{31}}^2)} \qquad (1)$$

The figures below output the von Mises stress distribution during an extrusion 3D process. In Figure 21a, at the initial step there is no observable stress or strain.

In Figure 21b, where the von Mises stress is 1.29e+08Pa at the start of the reduction area, the material is undergoing significant plastic deformation as it is forced through the reduction area. This high level of equivalent stress indicates that it is experiencing a high level of shear stress, which is likely causing it to deform and experiences the most resistance.

Figure 22a, shows the maximum von Mises stress that is valued to 1.483e+08Pa and is located at the middle of the billet in the reduction area. The increase in stress suggests that the metal is encountering an even higher level of resistance as it continues to be extruded through. The location of the maximum stress suggests that it is now undergoing an equal amount of stress on both ends, indicating that has been evenly deformed.

In Figure 22b, where the material is cooling and the maximum von Mises stress has decreased to 1.31e+08Pa, it is no longer undergoing significant deformation. The decrease in stress is likely due to a reduction in both the applied loads and the temperature. The lower stress levels indicate that the billet has reached a state of stability and is unlikely to undergo further deformation, but in real conditions there would be residual stress.

In conclusion, these figures show the von Mises stress distribution during different stages of an extrusion process, including a stage of high stress as the material is deformed in the reduction area, a stage of increasing stress as it continues to be deformed, and a stage of cooling and stress reduction as the material reaches stability.

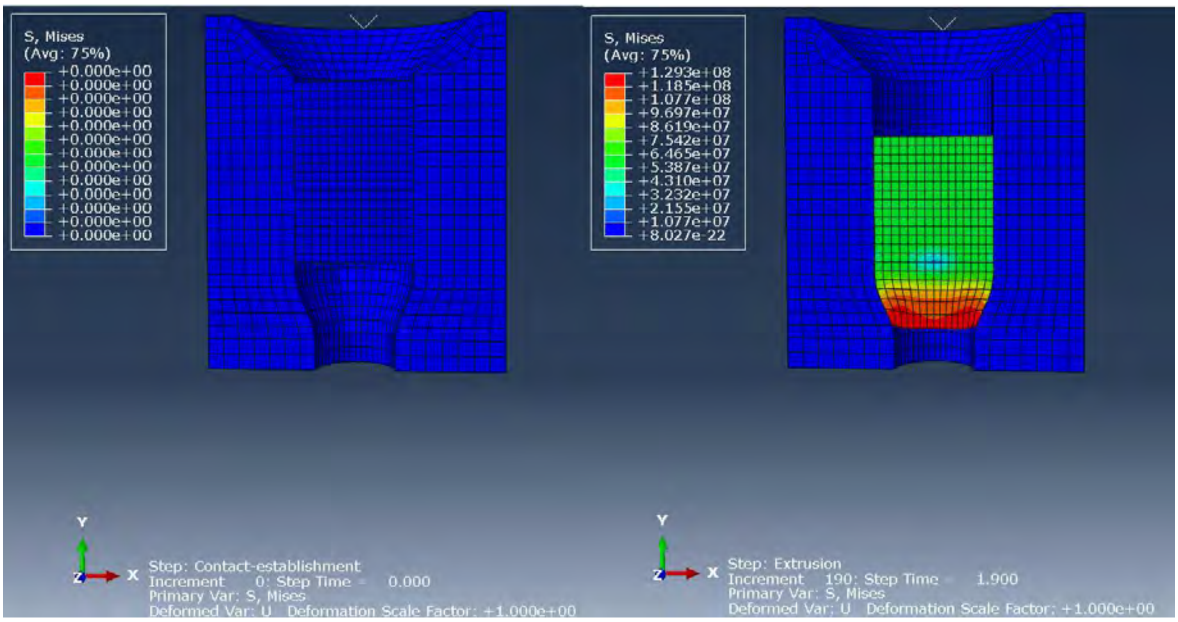

Figure 21 (a,b): S, Mises visualization- Contact/establishment, Extrusion

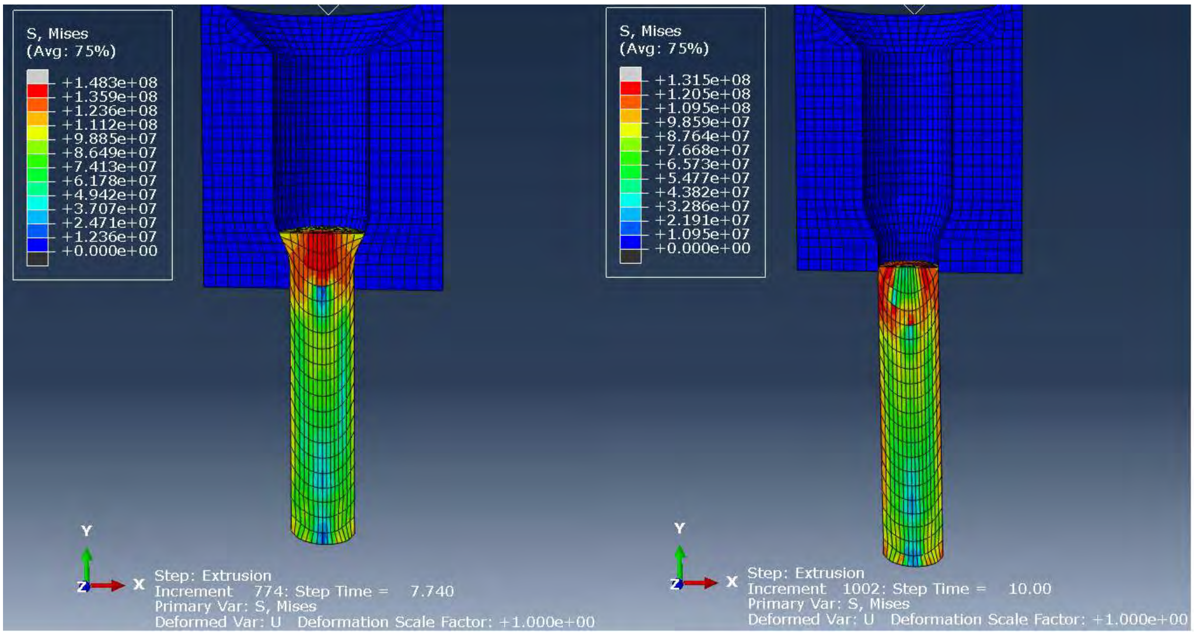

Figure 22 (a,b): S, Mises visualization- Extrusion

NT11

The figures below show the temperature distribution during an extrusion process as visualized by NT11. In Figure 23, the initial state is displayed, in which the billet is in a stable temperature condition of 20°C.

Figure 24a depicts the temperature distribution at the middle of the extrusion process, where the temperature has increased to 117°C, responding that the material is experiencing gradual heating and thermal stress as it is deformed.

The final temperature distribution is shown in Figure 24b at the end of the extrusion step, where the maximum temperature has increased to 163°C. This further increase in temperature suggests that the material is undergoing even more heating and thermal stress, likely due to increased resistance encountered.

In conclusion, these figures show the temperature distribution during an extrusion process, starting from an initial stage of low temperature and stable conditions, through a stage of gradual heating and thermal stress as the material is deformed, reaching the maximum value at end of the process and then the temperature slowly decreases during cooling phase, but of course with the occurrence of residual thermal stress.

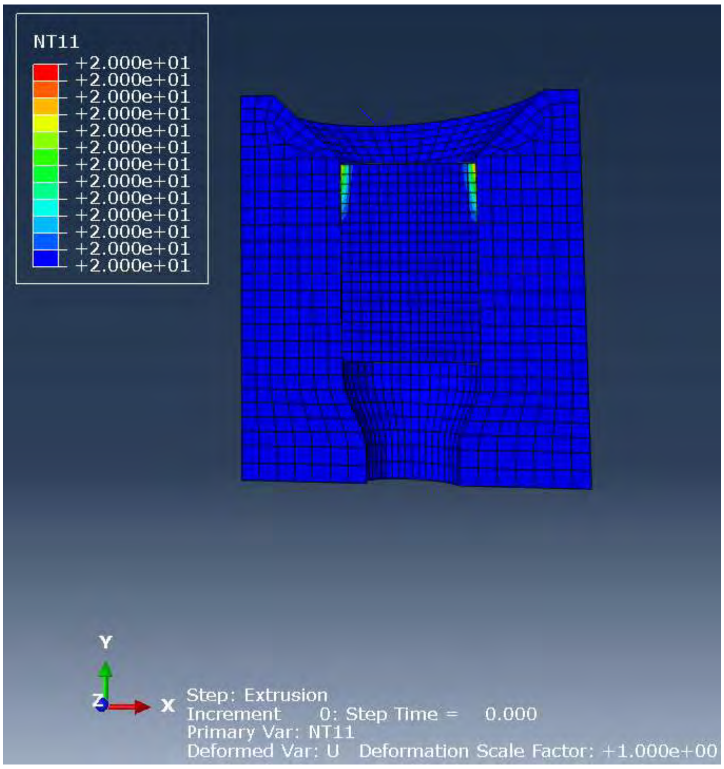

Figure 23 (a,b): NT11 output Extrusion(Initial)

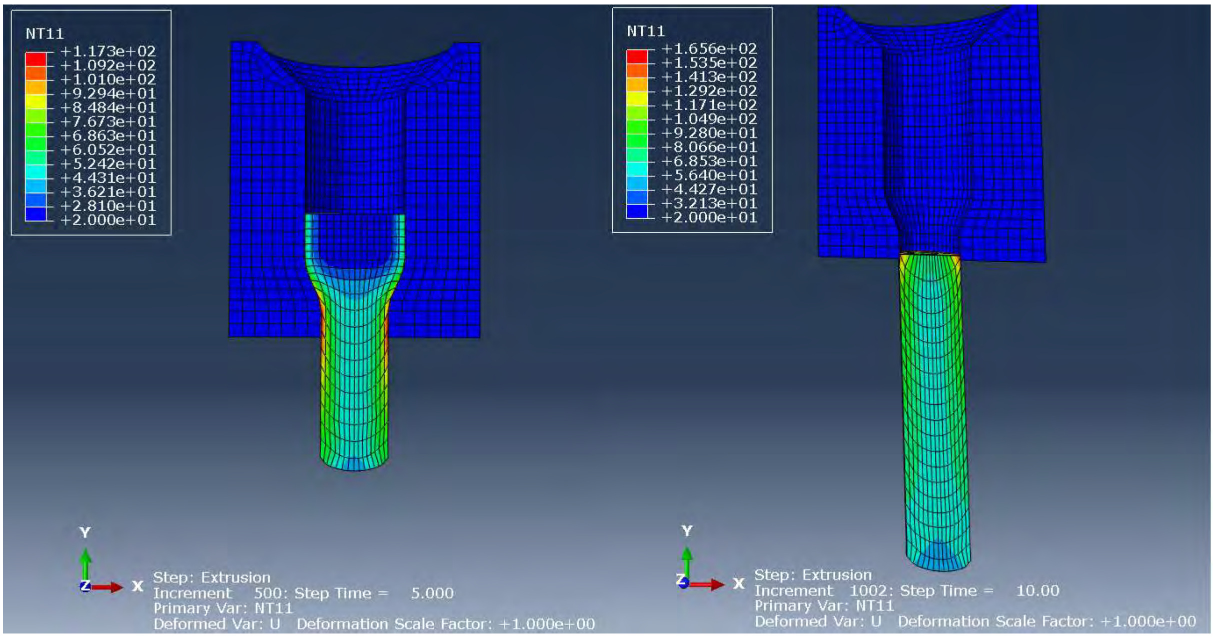

Figure 24 (a,b): NT11 output Extrusion(5-10seconds)

NT11-time plots

The temperature at each node in a simulation is determined by the heat transfer in the material, which is in turn affected by its conductivity, specific heat, and inelastic heat fraction. Conductivity is the capacity of a substance to transport or conduct heat and it is symbolized by “k”, "λ" and "κ". This quantity's reciprocal is known as thermal resistivity. The amount of heat needed to increase the temperature of one gram of a substance by one degree Celsius is known as its specific heat. In contrast, during plastic deformation, a certain percentage of the applied energy is converted into internal energy, leading to a temperature rise and represents the inelastic heat fraction.

If a material's conductivity and specific heat are both high, it will be subject to less temperature variation than one with lower values. Additionally, a material's temperature will increase more rapidly if its inelastic heat fraction is large compared to that of a material with a low inelastic heat fraction.

Node 1282, located about in the center of the interior, heats up to a maximum of 62.2°C throughout the course of the extrusion, as seen in the first plot (Figure 25). Node 1261 (Figure 26) on the billet's edge, meanwhile, heats up to a high of 124.2°C before cooling to 82°C. The degree to which each node is subjected to deformation influences the rate of heat transfer, which explains why there is a temperature disparity between those nodes. As a consequence, it is essential to take them into account while simulating the temperature profile at various nodes.

Figure 27 shows how the deformation and friction experienced by node 1656 cause its temperature to increase sharply throughout the extrusion operation. The extrusion will be followed by a cooling process that will cause a temperature reduction. The material's cooling rate may be estimated after the temperature stabilizes at a lower level.

The linear thermal expansion coefficient represents the change in length per unit length per degree Celsius rise in temperature, so when the temperature increases, the length of materials with larger linear thermal expansion coefficients will vary more noticeably. The internal stress and strain state of the material is affected by this, and residual stresses and strains may arise as a result. If the linear thermal expansion coefficient of the material is large, for instance, the surface nodes will suffer a greater increase in temperature than the interior nodes, resulting in a temperature difference throughout the billet. As a consequence, getting reliable simulation results requires taking expansion into account.

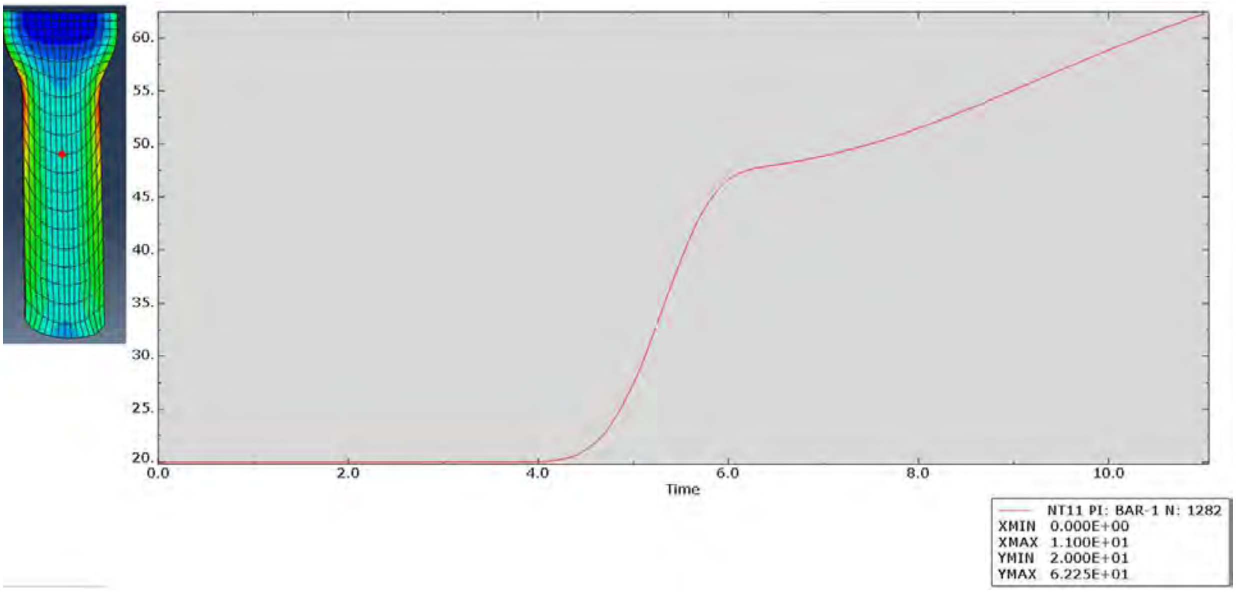

Figure 25: Node 1282: Temperature-time plot

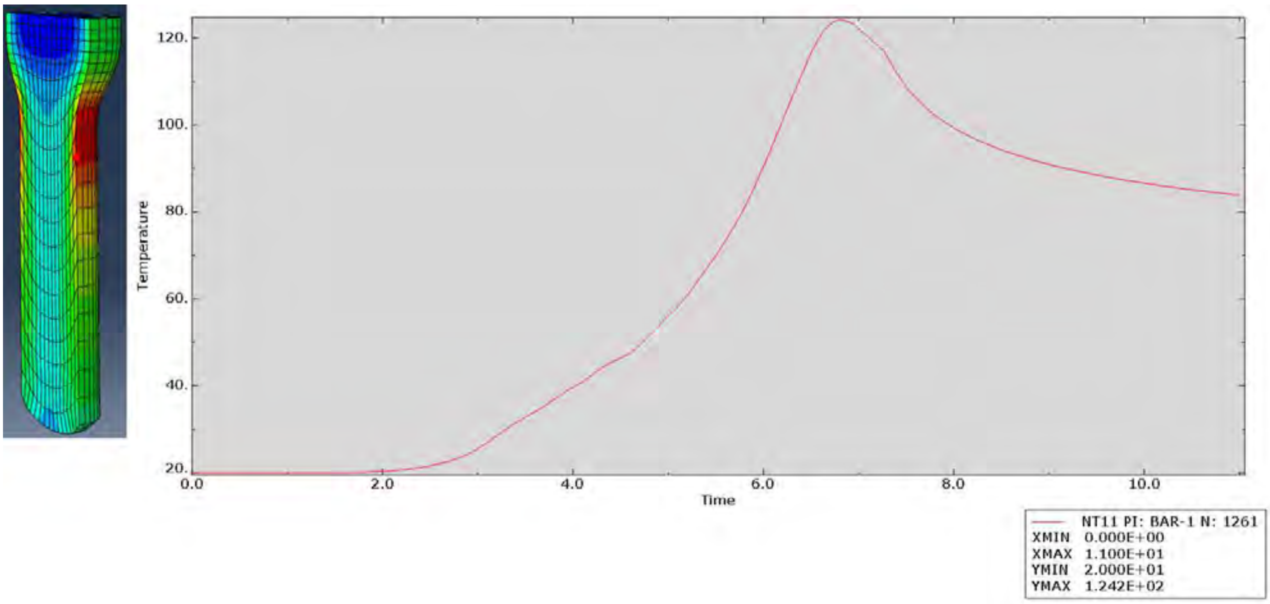

Figure 26: Node 1261: Temperature-time output

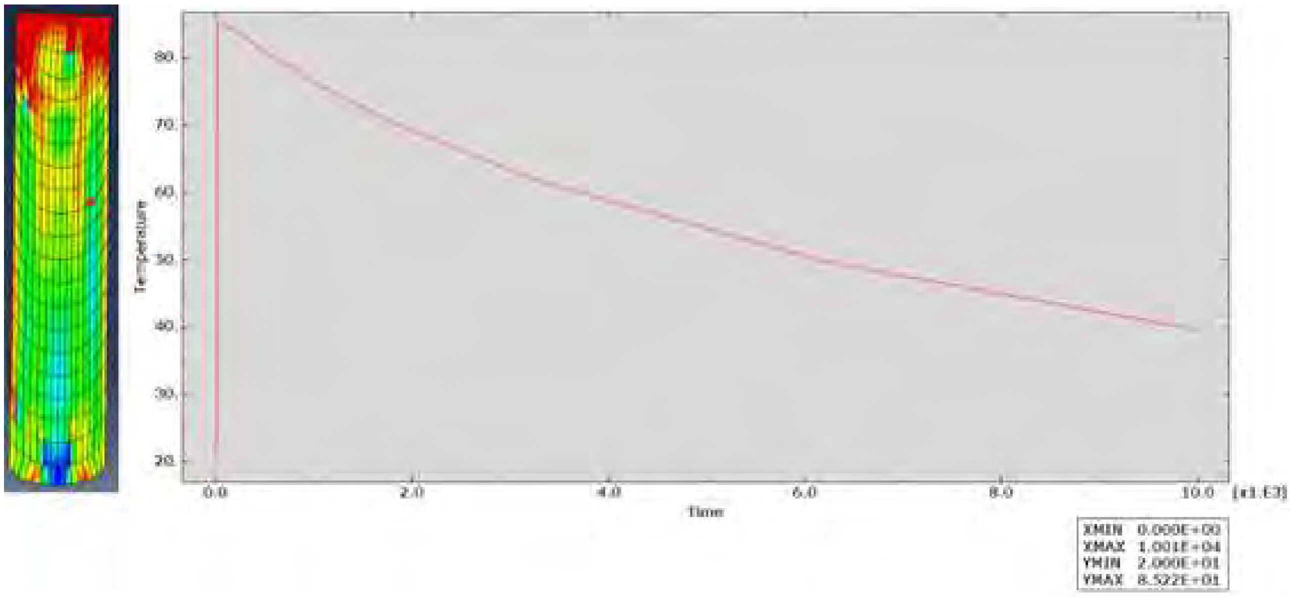

Figure 27: Node 1656 temperature-time

Stress-Strain plots

The two stress-strain plots demonstrate the differences in the stress and strain experienced by different nodes in a 3D extrusion analysis. The first plot (Figure 28), element 1183, which is located in the middle of the interior, reaches a maximum von Mises stress value of 1.72E+08Pa and a maximum strain of 0.8. Element 992 (Figure 29), located at the side of the billet, reaches a maximum von Mises stress value of 2.35E+08 Pa and a maximum strain of 1.2.

The difference in the von Mises stress values of the two nodes, element 1183 and element 992, can be attributed to various factors that influence the material behavior under stress. These factors include the material's density, plasticity, elasticity, thermal expansion, heat conductivity, inelastic heat fraction, and specific heat.

Materials that have more density than others tend to deform more difficult, but that is not always a general truth as factors such as material composition and structure can influence this. Plasticity refers to the material's ability to deform plastically under stress, elasticity refers to the material's ability to return to its original shape after the stress is removed and higher plasticity/elasticity materials would have a lower von Mises stress value.

Heat conductivity and specific heat affect the material's temperature profile under stress, and higher conductivity and specific heat materials would have a lower von Mises stress value compared to lower conductivity and specific heat materials. Instead, higher expansion materials would have a higher von Mises stress value compared to lower expansion materials. The values of these factors would influence the material behavior and therefore, the stress distribution in the material and result in the difference in the von Mises stress values of the aforementioned nodes.

A stress-strain plot of a specific node in an Abaqus simulation of a cooling step extrusion process can provide insight into the mechanical behavior of the material during the cooling process. After the extrusion step, the material will be cooled rapidly, which can result in residual stresses and deformation. By selecting a node of interest, it is possible to monitor the stress and strain changes in real-time during the cooling process.

In such a plot Figure 30, the stress will initially increase due to the deformation caused by the extrusion process. After the cooling step, the stress will decrease rapidly due to the reduction in temperature and the decrease in the material's ductility. The strain will also decrease due to the reduced plasticity of the material. This information can be used to optimize the cooling conditions and prevent deformation or failure of the material.

A butterworth-filter was applied to the stress of the data to cutoff noise and guarantee better visualization. In the field of signal processing, a butterworth filter is a typical electrical filter used to remove noise from signals. Noise in the data that may have compromised the reliability of the plots was eliminated by using the filter in this example. The transfer function of the butterworth filter (Equation 4) is:

$$|H(j\Omega)| = \frac{1}{\sqrt{1+(\frac{\Omega}{\Omega_c})^{2N}}} \quad (4)$$

where, N is the order of the transfer function and Ω_c is the cutoff frequency.

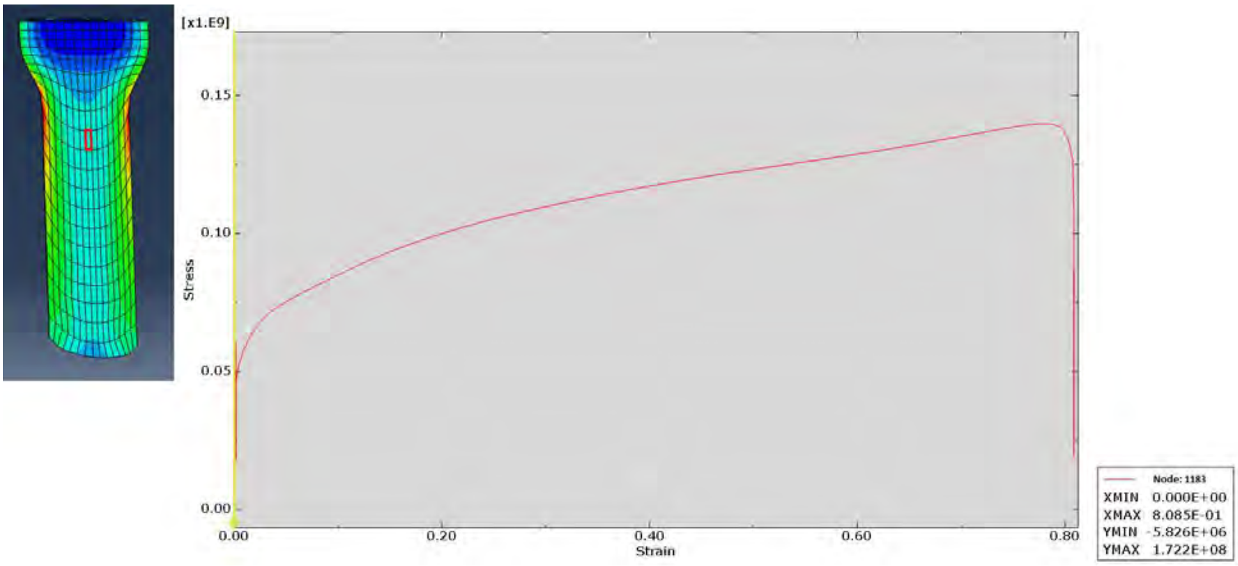

Figure 28: Element 1183: Stress-Strain output

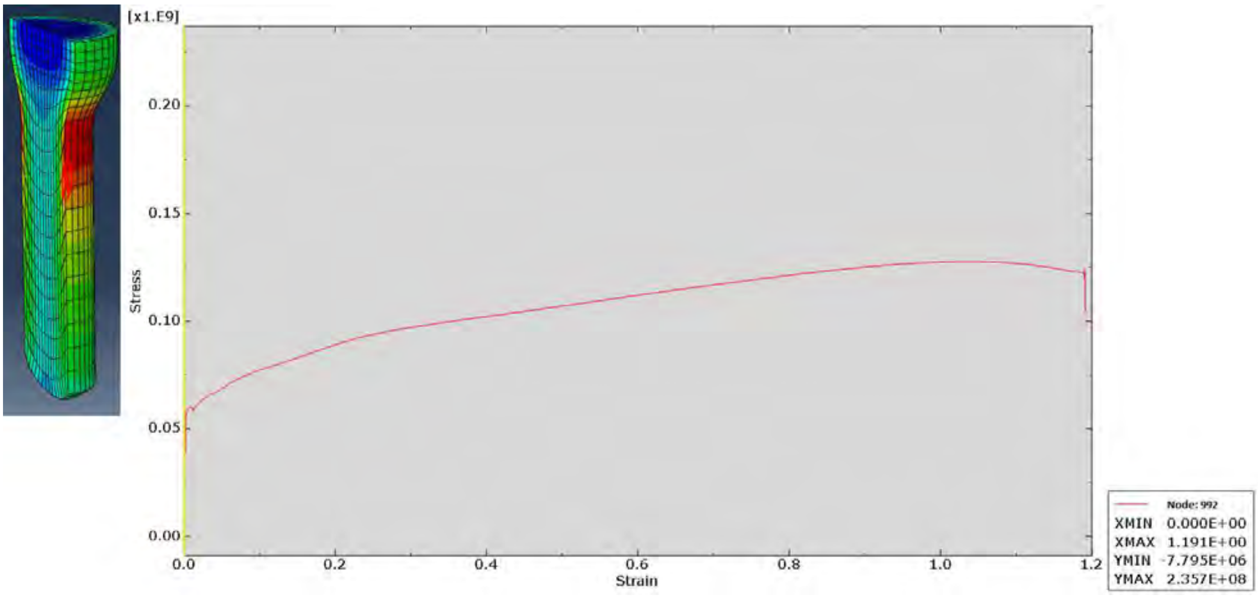

Figure 29: Element 992: Stress-Strain output

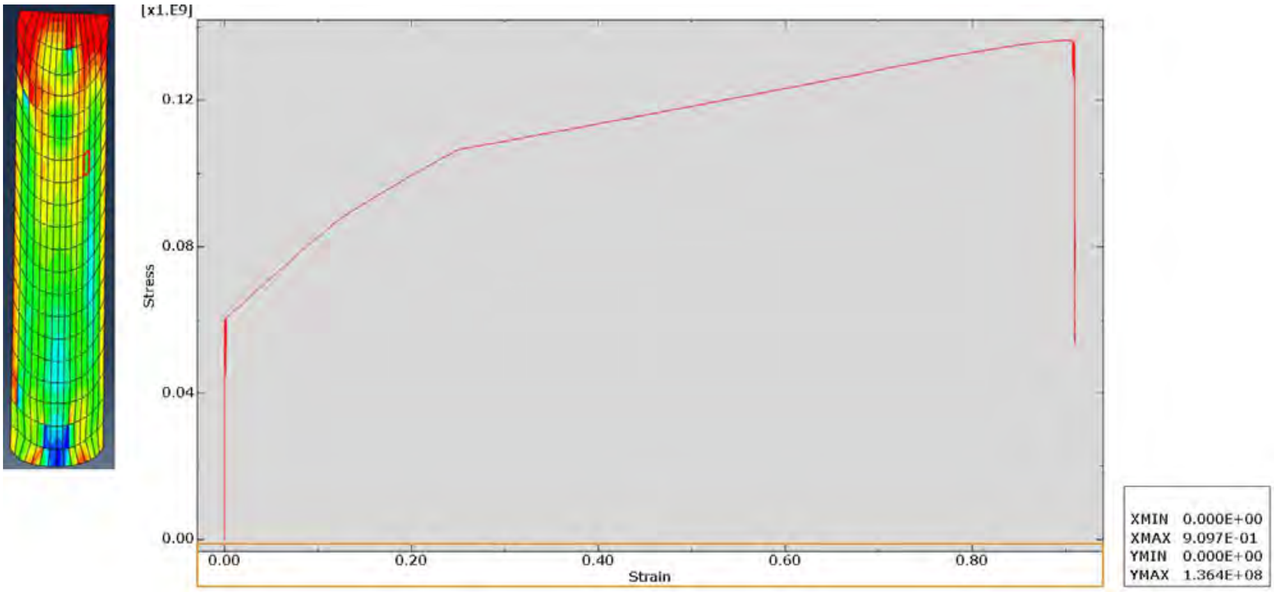

Figure 30: Cooling step extrusion, Element 1425 stress-strain

Force-Displacement plot

During a mechanical process like extrusion, the connection between the force exerted and the displacement of the material is represented by a force-displacement plot, figure (Figure 31). In this example, the extrusion step displacement is reflected in the displacement data, which was collected from a single node. The force data was collected by adding up the forces operating at the billet top's individual nodes and multiplying two times the force to account for the full billet. After smoothing the force data with a Butterworth filter with a cutoff frequency of 5 Hz, the two data sets were merged to produce the force-displacement figure. With a total of 8.032E+06 N of force, the maximum displacement in the graph is 0.48.

Elasticity, plasticity, and strength are only few of the material attributes that have an impact on this data. The plasticity of a material defines if it can be permanently deformed without breaking and the elasticity accounts for whether it will return to its former shape after. In contrast, the force that can be applied before the material breaks depends on its strength. The force-displacement plot may also be influenced by the material's temperature, density, and the existence of any faults or inclusions.

Due to the absence of external forces acting on the billet's surface during the cooling phase, the resulting plot is identical to what would be obtained if a cooling step were included in the load properties.

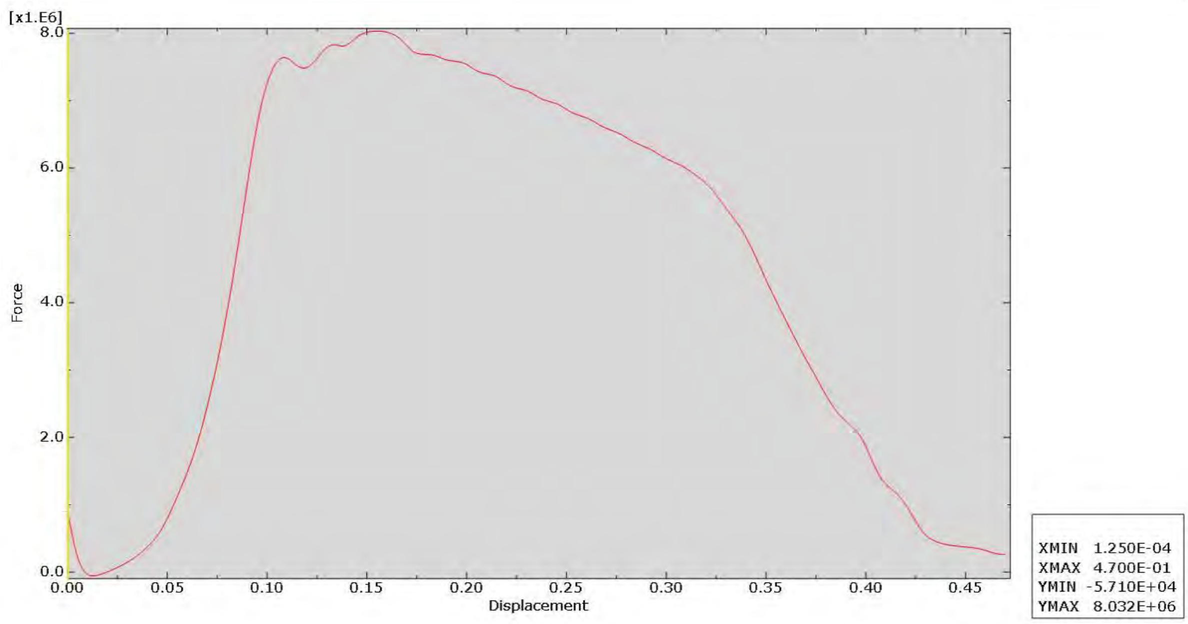

Figure 31: Force-Displacement plot

Chapter 4. Central burst extrusion simulation

4.1. Historical context

The appearance of chevron cracks or central bursts in a cold extrusion or drawing process can lead to significant difficulties, as these internal cracks are difficult to detect and can cause serious problems in the final product. The production of a large number of shafts through extrusion in the automotive industry highlights the importance of understanding how to prevent the occurrence of chevron cracks. As a result, predicting chevron cracking is crucial for the design and control of the process.

To predict internal cracks like chevron cracking in the cold extrusion process, it is important to study fracture criteria. Early research focused on determining fracture criteria and predicting crack initiation based on energy criteria. For example, Freudenthal (A. M. Freudenthal, 1950) used effective stress for the stress component and Cockcroft and Latham (M. G. Cockcroft and D. J. Latham, 1968) employed maximum principal stress to calculate the energy term. Clift (Clift et al., 1990) compared these criteria in simple upsetting, axisymmetric extrusion, and strip compression and tension. Wifi (A. S. Wifi, 1998) compared two different preforming processes of upsetting and forward extrusion. Kim (Kim et al., 1999) investigated three sequential stages in the manufacturing of an axisymmetric pin. These works show that it is possible to predict cracking positions in each process with Cockcroft-Latham and specific plastic work criteria, but there is no unique solution for general applications. Therefore, it is very crucial to determine a proper fracture criterion to analyze central bursting in cold extrusion.

The prediction of the defect was first tried by Avitzur (Avitzur, 1968) based on die angle and reduction of area. Zimmerman (Zimerman et al., 1971) demonstrated through experimentation that the safe zone is defined by a small die angle and high reduction in area. Based on their analysis, Zimmerman and Avitzur (Zimerman & Avitzur, 1970) proposed a criterion for central bursting using the upper bound method. For numerical analysis, Aravas (Aravas N., 1986) investigated the behavior of micro-voids formed during axi-symmetric extrusion with two different die designs. Saanouni (Saanouni et al., 2004) utilized continuum damage mechanics in a thermo-elastoplastic finite element (FE) program to predict crack formation in cold extrusion.

This thesis if based on the work by (Choi et al., 2010), that developed a numerical algorithm that combines the element deletion method and the rigid-viscoplastic finite element approach was utilized to analyze the occurrence of chevron cracking. The Cockcroft-Latham (Equation 3) and specific plastic work fracture criteria (Equation 2) were employed to predict the formation and progression of potential cracking of an aluminum alloy with properties presented below.

$$\int_0^{\bar{\varepsilon} f} \bar{\sigma} \mathrm{d}\bar{\varepsilon} = C_1 \qquad (2)$$

$$\int_0^{\bar{\varepsilon} f} \sigma_1 \mathrm{d}\bar{\varepsilon} = C_2 \qquad (3)$$

Here, $\bar{\sigma}$ is the effective stress, $\sigma_1$is the largest tensile principal stress and $\bar{\varepsilon}f$ is the effective strain at the initial cracking. C_1 and C_2 represent the critical damage value of each criterion in MPa, respectively, depending on the material.

Gurson, Johnson-Cook, Bai-Wierzbicki , and Cockcroft-Latham failure criteria are extensively used to forecast material failure during metal forming. The Cockcroft-Latham criteria is referenced more than 2,000 times in the scientific and technical literature, according to an examination of Internet-based publications during the last five years. Its success is mostly due to its simplicity and the little amount of material data necessary for computations.

It predicts the onset of crack initiation and growth and plays an important role in determining the residual life of a structure and its durability. The criterion is often used in combination with a plastic work failure criterion, but in this thesis it is studied as the integral damage value using the effective stress and equivalent strain.

Also, the works of Akram et al., 2018; Kathirgamanathan et al., 2006; Kubík et al., 2011; McAllen & Phelan, 2005; Ranjan Yadav et al., 2018; Sebek et al., 2015a, 2015b; Zeynep Parlar, 2015 greatly impact the understanding of the analysis and should be mentioned.

4.2. Dynamic, explicit analysis

The CAX4R, 4-node, reduced-integration, first-order, axisymmetric solid element and is used to model complex 2D axisymmetric shells (Abaqus Example Problems Guide, n.d.). It uses fewer integration points than a standard 8-node shell element. This reduced integration leads to improved computational efficiency, but still provides accurate results for axisymmetric problems. This type of element is designed to provide improved accuracy and computational efficiency when modeling axisymmetric geometries.

In this study, a CAX4R element with an approximate global size mesh of around 8000 elements is used to model a 0.29m height and 0.025m width meters axisymmetric deformable billet in a 2D extrusion analysis.

The die used in the simulation has dimensions described in the sketch (Figure 32) and is modeled as an analytical rigid wire. The reduction area is 36% and the semi die angle is 20° degrees and the step in the analysis is dynamic explicit in order to deal with nonlinear issues. Surface-to-surface contact is also considered in the simulation, with interaction properties defined as tangential behavior penalty and a friction coefficient of 0.1.

An encastre load is applied to the die, and the billet top has a velocity of 0.385 m/s. Additionally, an axisymmetric condition is applied to the internal surface of the billet. This condition ensures that the material is symmetrical about the central axis of the billet and reduces the size of the model and computation time required for the analysis. The figures below, Figure 33, Figure 34, Figure 35, depicts all the modules of the simulation.

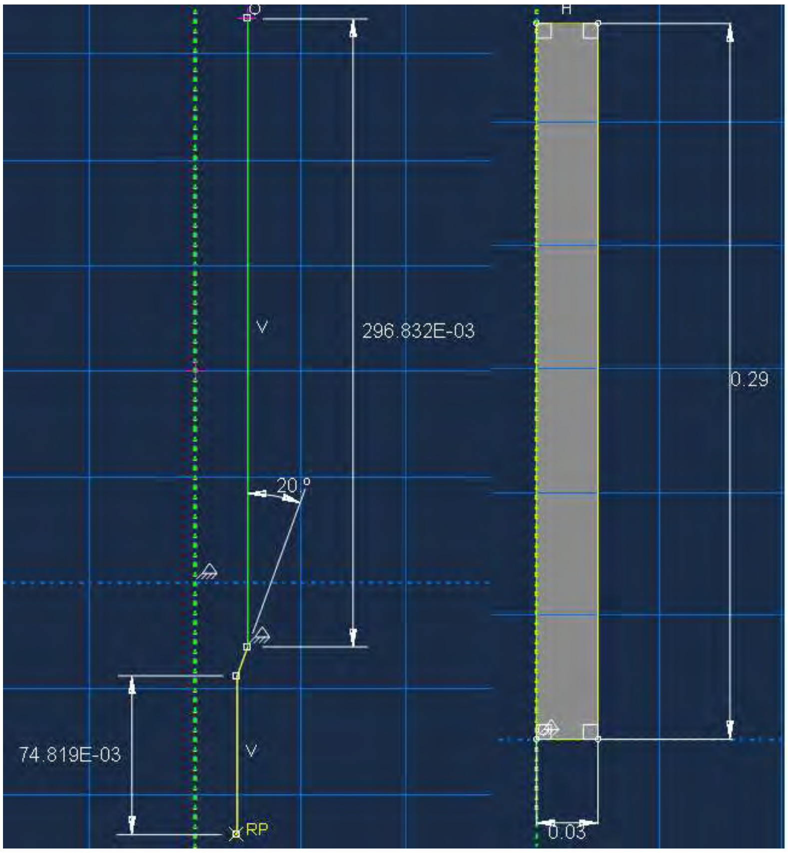

Figure 32: 2D extrusion sketch

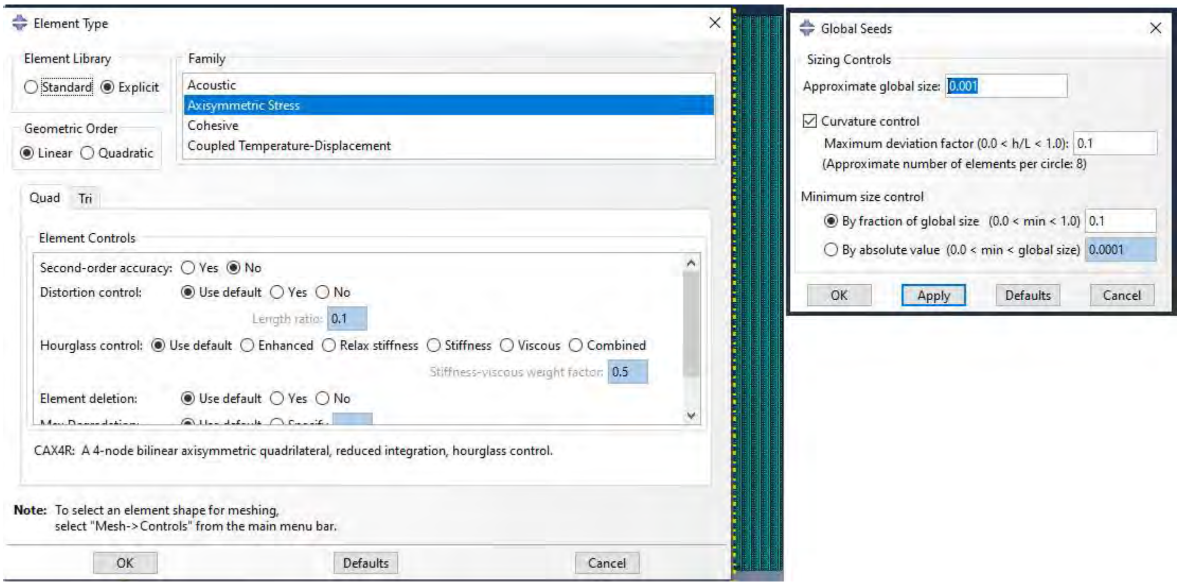

Figure 333: Mesh and element type editing windows

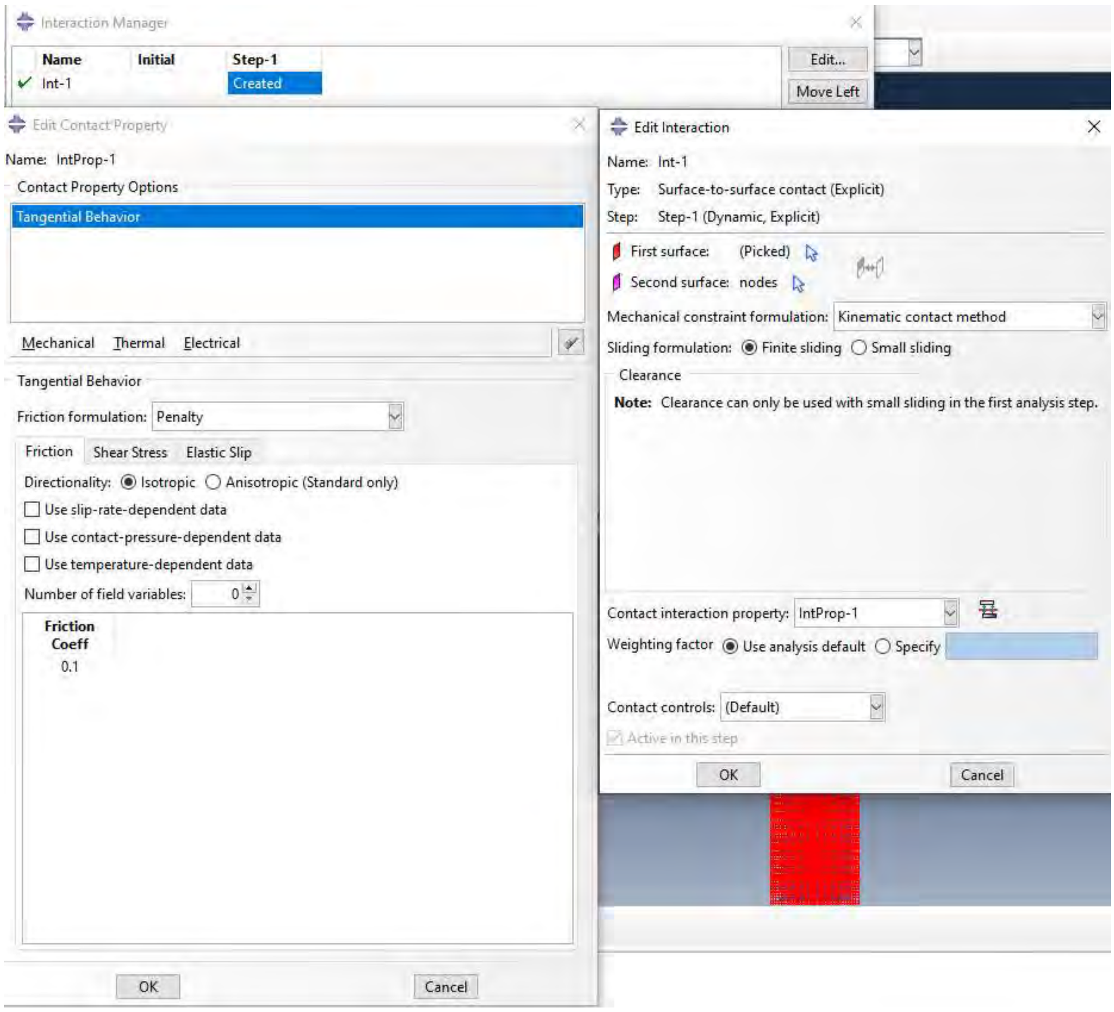

Figure 34: Contact properties and interaction windows

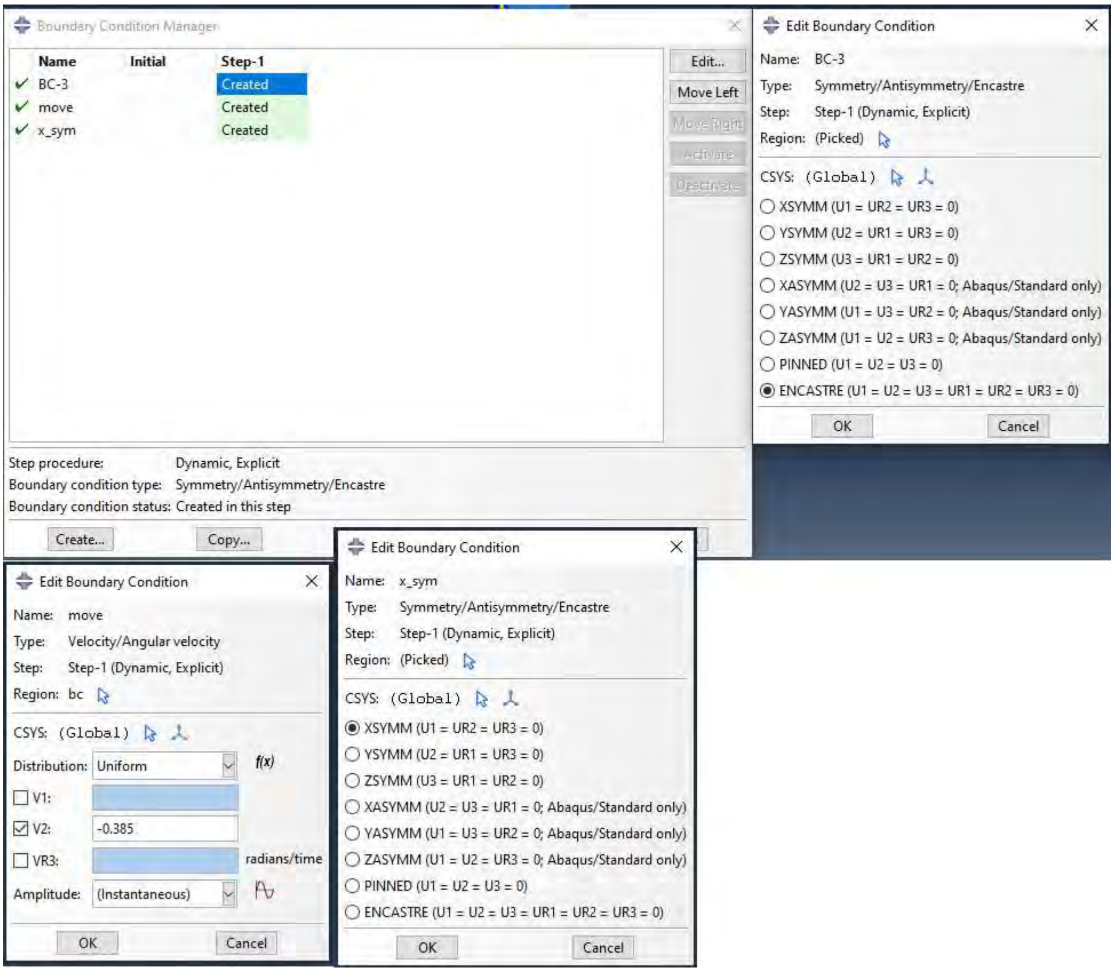

Figure 35: Boundary conditions

4.2. ALE adaptive meshing and mass scaling

Ale adaptive meshing

Adaptive meshing is a process in finite element analysis (FEA) simulation that automatically adjusts the mesh resolution during the simulation, by refining the mesh where it is needed most to improve the accuracy, without using an excessively dense mesh that would slow it down. It works by continuously monitoring the solution and identifying areas where the mesh needs to be refined and the algorithm then adds or removes elements as needed to ensure that the mesh is appropriately refined in regions where the solution is changing rapidly. This makes the simulation more efficient and saves time throughout the computing process.

The "Remeshing Sweep per Increment" and "Frequency" (Figure 36) values are parameters that control the behavior of the adaptive meshing algorithm. The "Remeshing Sweep per Increment" value determines how many times the mesh is updated per time increment. If the value is high, the mesh will be updated more regularly, otherwise, it will be updated less frequently. The "Frequency" value determines the interval between mesh updates, for example, a value of "2" means that the mesh will be updated every other time step. The combination of the "Remeshing Sweep per Increment" and

"Frequency" values determines how aggressively the adaptive meshing algorithm updates the mesh. Values closer to one indicate more frequent updates, while those further apart indicate fewer updates.

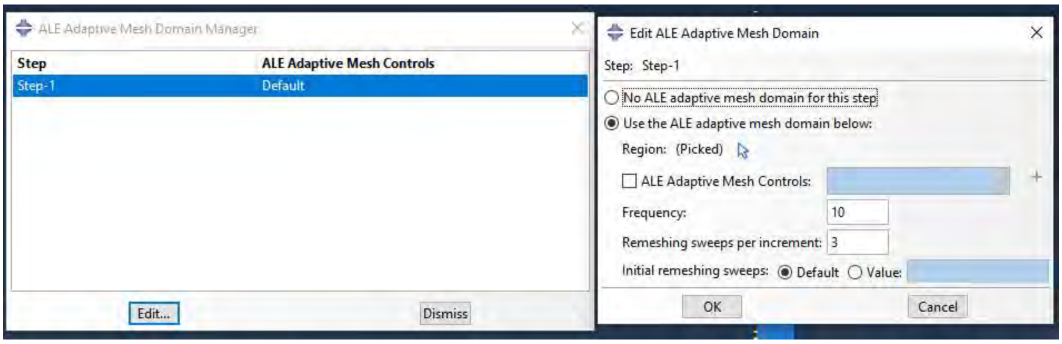

Figure 36: Ale adaptive meshing settings

Mass scaling

Typically, the Explicit Dynamics method is used to two types of issues: the computation of dynamic transient responses and the study of quasi-static problems with complicated nonlinear effects. Due to the fact that the Explicit approach integrates equations in time, the discrete mass matrix in equilibrium equations has an important function in enhancing computing efficiency and precision for both of these issues. In addition to enhancing the efficiency of the solution, if mass scaling is used in the appropriate and timely circumstances, it will also increase the correctness of a certain category of issues. The size and dimensions of these smaller components affect the stable time steps of the solution and cause the dissolution time to substantially rise.

- Scale the mass of the whole model or of particular components (or a collection of elements).
- Scale the mass at the start of the solution or throughout the solution process.
- In multiple-step analyses, the mass scaling mode may be applied to a particular step or stage.

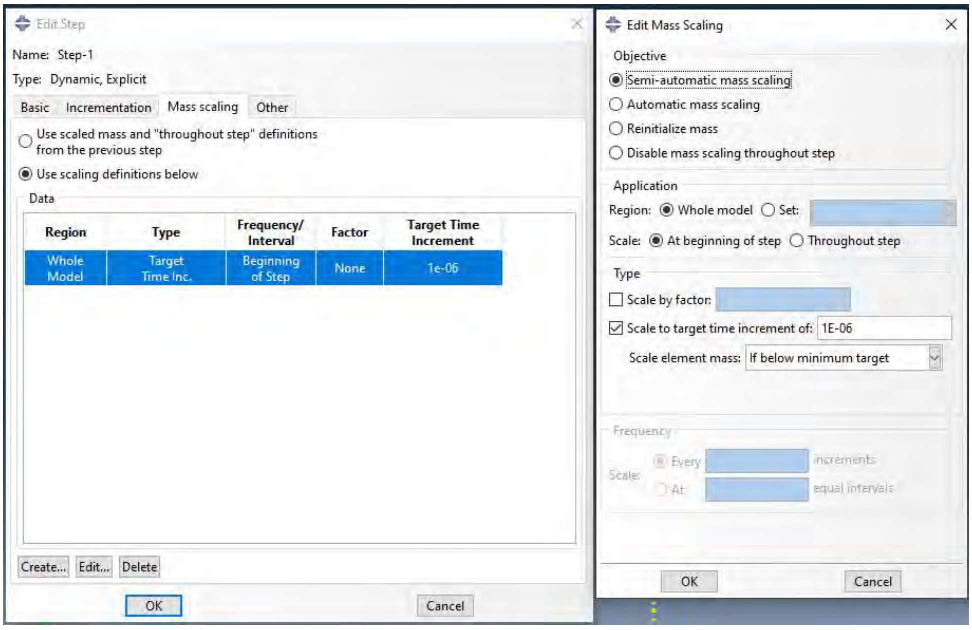

Figure 37: Mass scaling settings

4.4. Dynamic, Temp-Explicit

Coupled temp-disp is used for thermo-mechanical process which does not involve large deformations or is relatively static throughout whereas the dynamic, temp explicit is used specifically for a highly nonlinear process in which elements undergo extreme deformation. The explicit algorithm allows for handling of these large distortions, is computationally expensive and can be used to analyze the deformation, temperature distribution, and other related properties.

Additionally, an element type of coupled temperature-displacement, CAX4RT (Figure 38), is a type of finite element used to model thermal and mechanical behavior simultaneously. Is the same element type as used in dynamic explicit case but with the addition of hourglass control.

When these two methods are combined, a simulation can be conducted to analyze the extrusion process while taking into account both the temperature and mechanical behavior of the material. This can provide a more accurate representation of the material deformation during the extrusion process and help predict potential issues such as material defects or failure points.

Heat generation is a phenomenon that can occur in materials and structures when they are subjected to mechanical or thermal loading and three different techniques can be used to implement it in Abaqus. The first one is using a user-defined subroutine (UMAT), the second one is via built in material models such as Johnson Cook and the third is with built in features that allow the user to include heat generation due to frictional heating, as in this study.

In this case, the default settings were used for heat generation, 0° degrees initial die and 20° degrees predefined billet temperature, (Figure 39).

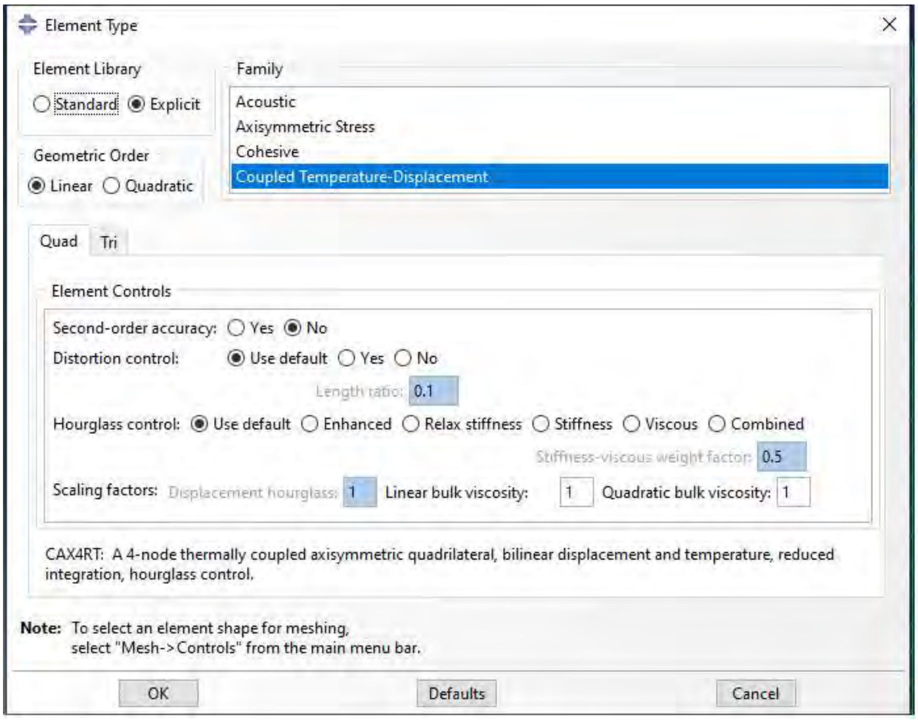

Figure 38: Element type settings

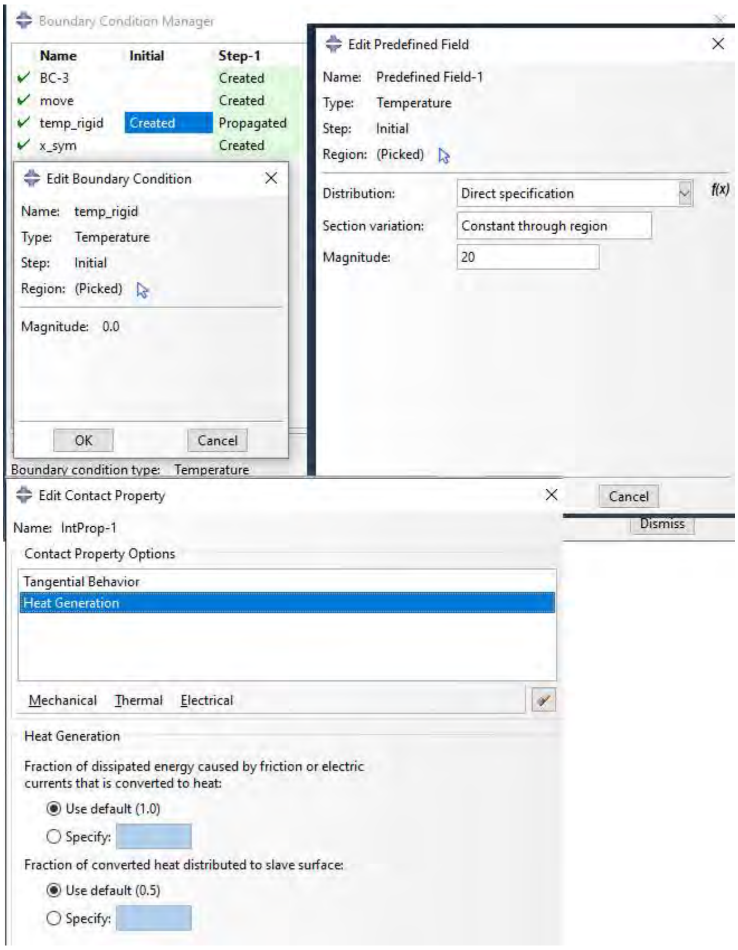

Figure 39: Boundary conditions editing windows

4.5. Results and discussion

SDV_D

The five different simulations that are presented below are:

- Dynamic explicit step analysis, axisymmetric stress, with plasticity property of $\bar{\sigma} = 190\bar{\varepsilon}^{0.21}$ with C_2=50MPa and C_2=20MPa. Initial radius, semi-cone angle and the reduction area is 25mm, 20° and 36%, respectively (Figure 40, Figure 41).
- Dynamic explicit step analysis, axisymmetric stress, with plasticity property of $\bar{\sigma} = 470\bar{\varepsilon}^{0.1}$ with C_2=80MPa. Initial radius, semi-cone angle and the reduction area is 25mm, 20° and 36%, respectively (Figure 42).
- Dynamic, Temp-disp, explicit analysis, coupled temperature-displacement, with plasticity and material properties as described at Chapter 3, 3D extrusion and C_2=40MPa. Initial radius, semi-cone angle and the reduction area is 25mm, 20° and 36%, respectively (Figure 43).
- Dynamic explicit step analysis, axisymmetric stress, with plasticity property of $\bar{\sigma} = 190\bar{\varepsilon}^{0.21}$ with C_2=40MPa. Initial radius, semi-cone angle and the reduction area is 20mm, 26° and 36%, respectively (Figure 44).
- Dynamic explicit step analysis, axisymmetric stress, with plasticity property of $\bar{\sigma} = 470\bar{\varepsilon}^{0.1}$ with C_2=80MPa. Initial radius, semi-cone angle and the reduction area is 20mm, 26° and 36%, respectively (Figure 45).

Approximate values of the C_2 parameter of the Cockcroft-Latham criterion were used in the simulations below. The output visualization of SDV_D is a user defined output of the criterion, in which when value reaches 1 the element is deleted. As shown in the below figures the most severe damage is conducted in the case five, as in the paper. Also, it is clearly visible that decreasing the C_2 parameter, the damage increases as in the first two figures.

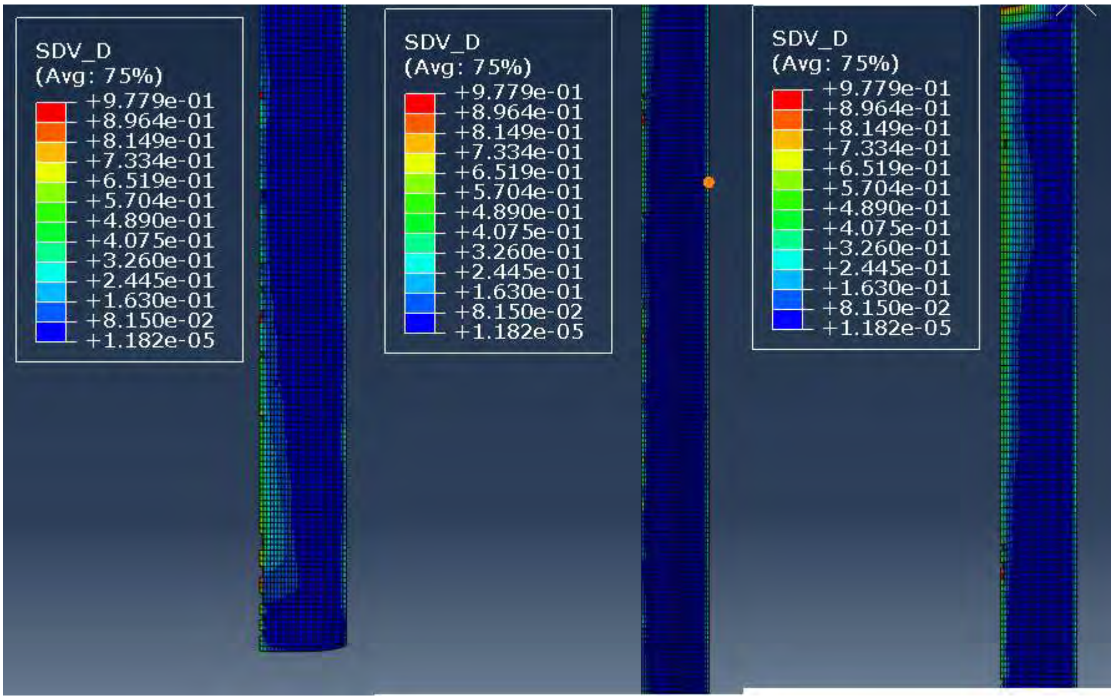

Figure 40: Aluminium 50Mpa C_2, 25mm, 20°

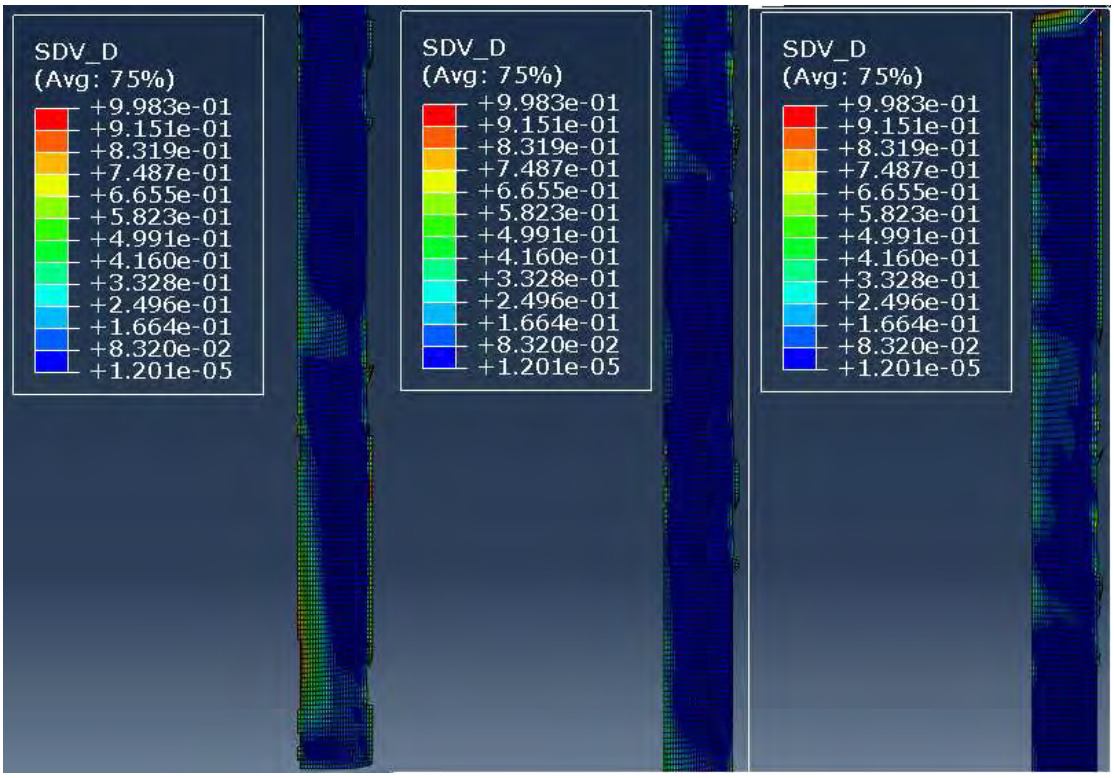

Figure 41: Aluminium 20Mpa C_2, 25mm, 20°

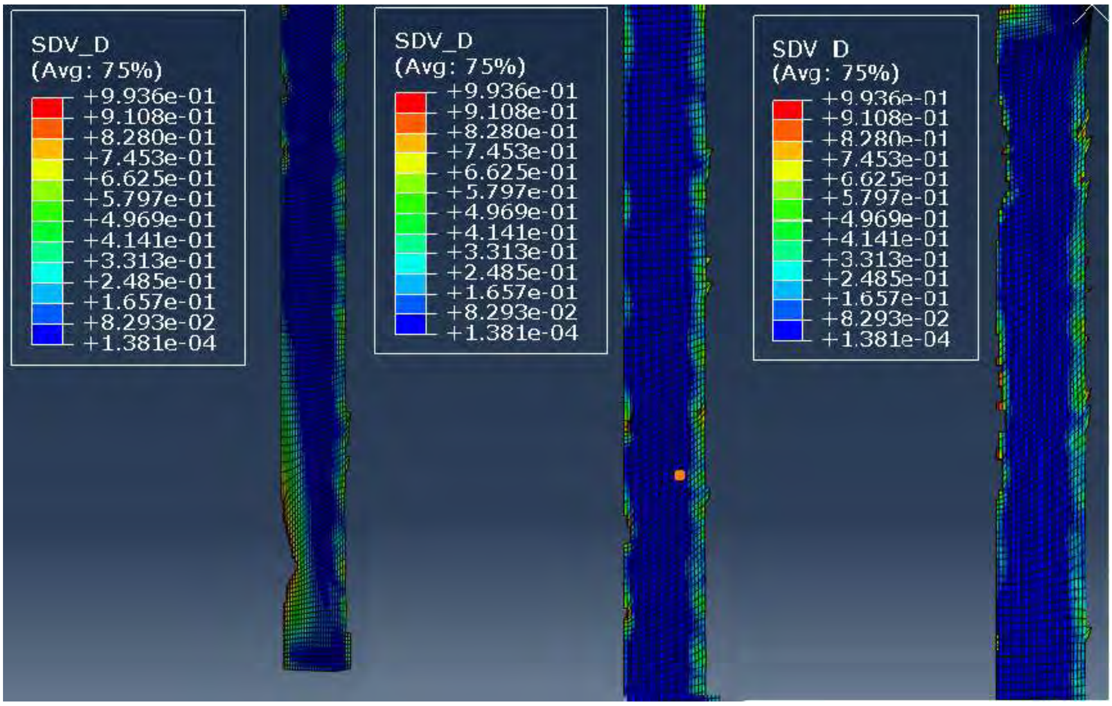

Figure 42: Steel 80Mpa C_2, 25mm, 20°

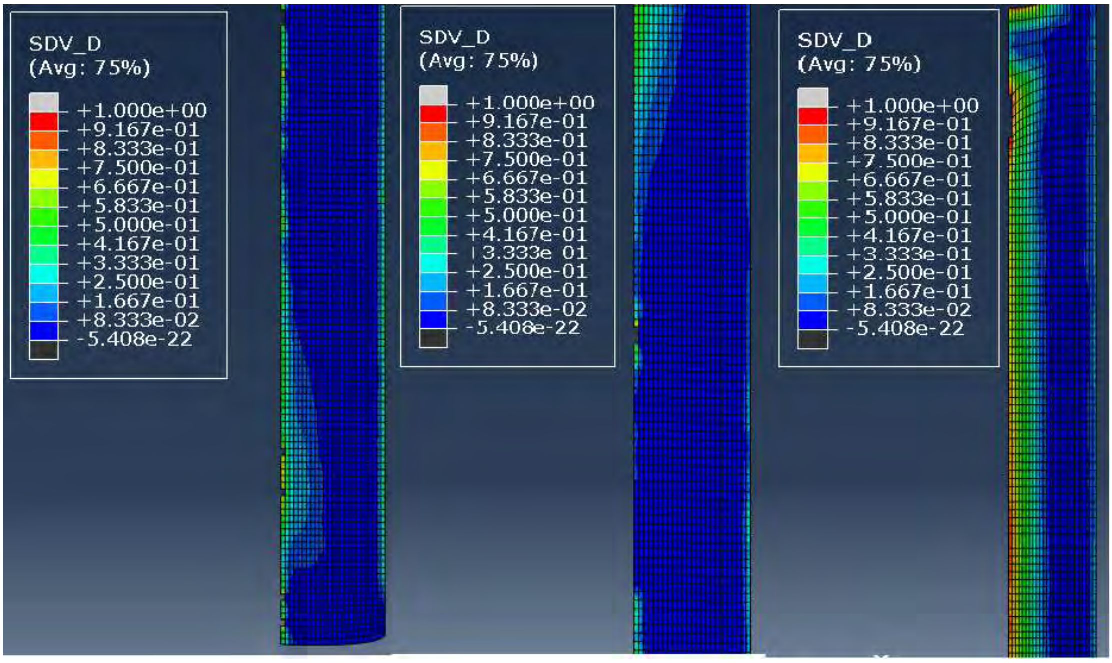

Figure 43: Coupled-Temp 40Mpa C_2, 25mm, 20°

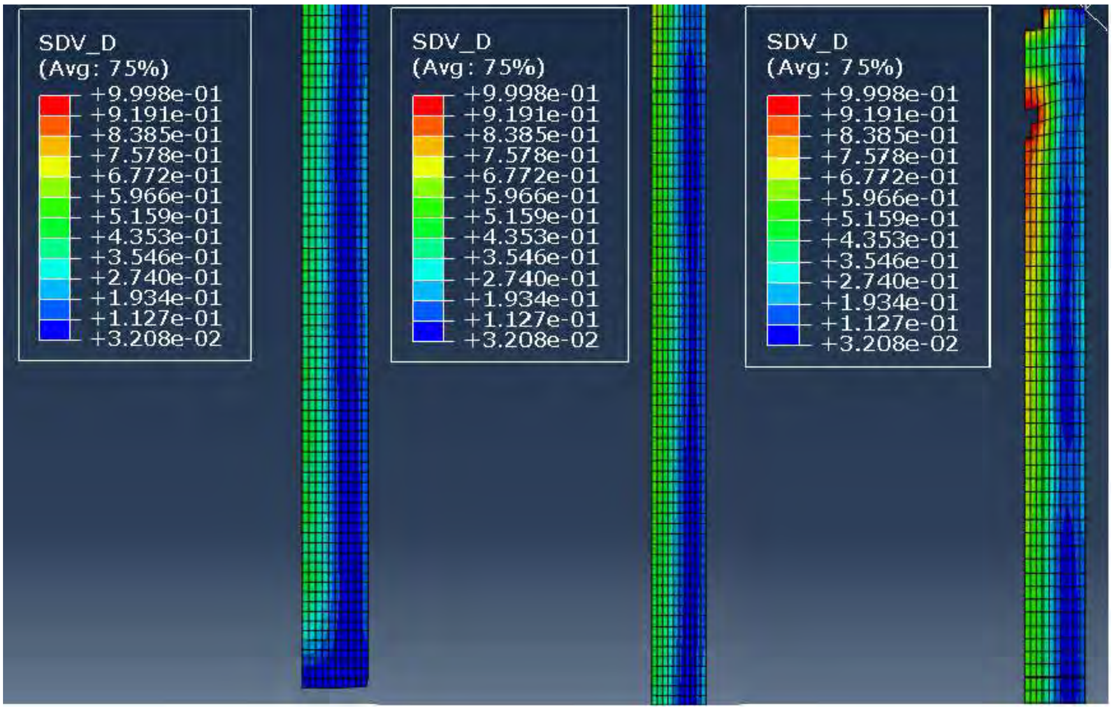

Figure 44: Aluminium 50Mpa C_2, 20mm, 18°

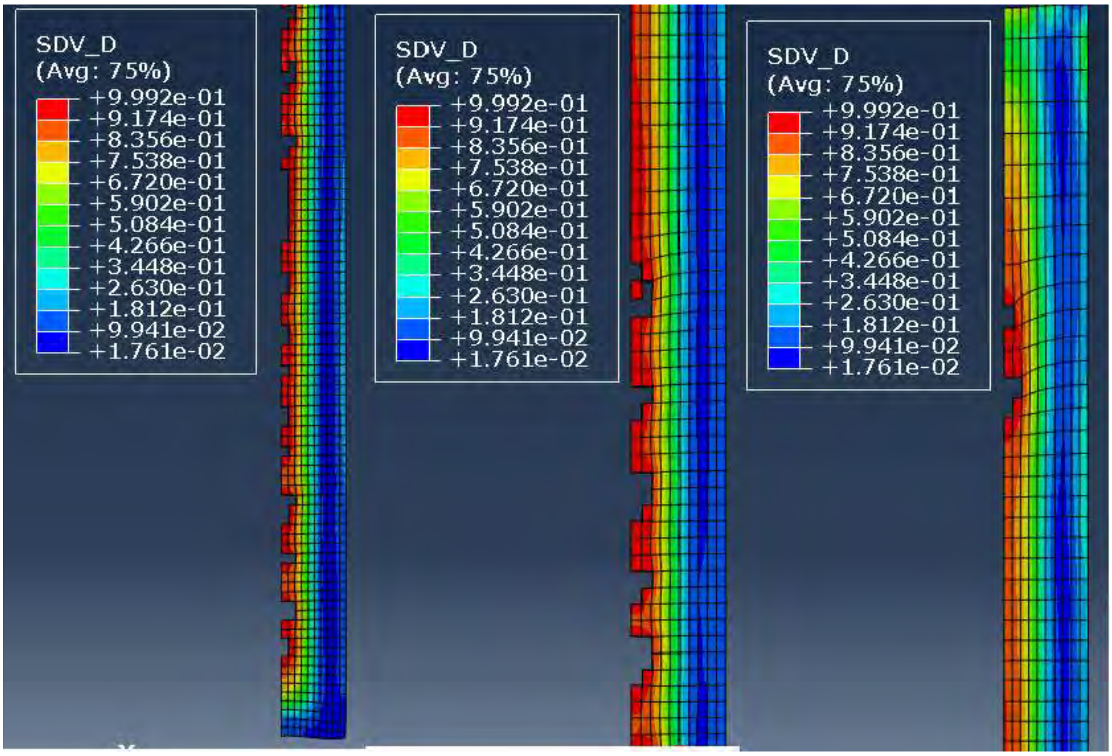

Figure 45: Steel 80Mpa C_2, 20mm, 18°

Force-Displacement

The history output field was used to collect the essential information for making the force-displacement plots seen below. The overall force was determined by adding the individual RF2 values produced by the nodes located on the billet's surface. Just one node's displacement was required for this computation, and the absolute values of the RF2 outputs were utilized. Each set of data was filtered using a unique cutoff frequency. By adjusting the filter's cutoff frequency, certain frequencies are accepted while rejecting all others.

Figure 47 shows that the maximum force occurred in the second case with the steel material having a value of 6.205E+06 N. This data is useful for optimizing processing settings and bettering component design, since it gives insight on the mechanical behavior of the billet during testing.

The mechanical behavior of a material may be shown by plotting the force exerted on the material with the displacement in a force-displacement plot as localized deformation caused by high radial stresses, more effort is needed to drive the material through the die as it is being formed. High stress regions at risk of developing the central burst defect may be seen in different force-displacement plots of nodes.

The extrusion process may be optimized to avoid the center burst defect by inspecting the force-displacement plot. Die design, material characteristics, and extrusion settings are all modifiable factors that may mitigate material stress and avoid the high radial stresses that might cause the center burst fault from forming.

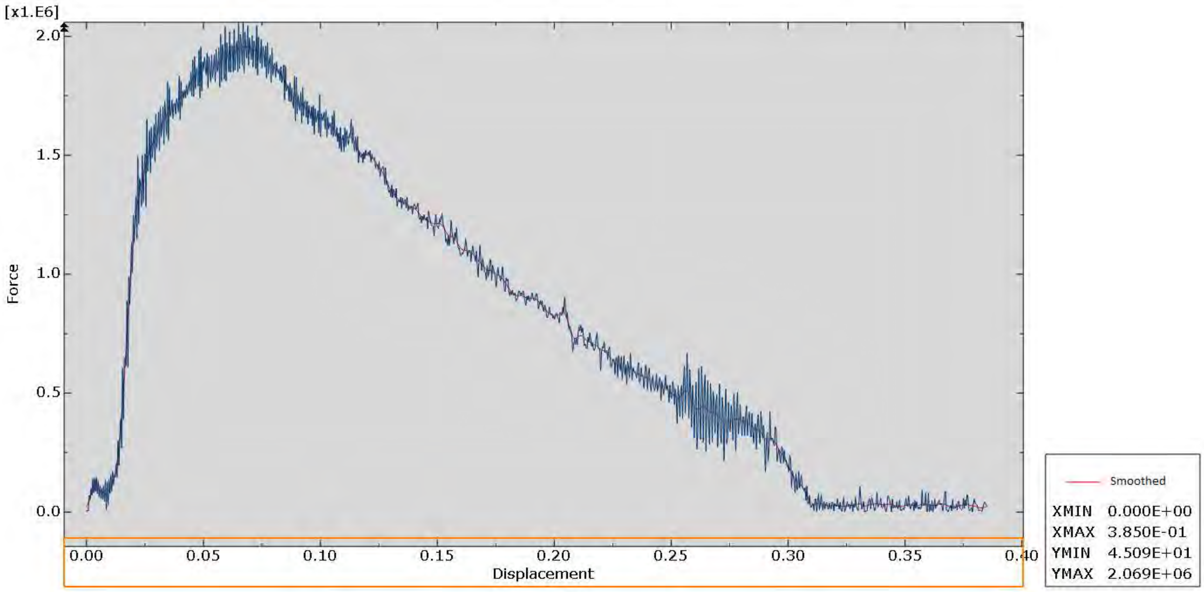

Figure 46: Aluminium 25mm, 20°

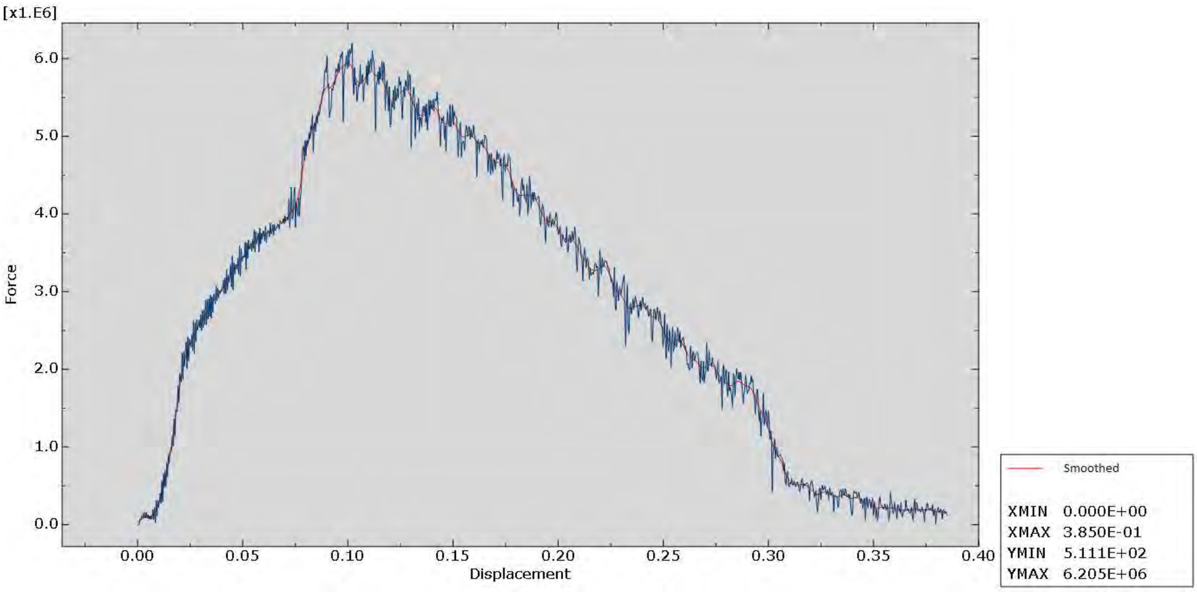

Figure 47: Steel 25mm, 20°

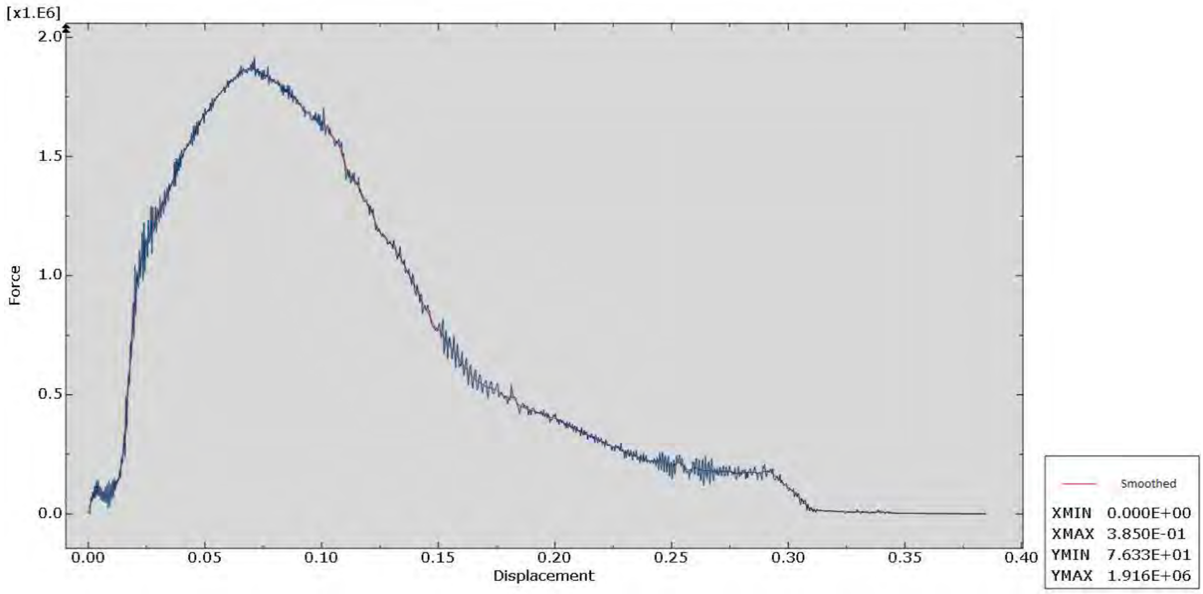

Figure 48: Coupled-Temperature 25mm, 20°

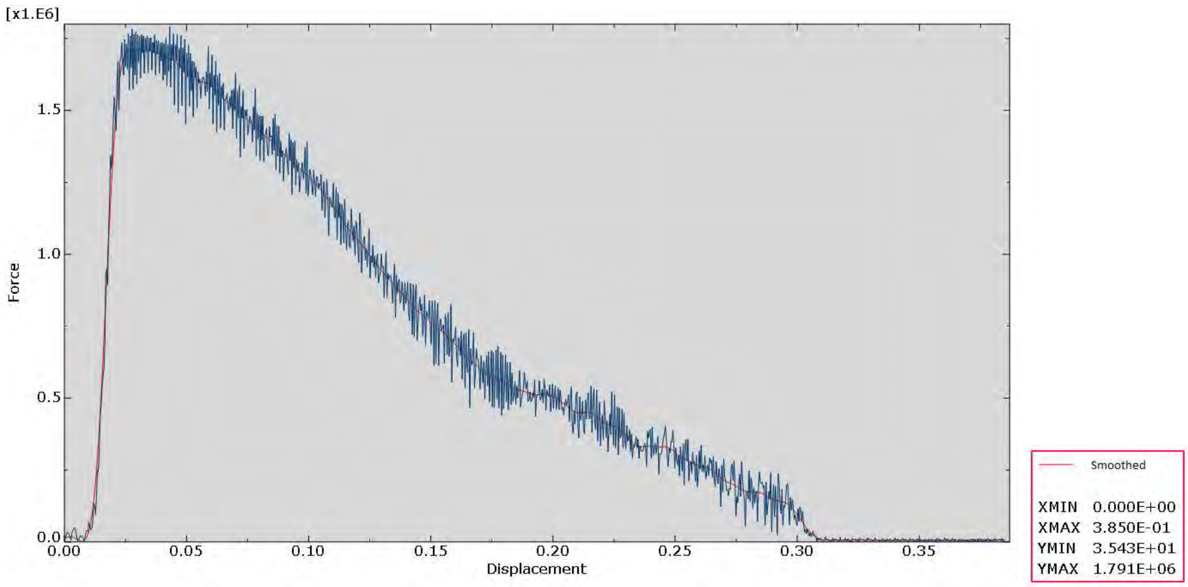

Figure 49: Aluminium 20mm, 18°

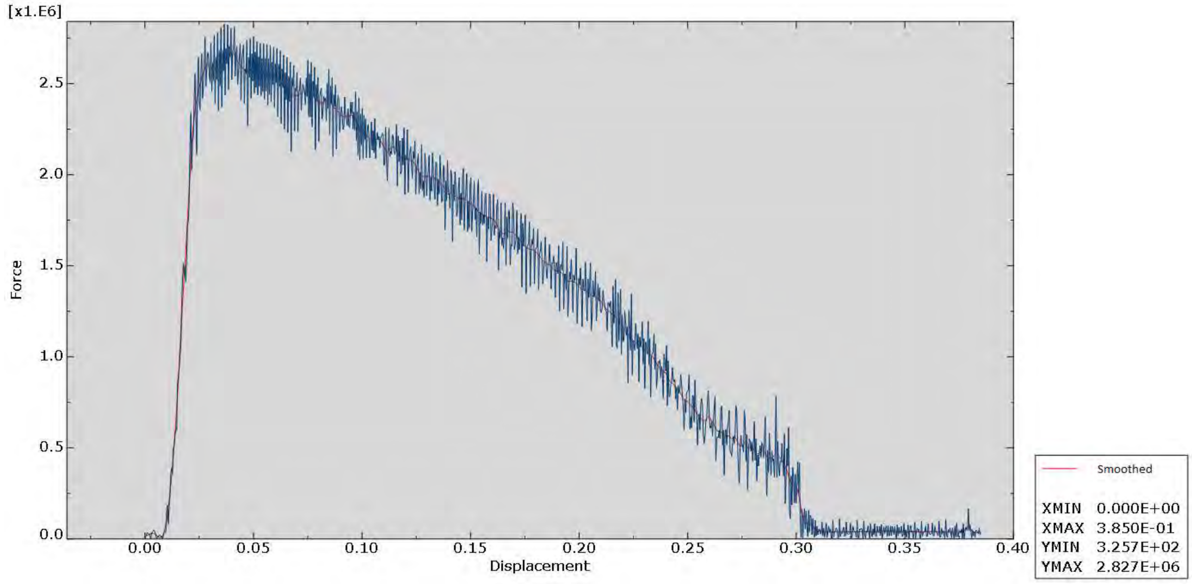

Figure 50: Aluminium 20mm, 18°

Stress-strain

An integration point was chosen for each figure in the stress-strain analysis of the billet. Information for the graphs, including the MISES stress components, was retrieved from the output's PEEQ and S fields. The information was then filtered in the same way as the force-displacement graphs.

Important data regarding the billet's mechanical behavior at various phases of processing may be acquired from stress-strain plots. To optimize processing settings and develop components with enhanced mechanical properties, understanding the material's deformation and fracture behavior via stress-strain plot analysis is crucial.

The stress-strain plots revealed that the second case resulted in the greatest amount of strain for the steel material. Overall, the data from the stress-strain plots may be utilized to better understand the billet's mechanical behavior during processing, which in turn can be used to fine-tune the processing itself redesign individual components and avoid the occurrence of defects.

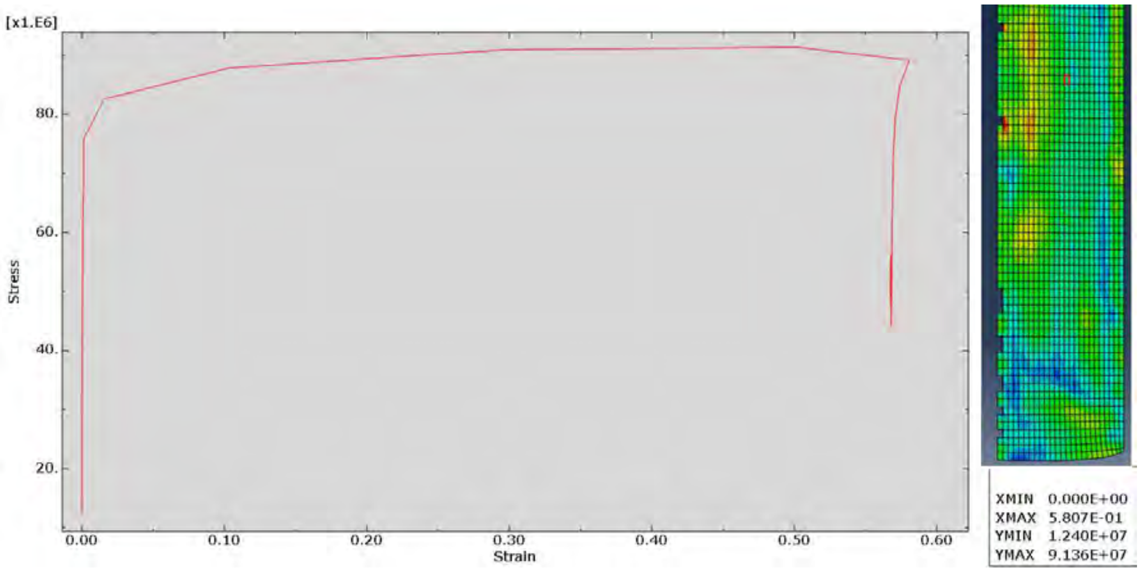

Figure 51: Aluminium 25mm, 20°

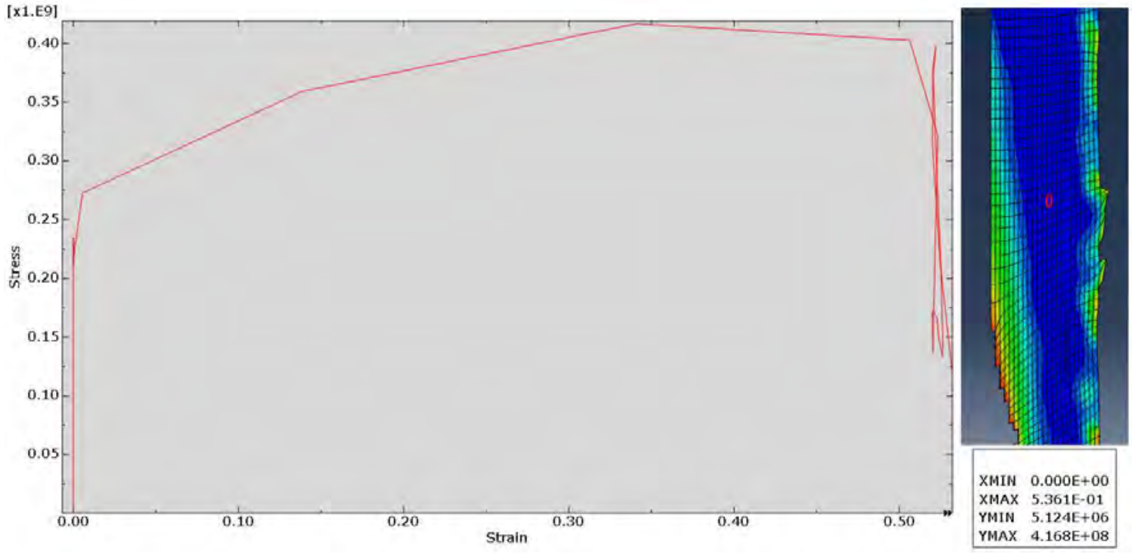

Figure 52: Steel 25mm, 20°

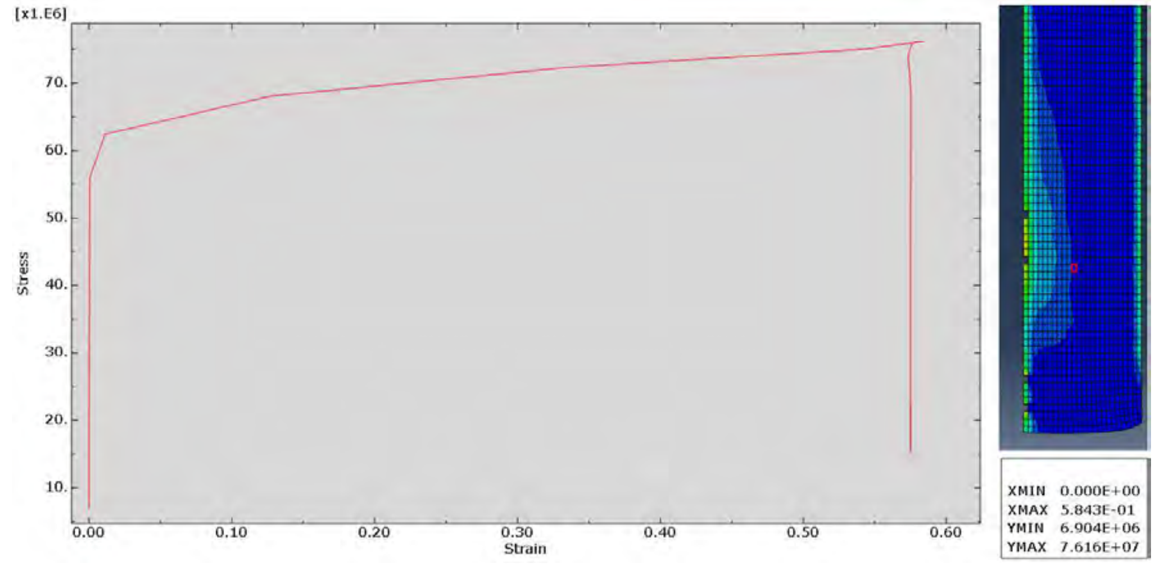

Figure 53: Coupled-Temperature 25mm, 20°

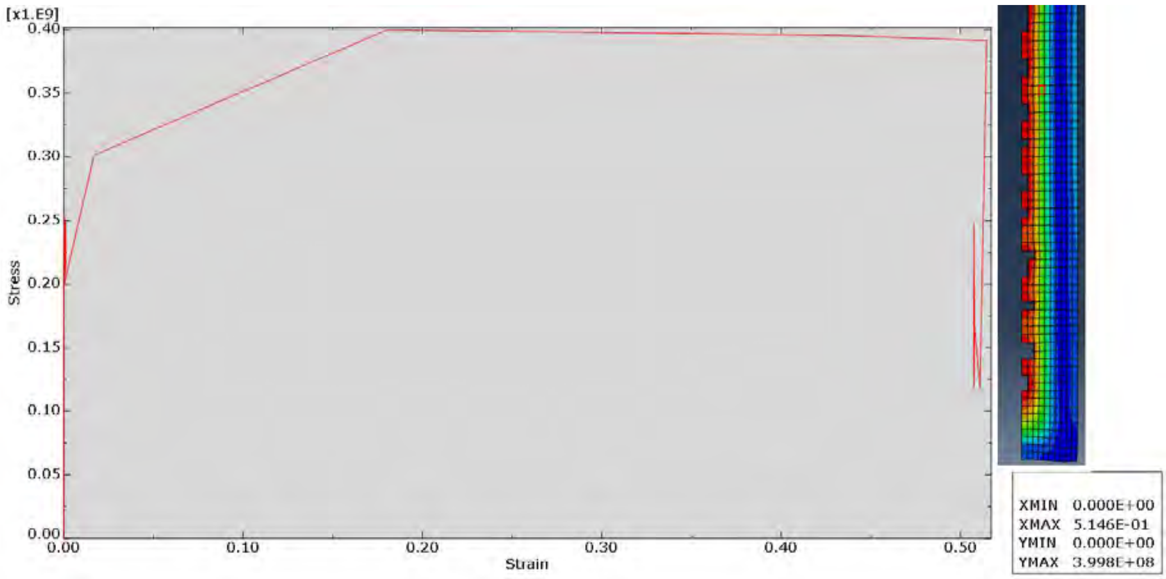

Figure 54: Steel 20mm, 18°

Chapter 5. Suggestions for further study

The simulations in chapter 4 are considered approximate estimates and for no reason can be used to predict central burst defects. Also, trial and error method was used throughout the simulations and a Matlab script was written in order to run multiple jobs simultaneously on each processor, as can be seen on the APPENDIX. This thesis projects the foundation to the central burst simulation and prediction, but a more in depth research should be conducted.

As someone wishes to elaborate more in the study of the central burst defects, has to consider the following roadmap. In order to obtain more accurate scientific solutions, it is necessary to calibrate three jobs, namely:

- Plastic work criterion
- Tensile experiments
- Subroutine debugging

5.1. Plastic work criterion

For the calculation and simulation of the central burst defects, an Abaqus subroutine must be written with the failure criterion that will be used. In the paper (Choi et al., 2010), two criteria are being used:

$$\int_0^{\bar{\varepsilon}f} \bar{\sigma} \mathrm{d}\bar{\varepsilon} = C_1 \qquad (2)$$

$$\int_0^{\bar{\varepsilon}f} \sigma_1 \mathrm{d}\bar{\varepsilon} = C_2 \qquad (3)$$

The first one is the failure criterion based on the plastic work and the second one is the Cockcroft-Latham. As can be seen in the Appendix (Cockcroft-Latham criterion), a VUDSFLD subroutine for the second criterion is being presented. For implementing it the following changes must be done in .inp file, the *Depvar below the density properties and the *User defined field below the plasticity data. The number 80.0e6 is the C_2 parameter of the criterion.

```
*Depvar, delete=3
    3,
1,PEEQ,"Equivalent plastic strain"
2,D,"Damage variable"
3,STATUS,"Failure status"


*User defined field, properties=1
80.0e6
```

All the simulations above were considered by only the second criterion, so the first one must be implemented also. An effort was made [Combined subroutine] for implementing the equation (1), supposing the $\bar{\sigma}$ value is the flow stress $\bar{\sigma} = 475\bar{\varepsilon}^{0.1}$.

*User defined field, properties=2

433.0e6 (C1)

20.0e6 (C2)

In continuation of the previous simulation the billet experienced deletion in the front element row and loses convergence in the semi-cone angle, due to the false estimation of the Cockcroft-Latham parameter and flaws in subroutine, and for that reason it was rejected.

5.2. Tensile experiment

After implementing the criteria, tensile experiments must be done in order to scientifically calculate the values of C_2.

Description of the steps for completing a tensile stress failure criteria analysis in Abaqus (Figure 55):

1. Laboratory experiment: The first step is to conduct a tensile test on a dog bone specimen in the laboratory accordance with a recognized standard test method, such as ASTM E, and measure the force displacement data.
2. Calculate stress-strain curve: Based on the force-displacement data from the laboratory experiment, the stress-strain curve of the material is then calculated.
3. Acquire plastic data: The stress-strain curve may be used to determine the material's plastic characteristics, including its yield strength and ultimate tensile strength.
4. Abaqus model: With the material properties obtained from the laboratory experiment, an Abaqus model of the dog bone specimen can be created.
5. Failure criteria value: In Abaqus, the failure criteria value C_2 is used to define the stress at which the material is considered to have failed. The value of C_2 can be changed in the model to approach the experiment force-displacement curve and to accurately predict the behavior of the material under tensile loading.
6. Model validation: To validate the Abaqus model, the results of the analysis should be compared with the experimental data obtained from the laboratory experiment.
7. Determine the correct value of C_2 : By adjusting the value of C_2 and comparing the results with the experimental data, the correct value of C_2 that accurately predicts the tensile stress failure behavior of the material can be determined.
8. Conclusion: Finally, the correct value of the failure criteria C_2 can be concluded, and the Abaqus model can be used to predict the behavior of the material under tensile loading and to determine the tensile stress failure criteria of the material.

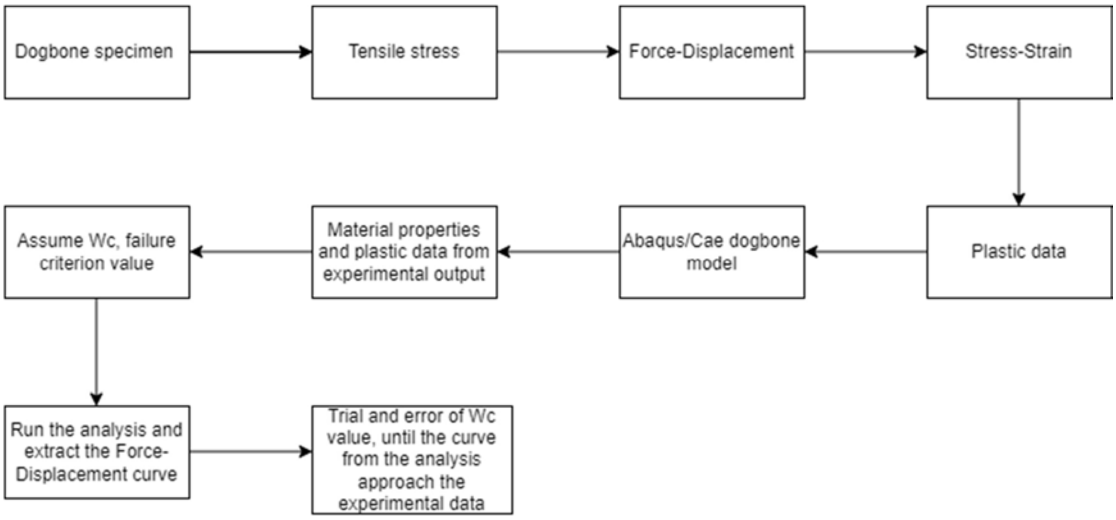

Figure 55: Tensile stress workflow

To better comprehend how materials respond to stress, it is useful to visualize a dogbone tensile stress using a failure criteria subroutine. A static general analysis with a duration of 1 second was carried out utilizing kinematic coupling as the constraint parameter. U1, U2, and UR3 were the constrained degrees of freedom that helped improve the simulation's accuracy. Specimen were subjected to an encastre load at the base and a 0.005 upward direction displacement load at the top.

In Figure 56a, the analysis has not yet begun, therefore the SVD D (the failure criterion value) is 0. Figure 56b shows the development of the SVD D and the localization of damage inside the billet's core. After the SVD D reaches its maximum value of 1, a central fracture forms in the third figure (Figure 57a). The entire amount of the damage is seen in the final figure (Figure 57b), in which the material fully separates.

Visualizing a dogbone tensile stress with a failure criteria subroutine provides useful insights on the material's behavior under stress. The simulation is a useful tool for learning about the behavior of materials and developing new ones with improved resistance to stress and damage because it provides a realistic depiction of the material's behavior by combining constraint parameters with algorithms.

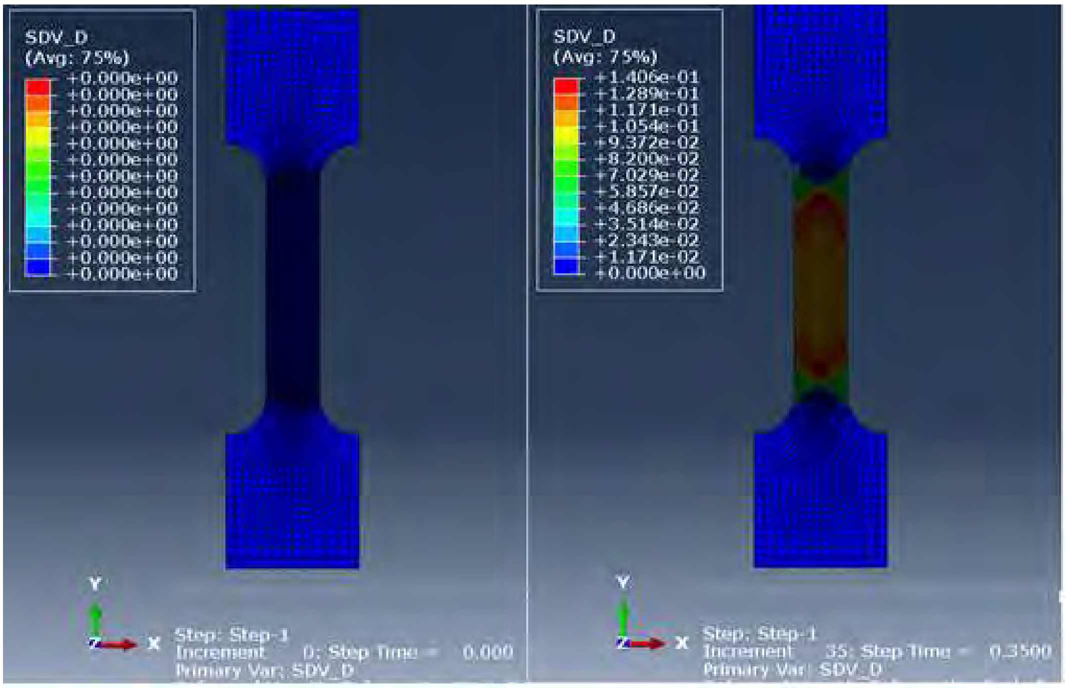

Figure 56(a,b): Tensile stress

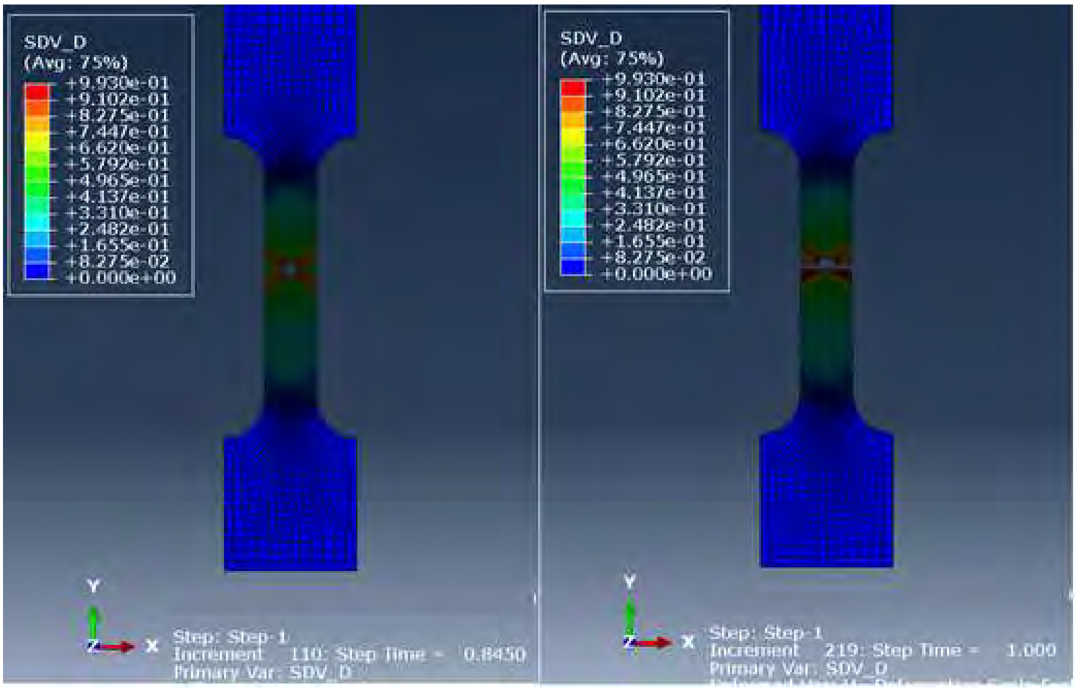

Figure 57(a,b): Tensile stress

5.3. Subroutine debugging

The term "debugging" refers to the process of finding and fixing flaws in a program's code so that it performs as designed. The purpose of debugging is to fix bugs and enhance the quality of the program as a whole. To prevent accidental deletion of components and interference with Abaqus commands and solvers, it is essential to do thorough debugging after coding a new combined subroutine. The fundamental steps of debugging are:

- Detection of Mistakes

As software is being tested or used, developers, testers, and end users all report any issues they come across. In order to fix an issue, programmers must first pinpoint where it exists in the code. This may be a long and boring task.

- Evaluation of Errors

Engineers investigate the glitch by documenting every program state change and data value. In addition, they rank the severity of the issue and its effect on the software's operation while deciding the order in which to correct the bugs. A schedule for correcting bugs is also established by the software team in accordance with the project's objectives.

- Repair and verify

After the problem repair, developers will perform tests to confirm the product is functioning as planned. To see whether the issue comes again, they could create some fresh tests.

Chapter 6. Conclusion

This thesis presents a complete analysis of extrusion operations using the Abaqus software. The emphasis of the third chapter was on a three-dimensional extrusion process, which is a difficult procedure involving substantial material deformation. This chapter's findings revealed the significance of the force-displacement plot, the temperature output, and the stress-strain plots for analyzing the performance of an extrusion process. These findings provide light on the behavior of the material throughout the extrusion process and may serve as a basis for future study in this area.

The fourth chapter examined the central burst defect simulated in a 2D extrusion, implementing a subroutine using the Cockcroft-Latham failure criteria. Also, a dynamic explicit analysis of steel and aluminum materials for two distinct die and billet geometries was conducted. This chapter's findings illustrated the effect of the Cockcroft-Latham failure criteria on the end product and established that the criterion is an essential extrusion process parameter. In addition, a dynamic explicit temperature investigation was carried out on aluminum, and the findings were also displayed.

Quite apart from the substantial contributions of this thesis to the topic of extrusion processes, further research is still required. Particularly, it is necessary to combine the Cockcroft-Latham failure criteria with an additional plastic work failure criterion. In addition, a tensile stress test must be performed to determine the Cockcroft criterion's parameter, and the combined subroutine must be debugged to confirm its correctness. The results of this thesis may be utilized to improve the quality of extruded products and increase the efficiency of the extrusion process after these difficulties have been resolved.

This thesis concludes by providing significant insights into the extrusion process and highlighting the effect of various factors on the end result. These findings may constitute the foundation for future study in this field. To increase the accuracy of the results and assure the reliability of the findings, it is necessary to solve the difficulties outlined in this study. Through more research, the results of this thesis can be applied to create more efficient and effective extrusion simulations and to enhance the quality of the end products.

References

A. M. Freudenthal. (1950). *The Inelastic Behavior of Engineering Materials and Structures*. John Wiley & Sons.

A. S. Wifi, A. N. A.-H. and E. A. (1998). *Computer aided evaluation of workability in bulk forming processes*. J. Mater. Process. Technol.

Abaqus Example Problems Guide. (n.d.). Abaqus 6.14. http://130.149.89.49:2080/v6.14/books/exa/default.htm

Abaqus/CAE User's Guide. (n.d.). Abaqus 6.14. http://130.149.89.49:2080/v6.14/books/exa/default.htm

Akram, S., Jaffery, S. H. I., Khan, M., Fahad, M., Mubashar, A., & Ali, L. (2018). Numerical and experimental investigation of Johnson–Cook material models for aluminum (AL 6061-t6) alloy using orthogonal machining approach. *Advances in Mechanical Engineering*, *10*(9), 1–14. https://doi.org/10.1177/1687814018797794

Aravas, N. (1986). THE ANALYSIS OF VOID GROWTH THAT LEADS TO CENTRAL BURSTS DURING EXTRUSION. In *J. Mech. Phys. Solids* (Vol. 34, Issue 1).

Avitzur, B. (1968). Analysis of Central Bursting Defects in Extrusion and Wire Drawing. *Journal of Engineering for Industry*, *90*(1), 79–90. https://doi.org/10.1115/1.3604608

Choi, J. S., Lee, H. C., & Im, Y. T. (2010). A study on chevron crack formation and evolution in a cold extrusion. *Journal of Mechanical Science and Technology*, *24*(9), 1885–1890. https://doi.org/10.1007/s12206-010-0605-z

Clift, S. E., Hartley, P., Sturgess, C. E. N., & Rowe, G. W. (1990). Fracture prediction in plastic deformation processes. *International Journal of Mechanical Sciences*, *32*(1), 1–17. https://doi.org/10.1016/0020-7403(90)90148-C

Engineers Edge, https://www.engineersedge.com/properties_of_metals.htm

Github Forum, https://github.com/davidmorinNTNU/ABAQUS_subroutines/tree/main/MJC_example

Groover P. Mikell, Fundamentals of Modern Manufacturing: Materials, Processes, and Systems, John Wiley & Sons, 2010

Kathirgamanathan, P., Neitzert, T., Kathirgamanathan, P., & Neitzert, T. (2006). *Modeling of metal extrusion using Abaqus Modelling of metal extrusion using ABAQUS*. https://www.researchgate.net/publication/265088009

Kim, H.-S., Im, Y.-T., & Geiger, M. (1999). Prediction of Ductile Fracture in Cold Forging of Aluminum Alloy. *Journal of Manufacturing Science and Engineering*, *121*(3), 336–344. https://doi.org/10.1115/1.2832686

P. Kubík , J. Petruška , J. Hůlka, DUCTILE FRACTURE CRITERIA IN PREDICTION OF CHEVRON CRACKS, 9 – 12 May 2011, 17th International Conference ENGINEERING MECHANICS 2011 Svratka, Czech Republic

M. G. Cockcroft and D. J. Latham. (1968). Ductility and workability of metals. *J. Inst. Met*, 33–39.

McAllen, P., & Phelan, P. (2005a). A method for the prediction of ductile fracture by central bursts in axisymmetric extrusion. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, *219*(3), 237–250. https://doi.org/10.1243/095440605X16820

Metal Extrusion. (n.d.). Thelibraryofmanufacturing.Com. https://thelibraryofmanufacturing.com/extrusion.html

Ranjan Yadav, R., Dewang, Y., Raghuwanshi, J., & Sharma, V. (2018). *ScienceDirect Finite element analysis of extrusion process using aluminum alloy*. www.sciencedirect.com

Saanouni, K., Mariage, J. F., Cherouat, A., & Lestriez, P. (2004). Numerical prediction of discontinuous central bursting in axisymmetric forward extrusion by continuum damage mechanics. *Computers & Structures*, *82*(27), 2309–2332. https://doi.org/10.1016/j.compstruc.2004.05.018

Sebek, F., Kubik, P., & Petruska, J. (2015a). Chevron crack prediction using the extremely low stress triaxiality test. *MM Science Journal*, *2015*(June), 617–621. https://doi.org/10.17973/MMSJ.2015_06_201518

Serope Kalpakjian, S. S. (2013). *Manufacturing Engineering & Technology* (7th ed.). Pearson.

Zeynep Parlar, C. G. G. C. S. M. O. M. O. I. (2015). *Extrusion die and attachment design with self lubrication*. Instabul techincal university.

Zimerman, Z., & Avitzur, B. (1970). Analysis of the Effect of Strain Hardening on Central Bursting Defects in Drawing and Extrusion. *Journal of Engineering for Industry*, *92*(1), 135–145. https://doi.org/10.1115/1.3427698

Zimerman, Z., Darlington, H., & Kottcamp, E. H. (1971). Selection of Operating Parameters to Prevent Central Bursting Defects During Cold Extrusion. In *Metal Forming: Interrelation*

Appendix

VUSDFLD subroutine

The subroutine VUSDFLD is written in Fortran and is used to model the damage and fracture of materials in a simulation (Github Forum). It takes multiple inputs, including the number of blocks (**NBLOCK**), the number of state variables (**NSTATEV**), the number of field variables (**NFIELDV**), the number of properties (**NPROPS**), the number of direct stresses (**NDIR**), and the number of shear stresses (**NSHR**).

The subroutine performs the following tasks:

1. Reads in material properties, stress tensor, and equivalent plastic strain from ABAQUS using **vgetvrm** subroutine.
2. Computes the equivalent plastic strain increment and updates the damage variable using the computed stress and plastic strain.
3. Updates the state variables (plastic strain and damage) and checks for fracture. If the damage exceeds 1, the fracture state is set to 0, otherwise, it is set to 1.
4. Returns the updated state variables to ABAQUS.

The values of **SIG1, damage,** and **dp** are updated during the subroutine. First, the equivalent plastic strain and the stresses from ABAQUS are read using the **vgetvrm** subroutine. Then, **dp** is updated as the difference of equivalent plastic strain between the current and previous steps. **SIG1** is updated as the maximum principal stress of each block, which is computed using the vsprinc subroutine. The damage is updated by adding the product of the maximum of zero and the maximum principal stress divided by the critical parameter C_2 to the previous damage. Finally, the statenew is updated as the equivalent plastic strain, the updated damage, and the fracture flag, which is set to 0 if the damage is greater than or equal to 1, otherwise it is set to 1.

Cockcroft-Latham subroutine

```
!$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
!$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
!     Subroutine VUSDFLD
!$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
!$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
      subroutine VUSDFLD(
     + NBLOCK, NSTATEV, NFIELDV, NPROPS, NDIR, NSHR,
     + JELEM, KINTPT, KLAYER, KSECPT,
     + STEPTIME, TOTALTIME, DT, CMNAME,
     + COORDMP, DIRECT, T, CHARLENGTH, PROPS,
     + STATEOLD,STATENEW,FIELD)
      include 'vaba_param.inc'
!-----------------------------------------------------------------------
!-----Declaration ABAQUS variables
!-----------------------------------------------------------------------
      dimension JELEM(NBLOCK),COORDMP(NBLOCK,*),DIRECT(NBLOCK,3,3),
     .          T(NBLOCK,3,3),CHARLENGTH(NBLOCK),PROPS(NPROPS),
     .          STATEOLD(NBLOCK,NSTATEV),STATENEW(NBLOCK,NSTATEV),
     .          FIELD(NBLOCK,NFIELDV)
      character*80 CMNAME
!-----Data from ABAQUS
      dimension stressdata(maxblk*(ndir+nshr))
```

```fortran
      integer jSData(maxblk*(ndir+nshr))
      character*3 cSData(maxblk*(ndir+nshr))
      integer jStatus
      dimension peeqdata(maxblk)
      integer jPData(maxblk)
      character*3 cPData(maxblk)
!-----------------------------------------------------------------------
!-----Declaration internal variables
!-----------------------------------------------------------------------
      integer i
      real*8 s(NBLOCK,NDIR+NSHR),SIG1(NBLOCK)
      real*8 eigVal(NBLOCK,3)
      real*8 damage(NBLOCK),dp(NBLOCK)
!-----------------------------------------------------------------------
!-----Declaration material parameters
!-----------------------------------------------------------------------
      real*8 C2
!-----------------------------------------------------------------------
!     Read material properties
!-----------------------------------------------------------------------
      C2 = props(1)
!-----------------------------------------------------------------------
!     Access stress tensor
!-----------------------------------------------------------------------
      call vgetvrm( 'S', stressdata,jSData,cSData,jStatus)
!-----------------------------------------------------------------------
!     Access equivalent plastic strain
!-----------------------------------------------------------------------
      call vgetvrm('PEEQ',   peeqdata,jPData,cPData,jStatus)
!-----------------------------------------------------------------------
!     Extract data
!-----------------------------------------------------------------------
      do i=1,nblock
         dp(i)      = peeqdata(i)-stateOld(i,1)
         damage(i) = stateOld(i,2)
      enddo
!-----------------------------------------------------------------------
!     Extract data from stressdata
!-----------------------------------------------------------------------
      if(NSHR.gt.1)then
         do i=1,nblock
            s(i,1) = stressdata(i)
            s(i,2) = stressdata(i+nblock)
            s(i,3) = stressdata(i+nblock*2)
            s(i,4) = stressdata(i+nblock*3)
            s(i,5) = stressdata(i+nblock*4)
            s(i,6) = stressdata(i+nblock*5)
         enddo
      else
         do i=1,nblock
            s(i,1) = stressdata(i)
            s(i,2) = stressdata(i+nblock)
            s(i,3) = stressdata(i+nblock*2)
            s(i,4) = stressdata(i+nblock*3)
         enddo
      endif
!-----------------------------------------------------------------------
!     Compute principal stress
!-----------------------------------------------------------------------
      call vsprinc(nblock,s,eigVal,NDIR,NSHR)
```

```
      do i=1,nblock
         SIG1(i) = max(eigVal(i,1),eigVal(i,2),eigVal(i,3))
      enddo
!-----------------------------------------------------------------------
!     Update damage variable
!-----------------------------------------------------------------------
      do i=1,nblock
         if(dp(i).gt.0.0)then
            damage(i) = damage(i)+max(0.0,SIG1(i))*(dp(i)/C2)
         endif
      enddo
!-----------------------------------------------------------------------
!     Update state dependent variables
!-----------------------------------------------------------------------
      do i=1,nblock
         stateNew(i,1) = peeqdata(i)
         stateNew(i,2) = min(1.0,damage(i))
      enddo
!-----------------------------------------------------------------------
!     Check for fracture
!-----------------------------------------------------------------------
      do i=1,nblock
         if(damage(i).ge.1.0)then
            stateNew(i,3) = 0.0
         else
            stateNew(i,3) = 1.0
         endif
      enddo
!-----------------------------------------------------------------------
!     End of subroutine
!-----------------------------------------------------------------------
      return

         end
```

Combined subroutine

```
!$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
!$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
!     Subroutine VUSDFLD
!$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
!$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
      subroutine VUSDFLD(
     + NBLOCK, NSTATEV, NFIELDV, NPROPS, NDIR, NSHR,
     + JELEM, KINTPT, KLAYER, KSECPT,
     + STEPTIME, TOTALTIME, DT, CMNAME,
     + COORDMP, DIRECT, T, CHARLENGTH, PROPS,
     + STATEOLD,STATENEW,FIELD)
      include 'vaba_param.inc'
!-----------------------------------------------------------------------
!-----Declaration ABAQUS variables
!-----------------------------------------------------------------------
      dimension JELEM(NBLOCK),COORDMP(NBLOCK,*),DIRECT(NBLOCK,3,3),
     .          T(NBLOCK,3,3),CHARLENGTH(NBLOCK),PROPS(NPROPS),
     .          STATEOLD(NBLOCK,NSTATEV),STATENEW(NBLOCK,NSTATEV),
     .          FIELD(NBLOCK,NFIELDV)
      character*80 CMNAME
!-----Data from ABAQUS
      dimension stressdata(maxblk*(ndir+nshr))
      integer jSData(maxblk*(ndir+nshr))
```

```fortran
      character*3 cSData(maxblk*(ndir+nshr))
      integer jStatus
      dimension peeqdata(maxblk)
      integer jPData(maxblk)
      character*3 cPData(maxblk)
!-----------------------------------------------------------------------
!-----Declaration internal variables
!-----------------------------------------------------------------------
      integer i
      real*8 s(NBLOCK,NDIR+NSHR),SIG1(NBLOCK)
      real*8 eigVal(NBLOCK,3)
      real*8 damage(NBLOCK),dp(NBLOCK)
      real*8 damage2(nblock)
!-----------------------------------------------------------------------
!-----Declaration material parameters
!-----------------------------------------------------------------------
      real*8 Wc
      real*8 wc2
!-----------------------------------------------------------------------
!     Read material properties
!-----------------------------------------------------------------------
      C2 = props(1)
      C1 = props(2)
!-----------------------------------------------------------------------
!     Access stress tensor
!-----------------------------------------------------------------------
      call vgetvrm( 'S', stressdata,jSData,cSData,jStatus)
!-----------------------------------------------------------------------
!     Access equivalent plastic strain
!-----------------------------------------------------------------------
      call vgetvrm('PEEQ',   peeqdata,jPData,cPData,jStatus)
!-----------------------------------------------------------------------
!     Extract data
!-----------------------------------------------------------------------
      do i=1,nblock
         dp(i)     = peeqdata(i)-stateOld(i,1)
         damage(i) = stateOld(i,2)
         damage2(i)= stateOld(i,3)
      enddo
!-----------------------------------------------------------------------
!     Extract data from stressdata
!-----------------------------------------------------------------------
      if(NSHR.gt.1)then
         do i=1,nblock
            s(i,1) = stressdata(i)
            s(i,2) = stressdata(i+nblock)
            s(i,3) = stressdata(i+nblock*2)
            s(i,4) = stressdata(i+nblock*3)
            s(i,5) = stressdata(i+nblock*4)
            s(i,6) = stressdata(i+nblock*5)
         enddo
      else
         do i=1,nblock
            s(i,1) = stressdata(i)
            s(i,2) = stressdata(i+nblock)
            s(i,3) = stressdata(i+nblock*2)
            s(i,4) = stressdata(i+nblock*3)
         enddo
      endif
```

```fortran
!-----------------------------------------------------------------------
!     Compute principal stress
!-----------------------------------------------------------------------
      call vsprinc(nblock,s,eigVal,NDIR,NSHR)
      do i=1,nblock
         SIG1(i) = max(eigVal(i,1),eigVal(i,2),eigVal(i,3))
      enddo
!-----------------------------------------------------------------------
!     Update damage variable
!-----------------------------------------------------------------------
      do i=1,nblock
         if(dp(i).gt.0.0)then
            damage(i) = damage(i)+max(0.0,SIG1(i))*(dp(i)/C2)
            damage2(i)=(431820000.0*(peeqdata(i))**1.1)/C1
         endif
      enddo
!-----------------------------------------------------------------------
!     Update state dependent variables
!-----------------------------------------------------------------------
      do i=1,nblock
         stateNew(i,1) = peeqdata(i)
         stateNew(i,2) = min(1.0,damage(i))
         stateNew(i,3) = min(1.0,damage2(i))
      enddo
!-----------------------------------------------------------------------
!     Check for fracture
!-----------------------------------------------------------------------
      do i=1,nblock
         if(damage(i).ge.1.0 .or. damage2(i).ge.1.0 )then
            stateNew(i,4) = 0.0
         else
            stateNew(i,4) = 1.0
         endif
      enddo
!-----------------------------------------------------------------------
!     End of subroutine
!-----------------------------------------------------------------------
      return

       end
```

Matlab script

```
% Prompt user for number of jobs
num_names = input('Enter the number of names: ');

% Initialize a cell array to store the names
job_names = cell(1, num_names);

% Loop through and prompt user to enter each name
for i = 1:num_names
    name = input(sprintf('Enter name #%d: ', i), 's');
    job_names{i} = name;
end

% Define the Abaqus command to run the simulation
abaqus_command = 'abaqus job=JobName input=InputFileName
user=SubroutineName';

% Define the name of the subroutine
subroutine_name = 'VUSDFLD_V2.f';

% Create a cell array to store the system commands for each job
job_commands = cell(length(job_names),1);

% Loop through each job and store the system command in the cell array
for i = 1:length(job_names)
    job_name = job_names{i};
    input_file = sprintf('%s.inp',job_name);
    job_commands{i} = strrep(abaqus_command,'JobName',job_name);
    job_commands{i} = strrep(job_commands{i},'InputFileName',input_file);
    job_commands{i} =
strrep(job_commands{i},'SubroutineName',subroutine_name);
end

% Launch the jobs simultaneously using the parfor loop
parfor i = 1:length(job_names)
    system(job_commands{i});
end
```