STRESS RELAXATION IN SCREW FITTINGS IN CAST AZ91D COMPONENTS

Alfred Ramírez Opisso

EXAM WORK 2013
COMPONENT TECHNOLOGY – CASTINGS
This exam work has been carried out at the School of Engineering in Jönköping in the subject area Component Technology - Castings. The work is a part of the International ERASMUS exchange program between Jönköping University and Escola Tècnica Superior d’Enginyeria Industrial de Barcelona, Universitat Politècnia de Catalunya (Spain).

The author takes full responsibility for opinions, conclusions and findings presented.

Examiner: Anders E. W. Jarfors

Supervisor: Nils-Eric Andersson

Scope: 30 credits ECTS

Date: August 2013
Abstract

This work studies the behavior of screw fittings in high temperatures (120°C). The screw fittings are casted in AZ91D magnesium alloy (9% Al, 1% Zn). This part is jointed together with an aluminum part and the fastener is a M5 bolt made of steel. These three parts form the model which is simulated using Finite Element Analysis (FEA) procedures.

Creep behavior and thermal expansion are included in the simulation. The results show the importance of both phenomena for the stress relaxation. Besides, deformation due to creep is responsible of most of the relaxation found in the AZ91D part. The stresses get reduced up to 40 and 50% in some areas of the magnesium alloy part. However, all the parts remain in contact all the time assuring the usefulness of the model despite the stress reduction.
Summary

Stress Relaxation in Screw Fittings in cast AZ91D Components is a master thesis work done in the School of Engineering of the Jönköping University by a student of the Escola Tècnica Superior d’Enginyeria Industrial de Barcelona.

The growing concern about the environment and the impact of manufacture puts light metals like magnesium and its alloys into the focus. This work contributes to this research and analyses how stress relaxation affects magnesium alloys. In particular, this work aims to investigate a specific usage of a specific magnesium alloy.

This master thesis is about the magnesium alloy AZ91D, which contains magnesium together with aluminum and zinc. It has similar qualities as magnesium, some of them improved. However, it shows poor creep resistance at elevated temperatures.

The model studied consists of three parts: a bolt and two parts, one made of aluminum and the other one made of AZ91D. This last part contains a threaded hole, its screw fittings are the ones that are studied.

In order to assess the stress relaxation in the AZ91D part, the whole model is simulated numerically using a finite element analysis (FEA) software. This software helps to find if the expected stress relaxation happens, it also helps to see the amount of stress relaxation and its cause.

This simulation is carried out after a careful examination of all the physical and mechanical properties that can affect the stress distribution and the deformation. Special attention is given to creep, which is implemented only in the aluminum and AZ91D parts. The objective of this work is to do a first analysis of the problem, as simple as possible, without losing accuracy and quality of the results.

The results are presented in the last part of this report, confirming the stress relaxation of the AZ91D part. Stresses are reduced in the whole part, especially in the threads, where stresses are higher. Thus, stress relaxation reduces the stresses up to 40 or 50% in some zones of the material.

The stress relaxation found in this work is originated by deformation due to creep at high temperature of AZ91D. No other causes are investigated in this project. Despite the high amount of stress relaxation in some areas, contact between the parts does not disappear and the joint of the two parts is secure and the model is still useful.

Keywords

Magnesium alloy, AZ91D, Finite Element Analysis, Creep modeling, Creep simulation, Stress Relaxation, Screw Fitting, Threaded hole
Contents

1 Introduction .................................................................................................................. 1
  1.1 Background .............................................................................................................. 1
  1.2 Purpose and Research Questions ............................................................................. 2
  1.3 Delimitations .......................................................................................................... 2
  1.4 Outline .................................................................................................................... 3

2 Theoretical background ................................................................................................. 5
  2.1 Magnesium Properties and Its Alloys ................................................................... 5
      2.1.1 AZ91D magnesium alloy .................................................................................. 6
      2.1.2 High temperature effects in AZ91D ................................................................. 8
  2.2 Screw Thread Mechanics ...................................................................................... 10
      2.2.1 Unscrewing, a major failure mode ................................................................ 11
      2.2.2 Important design parameters in bolted joints ................................................. 12
  2.3 Finite Element Method Introduction .................................................................... 13

3 Method and implementation ......................................................................................... 15
  3.1 Model Description ................................................................................................. 15
  3.2 Sketching the Geometry ....................................................................................... 15
  3.3 Assigning Properties to Parts ................................................................................ 16
      3.3.1 Steel properties for FEM application .............................................................. 17
      3.3.2 Aluminum properties for FEM application ..................................................... 18
      3.3.3 AZ91D properties for FEM application .......................................................... 18
      3.3.4 Defining creep behavior in Abaqus ................................................................. 20
  3.4 Contact Interaction ............................................................................................... 23
  3.5 Loads and Boundary Conditions ............................................................................ 23
      3.5.1 Loads ............................................................................................................... 23
      3.5.2 Boundary conditions ...................................................................................... 25
  3.6 Steps and Simulation Definition ............................................................................ 25
  3.7 Meshing and Element Type .................................................................................... 26
      3.7.1 Choosing element type ................................................................................... 26
      3.7.2 Meshing ......................................................................................................... 30
  3.8 Submit the Job and Getting Results ..................................................................... 31

4 Findings and analysis ..................................................................................................... 33
  4.1 Bolt Torqueing ....................................................................................................... 33
      4.1.1 Bolt ................................................................................................................... 34
      4.1.2 AZ91D part ...................................................................................................... 36
      4.1.3 Aluminum part .............................................................................................. 38
  4.2 Heating up to 120°C .............................................................................................. 39
      4.2.1 Bolt ................................................................................................................... 40
      4.2.2 AZ91D part ...................................................................................................... 41
      4.2.3 Aluminum part .............................................................................................. 43
  4.3 First cycle .............................................................................................................. 43
      4.3.1 First Creep period .......................................................................................... 43
      4.3.2 Decrease of Temperature and Low Temperature Creep .................................. 47
      4.3.3 End of the First Cycle .................................................................................... 48
  4.4 Next cycles ............................................................................................................. 50
      4.4.1 Aluminum-Bolt Contact Surface ................................................................... 52
STRESS RELAXATION IN SCREW FITTINGS IN CAST AZ91D COMPONENTS

Alfred Ramírez Opisso
Introduction

Stress Relaxation in Screw Fittings in cast AZ91D Components is a master thesis work done in the School of Engineering of the Jönköping University by a student of the Escola Tècnica Superior d'Enginyeria Industrial de Barcelona. This work takes place in an international study exchange between these universities.

This work has been carried out in the department of Mechanical Engineering of the University of Jönköping. This project is one of several collaborations that the university does together with the Swedish company Husqvarna. This work is part of the Component Casting Project Funded by the Knowledge Foundation.

Stress Relaxation in Screw Fittings in cast AZ91D Components investigates AZ91D's behavior in the threads of a threaded joint in room and high temperatures. More precisely, this work focuses on a joint in a little engine produced by the cited company.

This project consists, first of all, on an investigation dealing with the characteristics and properties of the material, the geometry and the different parts that compose the model. The second part of this work is a numerical simulation with finite elements of the virtual model of the cited threaded joint in different conditions of temperature and stresses.

Background

Since late 90’s, magnesium and magnesium alloys usage has become a growing industry full of opportunities. Magnesium and its alloys are considered light alloys; in fact, magnesium is the lightest structural metal with a density of 1.74 g/cm³.

The most common uses for magnesium are structural purposes; however there are a lot of more applications.

In addition to its low density, magnesium presents other good attributes like its high strength-to-weight ratio, the good electric and thermal conductivity, its recyclability and the vibration damping. All these qualities make magnesium a big opportunity for industry and the society. The growing concern about the environment and the impact of manufacture puts light metals into the focus. Magnesium is considered a good substitutive for many applications, especially structural.

This expansion on the magnesium applications would not be possible if it was not accompanied by a growing interest in the research of this material and its alloys. Research on magnesium and magnesium alloys is carried out by academic institutions but also takes place in industries, which are very interested in these material’s benefits.

Despite the progress done so far, there are still challenges that limit the use of magnesium. This work aims to investigate a specific usage of a specific magnesium alloy. This work attempts to investigate the properties of this specific alloy, focusing on creep phenomenon at elevated temperatures. This work seeks to contribute –knowing it is a master thesis– to this magnesium research.
Stress Relaxation in Screw Fittings in cast AZ91D Components is about a magnesium alloy called AZ91D. This alloy contains magnesium, aluminum and zinc. It has several of the qualities of magnesium, some of them improved. It is specially indicated for pressure die castings. However, one of the worst qualities of this material is the creep resistance at high temperatures.

The model which is going to be studied consists of three parts: a bolt and two parts, one made of aluminum and the other one made of AZ91D. This last one contains a threaded hole. These screw fittings are the ones that are studied and their stress relaxation is being studied in this work. The model works in a temperature range that varies from 20 to 120ºC in cycles of 800h.

Given the AZ91D bad behavior at elevated temperatures it is important to study the consequences to bring the whole model to 120ºC; this is the main reason to run this project.

1.2 Purpose and research questions

The present work will deal with, as said before, the stress relaxation that happens in the screw fittings of a AZ91D part. This part is under loads due to a bolt screwed and high temperature.

In order to assess the stress relaxation in the AZ91D part, the whole model will be simulated numerically using a finite element analysis software. In particular, the Abaqus 6.11 software will be used; with the student license of the Jönköping University.

The research questions that this work will try to answer are:

- Is there any stress relaxation at the screw fittings of the AZ91D part? Is there any stress relaxation in anywhere else of the model?
- What is the cause of the stress relaxation?
- If there is any stress relaxation, how much are the stresses reduced and what are the zones where this relaxation occurs?
- Does the stress relaxation affect to the usefulness of the model? Does it work after the stress reduction?

1.3 Delimitations

Given the amplitude that the problem could embrace, some decisions regarding limitations are need to be done. This is a master thesis work and has limited time available to meet the deadlines.

Thus, the present work will only consider creep phenomenon as a possible origin of stress relaxation. Besides, no other consequences will be considered but elastic, plastic and creep deformation.
No consideration of vibrations is made in this project, nor any other influences from the ambient and surroundings of the model. The simulation will only consider two dimensions, thus, no stresses nor deformations nor motion in the third dimension are considered.

The objective of this work is to do a first analysis of the problem, taking simplicity as a reference. However, this simplicity is not against accuracy and quality of the results. According to this simplicity premise, the creep law, the geometry modeling and the simulation are the most simple possible without losing accuracy.

1.4 Outline

The present work follows the template and guidance given by the Jönköping University.

In a first place, after the introduction, there is the theoretical background, where basic theory is exposed. This theory refers to three main points: magnesium and its alloys, with special attention to AZ91D and its properties; screw mechanics and their geometry, and a little introduction to finite element method.

The third chapter of this thesis explains how the model has been built using the Abaqus software. The important area this chapter deals with is the decision process taken while designing the model. This section comprises the design of the geometry of the model, the mechanical properties of each part and material, the contact interaction between the different parts and the finite element properties applied to get the results.

The fourth chapter presents the results from the simulation and compares the results between them. Finally, the conclusions close this project work answering the research questions announced before.
2 Theoretical background

2.1 Magnesium properties and its alloys

Magnesium is considered a light metal together with Aluminum, Titanium and Beryllium, because of their relative low densities ranging from 1.7 g·cm\(^{-3}\) on magnesium to 4.5 g·cm\(^{-3}\) on titanium. The designation ‘light metals’ is given because they are frequently used to reduce the weight of several structural components. This can be done because of magnesium’s excellent specific strength (the strength-to-density ratio – \(\sigma/\rho\)) and stiffness, machinability, dimensional stability and recycling capacity (Yang, et al., 2008). Magnesium and its alloys are mostly applied in different spheres of industry, including the aircraft and motor vehicle, where the mass reduction is an important issue, but they are also used in metallurgical, chemical, communication and electrical industries.

Magnesium is the sixth most abundant element on earth’s surface and the second of the light metals, following aluminum, which is the third on crustal abundance. Most of the magnesium produced worldwide (nearly three-quarters of world production) is used for alloying purposes, mainly with aluminum, steel and iron. The 16% of the remainder is used for magnesium alloys, mainly as die castings for the aerospace and general transport industries. Therefore, magnesium can be alloyed with several other metals such as aluminum, zinc, zirconium, thorium, rare earth elements, silver and lithium. The use of wrought magnesium is much lower due to the hexagonal crystal structure of this metal (Polmear, 1995).

Magnesium has an hexagonal crystal structure. Its atomic radius is 0.16 nm length. This length brings to magnesium favorable size factors with a diverse range of solute elements and facilitates alloying. It also presents good electric and thermal conductivities. In Table 2.1 the main physical and mechanical properties of magnesium can be found.

<table>
<thead>
<tr>
<th>Property</th>
<th>Property</th>
</tr>
</thead>
<tbody>
<tr>
<td>Atomic weight</td>
<td>Coefficient of expansion 10(^{-6})K(^{-1}) at °C</td>
</tr>
<tr>
<td>Atomic number</td>
<td>Young’s modulus GPa</td>
</tr>
<tr>
<td>Density (g·cm(^{-3}))</td>
<td>Rigidity modulus GPa</td>
</tr>
<tr>
<td>Melting point °C</td>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>Thermal conductivity (W·m(^{-1})·K(^{-1}))</td>
<td>Specific heat (J·kg(^{-1})·K(^{-1})) at °C</td>
</tr>
</tbody>
</table>

2.1.1 AZ91D magnesium alloy

Pure magnesium is not commonly used in engineering applications; it is usually used in form of magnesium alloys. There is still a limited number of magnesium alloys and often their properties do not match the properties sought by the designers. This is the main reason for these alloys to be on second place, behind aluminum alloys. However, magnesium alloys applications and research have considerably grown in the recent years.

Magnesium alloys have different naming depending on the standard followed. This work will follow the ASTM 275 specification where magnesium alloys are named by a short code that gives information about the approximate chemical composition of the alloy. In AZ91’s case, the first two letters denote that this magnesium alloy contains aluminum (A) and zinc (Z). The two numbers show the approximate chemical composition by weight, which is about 9% of aluminum and 1% of zinc.

This code is often followed by a letter that gives information on the usage of the alloy. Thus, the letter D at the end of AZ91D designation means this alloy is for pressure die casting. If a letter E were there instead of D would mean that the alloy is optimized for sand casting. These two alloys -D, or -E are the substitutes for the previously used -A, -B, or -C, which are now in lack of use because their impurities content was an order of magnitude higher than the current AZ91D/E.

Table 2.2 shows the chemical composition of AZ91D.

<table>
<thead>
<tr>
<th></th>
<th>Al</th>
<th>Zn</th>
<th>Mn</th>
<th>Cu</th>
<th>Si</th>
<th>Fe</th>
<th>Ni</th>
<th>Mg</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>8.5-9.5</td>
<td>0.45-0.9</td>
<td>0.15-0.40</td>
<td>0.015</td>
<td>0.020</td>
<td>0.005</td>
<td>0.0010</td>
<td>remainder</td>
</tr>
</tbody>
</table>


Magnesium alloys are likewise light metals (also called light alloys) and their density keeps being in the lowest range. These alloys, especially AZ91D, are characterized by a relatively high specific strength, good castability (in particular thin-wall capability), good weldability and good corrosion resistance. They also have a relative low cost of production. Some properties are shown in Table 2.3.

Furthermore, AZ91D is also characterized by a low Young’s modulus, relatively low hardness and limited creep resistance at elevated temperatures. Also corrosion resistance can be critical in some cases. They present a considerable decrease of mechanical properties with temperature increase (Kielbus, et al., 2006).

The AZ91 alloy is the most widely used magnesium alloy (Polmear, 1995) and has a good combination of high strength at room temperature, good castability and excellent corrosion resistance (Braszczynska-malik, 2011). This alloy contents aluminum as the principal alloying element, followed by zinc.
Aluminum is the most common alloying element in magnesium because of its low price, availability, low density and the advantageous effects on corrosion and strength properties. Additions of aluminum to magnesium alloys enhance strength and hardness at room temperature and improve the fluidity of the alloy. Increasing the percentage of aluminum increases ductility, however, deteriorates elevated temperature strength.

Small amounts of zinc in combination with aluminum give some increase in tensile properties. It also improves fluidity of the alloy. However, the amount of zinc is limited: increasing zinc percentage beyond 2% can cause hot cracking. Hot cracking is a solidification phenomenon where cracks appear due to the molten metal shrinkage because of the solidification.

Some alloys also include little amount of Manganese. This element does not affect the alloy’s mechanical properties but produces beneficial results in the control of corrosion by forming an Fe-Mn-Al intermetallic (Regev, et al., 1998).

Presence of aluminum usually results in the formation of eutectic Mg\textsubscript{17}Al\textsubscript{12}, deteriorating mechanical properties, especially creep resistance. Either rare elements or alkaline earth elements can be added to avoid or minimize Mg\textsubscript{17}Al\textsubscript{12} formation. This addition will improve creep resistance but increase the cost of production (Bronfin, et al., 2006).

The eutectic phase Mg\textsubscript{17}Al\textsubscript{12} is usually named β-phase (or less commonly γ-phase) and appears in magnesium alloys containing more than 2% aluminum. A network of β-phase appears around grain boundaries as the aluminum content is increased and ductility decreases rapidly above 8% aluminum. At Figure 2.1 there is the fragment of the Mg-Al diagram corresponding to the AZ91D ranges. This casting alloy is characterized by a solid solution structure α with eutectic α+β and β-phase at grain boundaries (Kielbus, et al., 2006).

### Table 2.3 AZ91D PHYSICAL PROPERTIES

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (g·cm\textsuperscript{-1})</td>
<td>1.83</td>
</tr>
<tr>
<td>Melting point °C</td>
<td>470</td>
</tr>
<tr>
<td>Thermal conductivity (W·m\textsuperscript{-1}·K\textsuperscript{-1})</td>
<td>84</td>
</tr>
<tr>
<td>Specific heat (J·kg\textsuperscript{-1}·K\textsuperscript{-1}) at °C</td>
<td>1000</td>
</tr>
<tr>
<td>Coefficient of expansion 10°K\textsuperscript{-1} at °C</td>
<td>27.0</td>
</tr>
</tbody>
</table>

Find more AZ91D’s properties in section 3.3.3

Theoretical Background

2.1.2 High temperature effects in AZ91D

Although AZ91D’s good attributes, temperature increase has adverse effects on its mechanical properties. Increasing the working temperature up to 100-150°C brings the alloy to weaken its properties.

At high temperature the stress-strain curve is lower and flatter than at low temperature. This is explained by a balance between hardening and softening. When AZ91D deforms plastically, dislocations coming from this plastic deformation tend to harden the material. At elevated temperatures, this hardening phenomenon is compensated by recovery, which is enabled by high temperatures, which mainly is a rearrangement of these dislocations (Lukáč, et al., 2011).

2.1.2.1 Creep phenomena

Most of metals follow Hooke’s law when they are stressed up to some yield stress; beyond this point plastic deformation starts. If the metal is strained to a certain point outside the elastic region, releasing the load brings the total strain to immediately decrease in parallel with the slope of the elastic part. A remaining strain appears and is commonly considered the permanent plastic deformation.

However, the remaining strain is not all permanent plastic strain. Depending on the metal and the temperature, a small amount of plastic strain will disappear with time and decrease to a certain point.
This phenomenon can have many causes and all of them are included in which is called the anelastic behavior of metals. Creep is one important manifestation of anelastic behavior. Anelastic effects are a rare phenomenon at room temperature but under certain circumstances they can be critical and can let the part useless.

Creep is defined by R.A. Higgins (Higgins, 1993) as a phenomenon of continuous gradual extension under a steady force. It occurs as a result of long term exposure to relative high levels of stress that are below the yield strength of the material.

Creep phenomenon is usually described with the creep curve (Figure 2.3). Engineering creep curve is created keeping load and temperature constant during the experiment so the strain (ε) is determined as a function of time.

The curve starts at time t₀, when load is applied and an immediate elongation occurs. After this initial rapid elongation, the creep rate decreases with time, then reaches a steady state and finally the creep rate increases rapidly with time until fracture occurs. This curve indicates that the plastic strain associated with creep occurs in three stages:

1. **Primary or transient creep**: is a period in which the creep resistance of the material increases because of its own deformation. For low temperatures and stresses this is the predominant creep process.

2. **Secondary or steady-state creep**: is a period of nearly constant creep rate which results from a balance between the competing processes of strain hardening and recovery. This step is usually referred to as steady-state creep. The average value of the creep rate during secondary creep is called the minimum creep rate. The minimum creep rate is the most important design parameter derived from the creep curve.

3. **Tertiary creep**: mainly occurs in constant-load creep tests at high stresses at high temperatures. Occurs when there is an effective reduction in cross-sectional area either because of necking or internal void formation. It is often associated with metallurgical changes such as coarsening of precipitate particles, recrystallization, or diffusional changes in the phases that are present.

### 2.1.2.2 **Creep in AZ91D**

In the AZ91D alloy, the formation of β precipitates occurs both within grains and at the vicinity of grain boundaries. The β phase present in grain boundaries is responsible for material strengthening at room temperature due to precipitation hardening. This is an important strengthening mechanism in magnesium alloys that occurs when the solid solubility decreases with decreasing temperature.
Nevertheless, the melting point for Mg$_{17}$Al$_{12}$ is 437 °C, much lower than magnesium’s 649 °C. Hence, with increasing temperature the β intermetallic phase softens in such a manner that weakens the grain boundaries so that they lose pinning capacity so grain boundary sliding occurs (Meshinchi Asl, 2011) (Ai, et al., 2011). This is the main factor for poor creep resistance of these alloys. Softening and coarsening of the β phase weaken the grain boundaries allowing them to slide, bringing deformation to the material.

The evaluation of creep resistance of magnesium alloys can be done by using the minimum creep rate $\dot{\varepsilon}$. The smaller $\dot{\varepsilon}$, the better the creep resistance. Minimum creep can be determined with following power law, relating it to the applied stress:

$$\dot{\varepsilon} = A\sigma^n \exp\left(-\frac{Q}{RT}\right)$$  \hspace{1cm} (1)

Where A is a material constant, $\sigma$ the average stress in the alloy and $n$ is the stress coefficient, $T$ the temperature, $R$ the gas constant and $Q$ surface activation energy.

Luo and Pekguleryuz (Luo, et al., 1994) state that only primary and secondary creep curves are of interest for engineering and they can be described by the following equations:

Primary creep strain $= at^b$

Secondary creep strain $= c + dt$

The creep strain coefficients $a$, $b$, $c$, and $d$ are dependent on temperature and stress. Some of them are given on Table 2.4.

| Table 2.4 PRIMARY AND SECONDARY CREEP COEFFICIENTS FOR AZ91 |
|-----------------|--------|----|----|----|
| Temperature (ºC) | Stress (MPa) | $a$ | $b$ | $c$ | $d$ |
| 100 | 100 | 0.078 | 0.45 | 0.280 | 0.0028 |
| 150 | 50 | 0.078 | 0.58 | 0.280 | 0.0092 |
| 200 | 30 | 0.064 | 0.69 | 0.310 | 0.0096 |


2.2 Screw thread mechanics

A screw thread mechanism is generally composed by a screw and a nut. The first one is a component with an external thread (e.g. setscrew, leadscrew, bolt and stud), and the second one refers to any component whose internal thread engages the screw, generally a nut or a cast threaded (or tapped) hole (Wright, 2005).
The objectives of a threaded fastener are to join other parts together and to keep these parts joined while different forces tend to separate them (Fenollosa i Coral, 1998).

The tightening process is composed by an elastic and a plastic deformation of the parts involved originated by the torque applied. The elastic deformation consists on a flattening of the component surfaces in contact, a bolt stretching and a compression of the parts to be joined. These elastic deformations, although small, are the origin of forces and reactions named as clamp load or preload.

Over-torquing the joint leads to plastic deformation of the parts involved, deforming the nut and bolt surfaces in contact with the joined parts, deforming the fastener and also damaging its thread. However, under-torquing can loose the joint and make it more susceptible to fatigue.

After the tightening process, some hours or maybe days later, joined parts and screws suffer from a second plastic deformation that can lead to the same effects exposed before. These second plastic deformation of the parts and the nut and bolt is called settlement and appears as a result of a creep deformation. These second deformation could relax the joint stresses at a point to be critical.

2.2.1 Unscrewing, a major failure mode

There are several failure modes of a screw thread joint: breaking of the bolt, destruction of the joined parts, joint opening (loss of function), excessive elastic deformation and unscrewing of the nut or bolt.

Breaking of the bolt, destruction of the joined parts and excessive elastic deformation have their origin in excessive external forces applied to the fastened parts. Joint opening can be originated by either under-torquing or unscrewing.

The screw thread joint can be unscrewed because of vibrations and shocks, excessive settlement or different thermal dilatation between the bolt and the fastened parts.

The external force applied to the joined parts is transmitted to the fastener mechanism by the threads. The threads’ helical inclination originates a torque which tends to unscrew the nut or bolt. Friction is a very important parameter that dominates the motion between nut and bolt and is the main source against unscrewing. The more friction, the more forces against unscrewing.

Joined parts provide its elastic reaction force to the screw fastener. If this force is kept, there is contact force between threads and the nut does not unscrew. Too much settlement can make the elastic deformation disappear and so the elastic reaction force leading to unscrewing of the nut.
2.2.2 Important design parameters in bolted joints

2.2.2.1 Screw thread geometry

Bolt’s geometry is of great importance when designing a part and also when simulating it. Most of the world-wide used bolts follow the international standard ISO 68-1 which defines the screw thread geometry. Figure 2.4 shows the thread geometry.

![Figure 2.4 Basic profile of all ISO metric screw threads](image)

A metric ISO bolt is designated by the letter M followed by the value of the nominal diameter \((D_{maj})\) in Figure 2.4) expressed in millimeters. ISO 261 and ISO 262 list the preferred (also more commonly used) combinations of nominal diameter and pitch \((P)\) for ISO metric screw threads. Bolts used in this work are metric M5. Knowing that, geometry becomes fully defined in Table 2.5.

<table>
<thead>
<tr>
<th>Design parameter</th>
<th>Symbol in Figure 2.4</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nominal diameter (major diameter)</td>
<td>(D_{maj})</td>
<td>5 mm</td>
</tr>
<tr>
<td>Pitch</td>
<td>(P)</td>
<td>0.8 mm</td>
</tr>
<tr>
<td>Height of the V-Shape</td>
<td>(H)</td>
<td>(0.8660\times P)</td>
</tr>
<tr>
<td>Minor diameter</td>
<td>(D_{min})</td>
<td>(D_{maj} - 1.0825\times P)</td>
</tr>
<tr>
<td>Pitch diameter</td>
<td>(D_{p})</td>
<td>(D_{maj} - 0.6495\times P)</td>
</tr>
</tbody>
</table>
2.2.2.2 **Preload**

Preloading a screwed joint means to set a torque value in order to tighten the joint. This torque determines the internal stresses in the joint. Some of the stress will disappear after some time due to settlement, defined early. This process is of great importance and either overtorquing or undertorquing can dramatically damage the parts.

Generally the maximum preload is limited by the mechanical characteristics of the bolt. This preload has to ensure a correct joint without damaging the parts and avoiding plastic deformation of the threads.

2.3 **Finite Element Method Introduction**

The Finite Element Method (FEM) is a numerical technique for finding approximate solutions to real problems. It brings the chance to avoid the expensive job of making several prototypes. Although the results can be greatly accurate, it is still an approximation of the actual model based on hypothesis and simplifications.

FEM is based on subdividing a whole domain into simpler parts and connecting these subdomains through mathematical relations. These subdomains are named finite elements. Each element is connected to others by a number of points situated on the outline of the element. These points are called nodes and are the basis of calculation. Displacements or temperatures in each node are calculated with FEM.

How this method works will not be discussed in this thesis. In general terms must be said that the basic principle of the FEM is to seek equilibrium in each node when external interactions (forces, pressures, heat...) are applied to one or more nodes.

As one can easily think, this is not a matter of just a few nodes, it is very easy to obtain thousands of nodes with a not-so-big model. This magnitude makes impossible to solve such problems manually and assistance of a computer is needed.

There are a great number of software solutions able to process the finite element method. In this work the FEM solver Abaqus has been used. Abaqus is a suite of different programs that provides a simple interface for creating, submitting and evaluating results from simulations.

Abaqus contains three main subprograms. Understanding what these subprograms are for may help the user to better perform the simulation.

- **Preprocessor:** This is where the problem is defined. The user draws the geometry, defines the material properties, boundary conditions, forces, initial conditions, etc. Here is also where the whole model is divided in elements, creating what is called the mesh. The user is able to easily do all these steps thanks to the interface that helps this work in a visual-intuitive way. As a result, a `.cae` file is created.
- **Processor**: Here is where the numerical development of the problem takes place. There is no visualization available for this subprogram. It is only about resolving the great amount of equations generated. It is the core and the main part of a FEM solver.

- **Postprocessor**: This subprogram helps to visualize the results obtained in the processor. Results are shown on the model, in graphics, etc. Deformed shape of the structure simulated or temperature distribution after heat transfer are typical result displays on the postprocessor. Temporal evolution of different variables is also possible.
3 Method and implementation

3.1 Model description

The model which is going to be simulated consists of a metric bolt M5 with a preload of 15 Nm. This bolt fastens two parts: the upper one is made of aluminum and is 8 mm thick; the next 18 mm of the bolt go through magnesium’s alloy AZ91D.

The whole model works on a range of temperatures from 20ºC (room temperature) up to 120ºC (high temperature). The model spends 400h at high temperature during each cycle.

There is an inner pressure of 35 bar (3.5 MPa) which is distributed on 4 equal screw fittings and results on a force which tends to separate the two parts.

The modeling task in Abaqus is divided in several modules. Each module contains only the tools that are relevant to the specific modeling task is carried out. Thus, Abaqus is divided in 11 modules and they are arranged in a logical sequence one may follow to create a model. The following sections describe the process to obtain the model, assign properties to it and running the simulation. These explanations will follow the module’s order, just taking in considerations only the ones which are relevant to this project.

3.2 Sketching the geometry

The part module is where individual parts are created by first sketching the geometry. Before drawing the model some decisions need to be made. The model is formed of three different parts: the upper part made of aluminum, the lower AZ91D part and the bolt which joints both parts together and is made of steel.

Although the whole model is important, given the object of this thesis results will be sought mainly at the threads of the lower part. In order to economize the use of computing resources but also to achieve accurate results, an axisymmetric simulation will be run. This means that only a 2D sketch is needed and Abaqus considers this sketch to be symmetric all around the Z axis, which is the longitudinal bolt axis.
This hypothesis doesn’t take into account the threads’ helical inclination, which is not axisymmetric along the bolt’s length. However, Professor Niels Leergaard (Leergaard Pedersen, 2013) confirms in his work *Optimization of bolt thread stress concentrations* the viability of such a procedure without losing accuracy in the results.

Values form section 3.1 and Table 2.5 have been used to draw the model, which is presented in Figure 3.2.

When sketching the parts, some decisions were taken that have to be mentioned. The aluminum part does not have threads; it only contains a clearance hole. This hole needs to be wider than the nominal bolt’s diameter. ISO standard specifies three clearance hole sizes: close fit, medium fit and free fit. The second one was the chosen one, with a diameter of 5.5 mm.

One more decision to be mentioned is about the width of the aluminum and magnesium alloy parts. In a first attempt 15 mm width are considered in both parts. After the first simulation will be considered to increase or reduce this value according with the tensions at both parts. However, one must remember that this is an axisymmetric analysis and that the real model does not have this symmetry at a global scale. This is just a hypothesis in a local scale around the bolt, so it would not be appropriate to extend much more the parts.

### 3.3 Assigning properties to parts

The analysis consists mainly on studying the stress evolution in AZ91D’s screw fittings. These stresses are caused by two factors: an inner force which tends to separate the aluminum and AZ91D parts and bolt pretension. Later on, these stresses could be modified due to thermal expansion when increasing temperature. Time and temperature also affect these stresses. Creep may be an important effect in the AZ91D’s threads, bringing stress relaxation at a point that could be critical.

With these assumptions, each part of the model has been assigned the following properties: thermal expansion coefficient, the stress-strain curve parameters and the creep law the different materials follow. All these characteristics have to be temperature dependent.

Special attention has been given to AZ91D’s properties, given that this material is the object of the study. Aluminum and steel properties are based on the general properties of these materials but still suitable for the simulation. In the following lines these properties are described.
3.3.1 Steel properties for FEM application

Metric bolts don’t have a regularized material which they must be made of. Instead of the material, the resistance (strength) of the bolt is specified by the ISO standard. The same resistance can be achieved using different materials and alloys. This should be a decision of the manufacturer and the designer (Fenollosa i Coral, 1998).

The most common classes for metric bolts are 5.6 for poor requirements, 8.8 for standard usage, and 10.9 and 12.9 for high resistance applications. Class 8.8 is considered in this work. This means the ultimate strength of the bolt is 800 MPa and the Yield Strength at 0.2% elongation is 640 MPa.

According to ASM Handbook, Volume 1 (ASM International, 1990) steel’s thermal expansion coefficient does not vary significantly between 20°C and 120°C, so it is considered a constant value in this simulation. However, there is an approximate 5% decrease in Yield strength. Young’s Modulus and Poisson’s ratio are considered constant at the given temperature interval. Table 3.1 and Figure 3.3 show the steel properties used in the simulation.

Table 3.1 STEEL PROPERTIES

<table>
<thead>
<tr>
<th>Steel properties for Abaqus</th>
<th>20°C</th>
<th>120°C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal Expansion Coefficient</td>
<td>$13 \times 10^{-6} \text{ K}^{-1}$</td>
<td>$13 \times 10^{-6} \text{ K}^{-1}$</td>
</tr>
<tr>
<td>Young’s Modulus</td>
<td>210 GPa</td>
<td>210 GPa</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.28</td>
<td>0.28</td>
</tr>
<tr>
<td>Yield Strength</td>
<td>640 MPa</td>
<td>608 MPa</td>
</tr>
<tr>
<td>Ultimate Tensile Strength</td>
<td>800 MPa</td>
<td>760 MPa</td>
</tr>
</tbody>
</table>


Figure 3.3 Stress-Strain curves of steel at 20 and 120°C used in simulation

### 3.3.2 Aluminum properties for FEM application

For the simulation of the aluminum part, a 6000 series aluminum alloy is considered. This aluminum alloy family is largely used in very different purposes; they contain magnesium and silicon and generally have good mechanical properties. According to ASM Handbook, Volume 2 (ASM International, 1990) aluminum’s thermal expansion coefficient increases from $24 \cdot 10^{-6}$ at room temperature up to $26.3 \cdot 10^{-6} \text{ K}^{-1}$ at 120°C. There is also an approximately 6.5% decrease in Yield strength at high temperature. On the other hand, Young’s Modulus and Poisson’s ratio are considered constant at the given temperature interval. Table 3.2 and Figure 3.4 show the steel properties used in the simulation.

![Aluminum stress-strain curves at 20 and 120°C](image)

**Figure 3.4** Aluminum stress-strain curves at 20 and 120°C.  

<table>
<thead>
<tr>
<th>Property</th>
<th>20°C</th>
<th>120°C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal Expansion Coefficient</td>
<td>$24 \cdot 10^{-6}$</td>
<td>$26.3 \cdot 10^{-6}$</td>
</tr>
<tr>
<td>Young’s Modulus</td>
<td>68.9 GPa</td>
<td>68.9 GPa</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.33</td>
<td>0.33</td>
</tr>
<tr>
<td>Yield Strength</td>
<td>276 MPa</td>
<td>258 MPa</td>
</tr>
<tr>
<td>Ultimate Tensile Strength</td>
<td>310 MPa</td>
<td>290 MPa</td>
</tr>
</tbody>
</table>

**Sources:** SolidWorks® Material Library, ASM Material Data Sheet (MatWeb), and ASM Handbook, Volume 2 (ASM International, 1990).

### 3.3.3 AZ91D properties for FEM application

To accurately do the screw fitting finite element simulation, AZ91D properties are needed as well as their dependency on temperature. In particular, temperature ranges between room temperature (20°C) and 120°C.

What makes it difficult to determine AZ91D’s properties is that the values change depending on specific alloy composition, microstructure, heat treatments,
impurities and defects of the sample, etc. In order to pick an average value, an extensive literature survey has been done looking for properties at different temperature ranges.

There are some properties that are considered invariant at the temperature ranges the simulation will run. In other words, their value remains (nearly) constant from room temperature up to 120ºC. In Table 3.3 these properties are listed and an average value that will be used in the finite element analysis.

Table 3.3  Not-temperature-dependent properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Average value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (g.cm⁻³)</td>
<td>1.83[1]</td>
</tr>
<tr>
<td></td>
<td>1.81[2]</td>
</tr>
<tr>
<td></td>
<td>1.81[2]</td>
</tr>
<tr>
<td></td>
<td>1.8</td>
</tr>
<tr>
<td>Thermal expansion coefficient (10⁻⁶ K⁻¹)</td>
<td>27[1]</td>
</tr>
<tr>
<td></td>
<td>25[2]</td>
</tr>
<tr>
<td></td>
<td>25.2 – 26.5[3]</td>
</tr>
<tr>
<td></td>
<td>26</td>
</tr>
<tr>
<td>Young Modulus* (GPa)</td>
<td>45[3]</td>
</tr>
<tr>
<td></td>
<td>45[3]</td>
</tr>
<tr>
<td></td>
<td>45</td>
</tr>
<tr>
<td>Poisson’s Ratios</td>
<td>0.35[3]</td>
</tr>
</tbody>
</table>


* According to Jürgen Göken’s article (Göken, et al., 2004) AZ91 Young’s modulus is considered to remain constant in the simulation temperature range. In fact, it is barely constant until temperatures beyond 350ºC, when it starts to decrease.

Mechanical properties such as Yield Strength and Ultimate Tensile Strength are quite difficult to collect and compare between different published data due to differences in sample, test conditions and type of test. Mihriban. O. Pekguleryuz (Pekguleryuz, 2006) collected some magnesium alloys properties, and these are the ones considered for this project work.

In the cited paper, Yield Strength, Ultimate Tensile Strength and elongation are given at 20, 150 and 175ºC. The attempt is to interpolate these values to obtain Stress-Strain curves at different temperatures between 20 and 150ºC.

To make it as simple as possible, lineal behavior is expected of the three properties given vs. temperature. And a regression line is plotted (Figure 3.5). Considering the correction of this hypothesis, these lines’ equations bring the
and implementation

possibility to draw the stress-strain curve at any temperature between 20 and 150°C.

AZ91D’s Yield strength at 20°C is 139 MPa and 108 MPa at 120°C. Ultimate tensile strength is 204 and 169 MPa at 20 and 120°C respectively.

3.3.4 Defining creep behavior in Abaqus

Creep models are added into the materials definition like other mechanical and physical properties. Due to the temperature ranges of the simulation (20 to 120°C) the steel of the bolt does not present relevant creep phenomenon so it is not considered. Only Aluminum and AZ91D parts in this simulation can be influenced by temperature and stress and present creep deformation. That’s why their definitions in Abaqus need to contain a creep model. In addition, creep at room temperature is only considered in AZ91D.

3.3.4.1 Power Law

Abaqus provides some predefined creep laws to help modeling the second stage of creep. The Power Law has been used and looks like follows:

\[ \dot{\varepsilon}^{cr} = A \sigma^n t^m \]

Where \( \dot{\varepsilon}^{cr} \) is the creep strain rate, \( \sigma \) are the stresses (von Mises) and \( t \) is time. \( A, n \) and \( m \) are parameters depending on the material’s properties.

This power law considers the strain rate, which is the change of the strain with time; in other words, this creep model considers the derivative with respect to time of the elongation (strain) due to creep. However, information regarding creep models in the literature is available mainly on creep strain (not strain rate) vs. time.
plots; in consequence, the creep power law has needed to be integrated in order to obtain the $A$, $n$ and $m$ parameters.

These parameters are the ones Abaqus asks for to fully define the creep model. Once they have been introduced, the solver software automatically calculates the creep strain in each node, taking in consideration the stress of the nodes and its own time scale. In the next figure, the parameters values are listed.

<table>
<thead>
<tr>
<th>Table 3.4 Material’s parameters for Creep Power Law</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>$A$</td>
</tr>
<tr>
<td>$n$</td>
</tr>
<tr>
<td>$m$</td>
</tr>
</tbody>
</table>

The $A$ parameter multiplies the creep law in order to decrease the value. Such a small exponent ($10^{-38}$) may need an explanation: Stress values are around 100 MPa, or, what is the same, $10^8$ Pa. The stress exponent $n$ raises that value to the power of four: $10^{32}$. The 6 or 7 orders of magnitude left can be explained given the length unit Abaqus uses, which is the meter; due to creep deformations are around $10^{-3}$ mm, the value of $10^{-38}$ may seem correct.

The value of the stress exponent can vary a lot, depending on the characteristics of the specific sample. Several authors relate this exponent with the material deformation mechanism (dislocations). In particular, stress exponent ranges between 4 and 6; however, it can change depending on the simulation properties. For this work a stress exponent of 4 has been chosen, which is in the normal range and gives a curvature to the plot that seems correct.

There is no big information regarding the time exponent. In this case, the -0.36 value has been chosen because the curvature of the plots with this value is close to the curvature of the plots in the literature.

The following figures plot the creep strain vs. time of AZ91D and Aluminum. Figure 3.7 shows the creep strain vs. time of magnesium at 120ºC and Figure 3.8 at 120ºC. Figure 3.9 plots the creep strain vs. time of aluminum at 120ºC (no creep at room temperature).

Method and implementation

Figure 3.7 AZ91D Creep strain vs. time at 120°C

Figure 3.8 AZ91D Creep strain vs. time at 20°C

Figure 3.9 Aluminum Creep strain vs. time at 120°C
3.4 Contact interaction

Although model’s geometry is already created in previous steps and some parts seem to be in contact with others, Abaqus only considers there is contact interaction between parts if it is explicitly specified.

The model presents three different contact surfaces, as shown in red in Figure 3.10: contact between bolt’s threads and magnesium threaded hole (1), contact between aluminum and magnesium parts (2), and contact between aluminum part and the head of the bolt (3).

These three contact zones are defined similarly: two parts are in contact by a surface. This condition allows motion between the two bodies but assuming that there will be relatively little sliding of one surface along the other.

The only difference is the friction coefficient, which is dependent on the material couple in contact.

Professor Fenollosa (Fenollosa i Coral, 1998) proposes some friction coefficients for steel bolts and nuts with different surface treatments and different types of lubrication. Considering no specific surface treatment and no lubrication, the friction coefficient ranges from 0.14 to 0.35.

However, the threaded hole in this problem (which acts as a nut) is made of magnesium, so the coefficient friction may be different. 0.25 will be used.

According to ASM Handbook, Volume 18 (ASM International, 1992), friction coefficient between aluminum and steel is 0.35. Friction coefficient between aluminum and AZ91D is 0.45.

3.5 Loads and Boundary Conditions

3.5.1 Loads

3.5.1.1 Bolt Load

Abaqus provides an easy way to model tightening forces in bolts or fasteners. Using the “bolt load” editor, one can define the tightening force in terms of a concentrated force applied across the bolt cross-section. With the purpose of using this feature, it is needed to convert the torque into a concentrated force.

The torque applied on the bolt is 8 Nm. This torque is transmitted to the threads and is responsible for keeping the different joined parts in contact. The process of applying this torque is called the tightening process and results in the clamp force, also called bolt load in Abaqus.
Josep Fenollosa (Fenollosa i Coral, 1998) explains that the tightening torque ($T$) is composed of two different torques: the Thread Torque ($T_T$) and the Head Torque ($T_H$). The first one is to overcome the friction originated at the threads; the second one is to overcome the friction originated at the head of the bolt in contact with the jointed part.

$$T = T_T + T_H$$

Each torque is calculated separately and then added together in the upper expression. $T_T$ takes into consideration the threads’ helical inclination and the friction while $T_H$ considers only the friction originated at the contact area between the upper part (Aluminum) and the head of the bolt. For metric bolts, Fenollosa directly gives the following simplified formula:

$$T = F_B \cdot [0.16P + 0.58 \cdot \mu \cdot d_2 + 0.5 \cdot \mu_s \cdot d_s]$$

Where $F_B$ is the Bolt Load, $P$ is the pitch of the threads, $\mu$ is the Threads friction coefficient, $d_2$ is equivalent to $d - 0.64953P$, $\mu_s$ is the Head friction coefficient, $d_s$ is the diameter of the bolt’s head and $d$ is the nominal diameter.

For a 5M metric bolt ($P=0.8$ mm, $d_2=4.48$ mm, $d_s=7$ mm, $d=5$ mm), and taking into consideration that the torque applied is 8 Nm, the formula for bolt load turns into:

$$F_B = \frac{8 \text{ Nm}}{10^{-3}[0.128 + 2.60 \cdot \mu + 3.5 \cdot \mu_s]}$$

At this point, bolt load is just a matter of the friction coefficients between bolt and Magnesium ($\mu$) and the head of the bolt and Aluminum ($\mu_s$).

Given the friction coefficients of section 3.4 ($\mu$ and $\mu_s$ are 0.25 and 0.35 respectively) the Tightening Force (Bolt Load, $F_B$) turns to be 5466 N.

### 3.5.1.2 Inner Pressure

As said in the Model Description (section 3.1), there is an inner pressure which is distributed on 4 equal screw fittings and results on a force which tends to separate the two parts. A 3.5 MPa pressure was applied on both aluminum and magnesium surfaces in contact with each other.
3.5.2 Boundary conditions

The axisymmetric condition of this simulation implicitly includes a restriction on the symmetry axis: no x- and y-axis displacement allowed at the points in contact with it.

Despite this restriction, no other boundary conditions are needed. However, Abaqus requires one explicit boundary condition which fix a point. That’s why the upper left vertex of the bolt was pinned (no displacement allowed, rotation permitted). This condition does not affect the results.

3.6 Steps and simulation definition

The way Abaqus organizes the simulations is with analysis steps. Within a model, one or more analysis steps can be defined. Each step allows modifying and/or introducing changes in the interaction between parts, forces, boundary conditions and other fields such as temperature.

In addition, steps allow changing the analysis procedure. This means that in one step time can be an important factor (e.g. creep) while in other step time is not considered (e.g. general stress analysis).

The model to simulate has been divided into three steps, defined below:

- **STEP 1: Apply Bolt Load**
  This first step simulates the process of torquing the bolt. All loads and all boundary conditions are applied. It is a static step, which means that time is not considered in this simulation. The attempt at this first step is to observe the initial stress distribution once the loads are applied.

- **STEP 2: Increase Temperature**
  At this point, the temperature is increased from 20 to 120°C. It is also a static step what means that time is not considered during calculations. However, temperature is increased gradually for better calculation. Thermal expansion is expected to occur, what may lead to some stress relaxation.

- **STEP 3: Creep**
  The creep law presented in section 3.3.4 gets activated in magnesium in this step. This third step is a quasi-static step, which means that time now is an important parameter. The objective is to observe if AZ91D part creeps and at what point stresses relax in a time function.

Steps two and three are used more than once during the simulation. The simulation consists in some cycles where temperatures will increase and decrease between 20 and 120°C. Between these changes of temperature creep will take place. Figure 3.11 illustrates the cycles of the simulation in a temperature vs. time plot.
The simulation consists of four cycles of 800h each where the whole model is heated up to 120°C for 400h and then cooled to 20°C for 400h too. Heating and cooling are considered instantaneous events. At time 0 the forces which interact (bolt load and inner pressure) are applied and they are acting during the whole simulation.

### 3.7 Meshing and element type

#### 3.7.1 Choosing element type

Elements are simple shapes used to discretize the simulated parts in order to do numerical calculations on it. All elements are connected to other elements through the nodes. Abaqus provides an extensive library of element types. Given the wide variety of elements available, it is important to select the correct element for a particular application. Choosing an element for a particular analysis can be simplified by considering specific element characteristics.

Each element has a unique name that identifies each of the five aspects of an element. These aspects are the following: family, number of nodes, degrees of freedom, formulation and integration. The element type decision process has been made following the guidance of Dr. Mohammad Mashayekhi (Mashayenkh, 2011).

#### 3.7.1.1 Family

A family of finite elements is the broadest category used to classify elements. All elements in the same family share many basic features but there are many variations within a family. Figure 3.12 shows some of the most commonly used element families.
For all the steps in the simulation (general static stress-displacement simulation and quasi-static) the solid (continuum) element family is the one used. Elements within this family are quite versatile and can be used for linear and nonlinear analyses involving contact, plasticity and large deformations.

3.7.1.2 Number of Nodes

An element’s number of nodes determines how the nodal degrees of freedom will be interpolated over the domain of the element. Abaqus includes elements with both first- and second-order interpolation.

In axisymmetric modeling there are available two kind of elements within the same family: the 3-nodes triangular elements (6 nodes if second-order) and the 4-nodes quadrilateral elements (8 nodes if second-order). See an example in Figure 3.13.

Second-order elements provide higher accuracy than first-order elements. They outperform first-order elements in problems with stress concentrations; they capture geometric features, such as curved edges, with fewer elements than first-order.

However, in problems where contact between different surface are important, avoiding second-order elements might be a good idea. Convergence difficulties
Method and implementation

can arise with these elements. In addition, first-order triangular elements should also be avoided in stress analysis problems because of their overly stiffness.

Nevertheless, first-order elements do not behave well in bending and present an overly-stiff behavior known as shear locking. This phenomenon occurs due to the four nodes connected by straight lines. Shear locking does not appears in second order elements because bending can be performed thanks to the parabolic lines (second order).

### 3.7.1.3 Degrees of Freedom

The primary variables that exist at the nodes of an element are the degrees of freedom in the finite element analysis. For a stress/displacement simulation the degrees of freedom are the translations and the rotations at each node. As the simulation is axisymmetric, not all the displacements and rotations are available. The degrees of freedom in axisymmetric elements are referred as follows:

1. \( r \)-displacement
2. \( z \)-displacement
3. Rotation about the \( z \)-axis (for axisymmetric elements with twist), in radians
4. Rotation in the \( r \)-\( z \) plane (for axisymmetric shells), in radians

Here the \( r \)- and \( z \)-directions coincide with the global X- and Y-directions, respectively.

### 3.7.1.4 Formulation

The mathematical formulation used to describe the behavior of an element is another broad category that is used to classify elements. There are two big formulation groups: the Lagrangian description of behavior, where the elements deforms with the material; and the Eulerian formulation, where elements are fixed in space as the material flows through them. Most stress/displacement elements in Abaqus are based on the Lagrangian formulation.

### 3.7.1.5 Integration

The stiffness and mass of an element are calculated numerically at sampling points called integration points within the element. The numerical algorithm used to integrate these variables influences how an element behaves.

Abaqus includes elements with both “full” and “reduced” integration. This choice can have a significant effect on the accuracy of the element for a given problem. As shown in Figure 3.14 reduced integration uses one order less integration points than the full integration.
Reduced integration elements tend to be somewhat more efficient: results are often as good as full integration at lower computational cost (lower running time).

### 3.7.1.6 Element choice

Summing up all the advices and decisions made on the prior sections, the element choice needs to meet the following requirements:

- It has to be of the continuum (solid) element family; particularly for an axisymmetric stress simulation.
- Second-order elements. Although Abaqus manual suggests avoiding second order elements because convergence problems may appear, they provide higher accuracy and overcome the shear-locking problem.
- Triangular elements may be avoided because of their stiffness problems leads to a 4-node element.
- Prioritize reduced integration over full integration.

These requirements bring the decision to the following element: CAX8R, a 8-node biquadratic axisymmetric quadrilateral, reduced integration. The element name fully identifies it: C for continuum element, AX for axisymmetric, 8 for 8-nodes and R for reduced integration.
3.7.2 **Meshing**

Each part was meshed separately with CAX8R elements. In order to get good results, elements need to be as right-angled as possible. Sometimes it is not feasible given that the software automatically meshes the parts only with few user guidance.

Since the most critical and interesting zones are threads and their proximities, elements are smaller there and become coarse with distance from the threads. This distribution allows to properly calculate the stress distribution at the threads, which present substantial changes in a short distance.

Several meshes have been tried, with a finer mesh each time. Refining the mesh means to create more nodes (easily hundreds or thousands of new nodes and elements) which increase in such manner the time needed to solve the equations. In consequence, the final mesh is presented below and it’s a trade-off between mesh accuracy and amount of time needed in the simulation.

The final mesh is composed by 4801 elements (220 in aluminum part, 1622 in the bolt, and 2959 in the AZ91D part) and 15290 nodes (723, 5277 and 9290). Figure 3.15 shows the mesh.

![Meshed model](image_url)
3.8 Submit the Job and getting results

Once the model is already created and all the properties are well-adjusted it’s time for the software to start to calculate the numerical solutions to the problem.

Depending on the complexity of the problem being analyzed and the power of the computer being used, it may take anywhere from seconds to days to complete an analysis run. In this case, the simulation was run in 1254.4 seconds.

When the calculations are done, Abaqus creates different output files. Some of them are only usable by itself for the visualization module. Other files are human-readable.
4 Findings and analysis

Once the simulation is done, numerical results are stored in different files. In order to analyze the different solutions of the different moments of the simulation, the Visualization module of Abaqus has been used. This module allows the user to easily see and understand the numerical results obtained.

Before starting with the analysis of the results a comment must be made regarding the geometry. On the last paragraph of section 3.2 the decision to make aluminum and AZ91D parts 15 mm width is presented. Some simulations have been carried out using different sizes for these parts. As a conclusion from these initial simulations, 15 mm are considered to be correct; more width would not have add more value while less than 15 mm may have changed the results.

As said in previous sections, the entire simulation is made of three steps, each with a different goal and some of them are repeated along the cycles. The interpretation of the simulation results will follow the timeline according Figure 4.1 in an attempt to make it as clear as possible.

Thus, the next three sections correspond each one to a simulation step: section 4.1 explains the results after applying the tightening torque (time 0), section 4.2 refers to the moment after heating the model up to 120°C (time 0), and section 4.3 deals with the results from the creep law application and the results obtained from the first cycle. The last section of this chapter, 4.4, deals with the three other cycles.

4.1 Bolt Torqueing

The first step deals with the tightening torque application. Its objective is to analyze how stresses are distributed throughout the model. The torque applied at the bolt was 8 Nm, which resulted to a Bolt Force of 5466 N. There is also an inner pressure of 3.5 MPa which tends to separate the joined parts. Figure 4.2 shows a
Findings and analysis

general view of the stress distribution along the model.

In the Figure 4.2 it is shown that the higher stresses are located at the upper half of the bolt. Stress plotting follows a colors gradient. Lower stresses correspond to blue, intermediate stresses are green and the highest stresses are red.

To fully understand the stress state of this first analysis a more precise observation is needed. In the three next sections the three different parts of the model are studied separately. Figure 4.3 shows the instant when the tightening torque is applied on the bolt. This step does not take time into account; it takes place during the first instant of the simulation.

4.1.1 Bolt

Figure 4.6 (left) shows the von mises stress distribution at the bolt. These stresses range from 6 MPa at the lowest parts of the bolt up to 815 MPa at the first valley (little red spot). However, green and blue are the predominant colors in this area, meaning that higher stresses are only present in local small areas.

Maximum stresses appear at the valleys of the three first threads. At these maximum stress points bolt’s yield strength (640 MPa) is exceeded so plastic deformation appears. Figure 4.4 shows these plastic deformation zones colored in red.

The stresses in the bolt after applying the tightening force are mainly tensile stresses. Figure 4.6 (middle and right) shows the stresses on the X and Y directions respectively. According to the scales next to each image, most of the stresses all over the bolt are tensile. Maximum tensile stresses are in the first valley of the threads. The purpose of the tightening torque applied on the bolt is to stress this part, so it seems reasonable that most of the stresses are tensile.

However, not all the stresses are tensile. There are some zones that present compressive stresses. Figure 4.5 shows the low part of the threads, which is pressing the AZ91D part, shows compressive stresses on the X and Y directions.

There are also compressive stresses on the head of the bolt. These stresses are due to two factors: the contact with aluminum brings compressive stresses in that zone; in the upper left part of the head of the bolt, there is a boundary condition which restricts translation of that point. This restriction also induces compressive stresses.
Findings and analysis

Figure 4.5 Compressive stresses under the threads

Figure 4.6 Stress distribution on the Bolt.
From left to right: Von Mises Stress, Stresses on X direction (S11), Stresses on Y direction (S22)

Figure 4.4 Plastic deformation at Bolt
Stresses (mises) on the outer surface of threads are plotted in Figure 4.7. It is easy to see that the three first threads are the ones that receive the highest stresses. After them, stresses start to gradually decrease down to the lowest stress levels.

Bolt’s contact zone with aluminum also contains some high stresses. These ones are located at the right angles of the head of the bolt. This is not surprising given that sharp edges like the ones present in this part of the model use to be stress accumulation points. However, these stresses are below the yield strength and do not present a problem to continue with the simulation.

4.1.2 AZ91D part

Stress distribution of the AZ91D part is shown in Figure 4.8. The stresses in this part range from 690 kPa at the blue zones up to 216 MPa at the red zones. As can be seen, most of the part is colored blue while, in the proximities of the three first threaded holes, colors evolve to green, yellow and, in very localized areas, red.

As happened on the bolt, maximum stresses appear at the first threaded holes. The highest stresses are also in very localized areas. However, stresses beyond the 139 MPa yield strength, meaning plastic deformation, are more common than in the bolt. In Figure 4.9 plastic deformation zones are shown colored in red.
Plastic deformation is located in the upper portions of the first five threaded holes. The bolt is tightened and it deforms pulling up the AZ91D part. The threads on the magnesium part deform following the bolt, what brings up a bending moment. Figure 4.10 shows this bending deformation on the magnesium threads.

On the other hand, no big stresses neither deformations can be seen from the contact between AZ91D and aluminum parts. Although having a 3.5 MPa separating pressure, the parts remain in contact without compromising the junction.

As happened on the bolt’s threads, there is a gradually decrease of the stresses in the threaded holes of the AZ91D part. Figure 4.11 plots the stresses evolution at these threaded holes.

![Figure 4.10](image1.png)  
**Figure 4.10** Undeformed and deformed first thread engagement. Deformed view scale was greatly increased so differences between both plots could be easily observed.

![Figure 4.11](image2.png)  
**Figure 4.11** Stress evolution at the threads of the AZ91D part
Findings and analysis

Figure 4.12 shows the stresses on the X and Y axis. As said before, all the important information in these part is located around the first threads. As one can see in the figure, almost all the figure is colored green, what means almost no stresses. In little areas, there are tensile (red, orange and yellow) and compressive (light and dark blue) stresses due to the already commented bending. Contact with the aluminum part also induces some compression.

4.1.3 Aluminum part

Aluminum part does not present much trouble in this step. Maximum stresses are about 181 MPa while yield strength is 276 MPa. In conclusion, there is no plastic deformation on this part. Figure 4.13 shows the stress distribution of this jointed part.

Aluminum areas close to the bolt are the most stressed zones. In fact, the head of the bolt and the magnesium act as a pin, gripping the aluminum and making such
stresses appear. This clamping force makes almost all the stresses in the aluminum part to be compressive.

4.2 Heating up to 120°C

Once the bolt load has been applied, the next step consists on increasing the temperature up to 120°C. This is a homogeneous temperature increase, what means that every node in the model experience the same temperature increase. A general view of the stress distribution at the model after the temperature increase is shown on Figure 4.15.

Figure 4.14 shows the instant when the temperature increase happens. This step does not take time into account; it takes place the instant after the tightening torque is applied.
The highest stresses still located at the bolt but they seem to be more spread along this part than before. The next sections focus on each part separately to fully understand the changes occurred.

### 4.2.1 Bolt

Figure 4.16 (left) shows the stress (Mises) distribution at the bolt. The stresses on this part vary from 25 MPa at the dark blue zones up to 822 MPa at the red areas. As happened at the previous analysis, high stresses areas are very small and are located at the valleys of the first threads and at the right angles of the head.

![Stress distribution at the bolt after the Temperature increase. From left to right: Von Mises Stress, Stresses on X direction (S11), Stresses on Y direction (S22).](image)

Figure 4.16 shows the stresses on the horizontal axis. As the previous state, almost all the stresses are tensile stresses. However, there is some compression stresses at the threads, due to the interaction between the bolt and the AZ91D part. This explanation also applies for the vertical stresses, Figure 4.16 (right).
As a result of this temperature rise there is a general increase on the stresses, in great measure originated by the thermal expansion. When temperature increases, the bolt expands. While experiencing this expansion, steel meets against the other parts, which are also expanding, and with the symmetry axis that bans any transfer of material beyond itself.

Figure 4.17 compares the stresses at the surface of the threads at 20 and 120°C (light and dark blue respectively). The figure shows in an easy way that average stresses inside the bolt are quite homogeneous. In addition, stresses at the threads increase at the lower threads. This point is dealt in next section AZ91D part.

4.2.2 AZ91D part

Figure 4.18 shows the stress distribution at the magnesium alloy part. These stresses range from 140 kPa at the dark blue zones up to 160 MPa in the red areas.
As observed on the previous section, the highest stresses still located at the threads zone but unlike the previous step, the most stressed threads are where the bolt ends. This might be related with the difference between steel and AZ91D thermal expansion coefficient.

Steel’s linear thermal expansion coefficient is $13 \times 10^{-6}$ K$^{-1}$, just half of the AZ91D’s, that is $26 \times 10^{-6}$ K$^{-1}$. In other words, steel expands half of what AZ91D does. Where there is contact between the bolt and the part, steel dominates the vertical expansion.

At the thread where the bolt ends, magnesium starts to expand twice as much as on thread above. This bigger vertical expansion originates these high stresses present at the mentioned threads.

According to tensile and compression stresses, Figure 4.19 illustrates the vertical stresses on the AZ91D part. Most of these vertical stresses are compressive, due to the vertical expansion and the clamping force. Horizontal stresses are not important in this part of the simulation. Given that the right part of the model is free, the horizontal expansion occurs without increasing the stresses on the X direction.

Figure 4.20 plots the stresses at the AZ91D threads in room and high temperature (light and dark blue respectively). High temperature stresses are similar for the first two thirds of material. Then stresses start to increase due to thermal expansion and the contact interaction between the bolt and AZ91D.

Comparing both analyses, one can see that in the first threads there is some stress relaxation due to the temperature increase. Nevertheless, the thermal expansion modifies the geometry causing high stresses to appear at zones previously low stressed.
4.2.3 Aluminum part

Figure 4.21 shows the stress distribution on the aluminum. This part correctly copes with the loads and deformations induced by the bolt tightening and the temperature increase.

The lowest stresses are located at the right end of the part and are around 1 MPa. Meanwhile the highest stresses are located near the head of the bolt and have values up to 176 MPa. There is no plastic deformation on this part.

4.3 First cycle

![Figure 4.22 First Creep period (red line)](image)

4.3.1 First Creep period

Once the Bolt Torque is applied and the whole model is heated up to 120°C creep starts its effect for 400 hours.

Figure 4.23 illustrate the stress distribution on the whole model. Some little changes on this distribution can be easily observed but, since the bolt is not creeping and the stresses are much higher in this part, the stress relaxation can't really be seen.

![Figure 4.23 Stress distribution on the global model Before Creeping (A) and after 400h creeping (B)](image)
Findings and analysis

Figure 4.24 shows the stresses evolution of some points during the first creep simulation. The points selected for this graph are nodes under the red line drawn in Figure 4.23 (A).

Relaxation of the highest stresses starts at the very beginning of the creep simulation. It is observable that the amount of relaxation decreases with time.

The amount of creep depends on the initial stress of the node and the deformation each node experiences. This explains why relaxation decreases with time; because stresses decrease and so does creep.

Darker lines plot nodes near the threads; the lines get lighter while the distance from the threads increases.

The lighter lines show a little increase of stress, which
contributes to homogenize the stress distribution. This increase is due to creep of more stressed points (closer to the threads), that get more deformed and compress the areas further the threads.

In order to illustrate better the stress relaxation, the AZ91D (Figure 4.25) and the aluminum (Figure 4.27) parts are illustrated separately, before and after creeping.

In AZ91D part, before creeping there are two zones of concentrated stresses: at the first two/three threads and at the end of the bolt. After 400h at 120ºC stresses are much lower all over the part and they appear to be more homogeneous all along the threads.

Creeping mostly occurs where stresses are high (this can be seen in Figure 4.24, where dark lines show more relaxation than light lines). The plot below (Figure 4.26) shows the stresses at the surface of the threads. Stresses before the creep are plotted in grey color. Red plot is for stresses after creep. There one can see the stress relaxation at the surface of the threads. Although the plot only shows the stresses on the surface, it can be affirmed that stresses in the material are lower and become lower with the distance from the threads.

Creep also affects the aluminum part. Stress distribution on that part is shown in Figure 4.27. After 400h at 120ºC stresses have been lowered and some deformation appears on the contact zone with the head of the bolt. Stress distribution at the upper surface is plotted in the following Figure 4.28.
Stresses are lower after creeping than were before. However, at the point where the head of the bolt ends there are high strains due to deformation. The aluminum under the head of the bolt creeps and deforms plastically while the rest of the aluminum does not. These high strains under the end of the head of the bolt bring high stresses. Despite the high stresses, they are not a problem for the aluminum, which can stand higher stresses.

After this first creep stage of 400 hours it can be concluded that stress relaxation affects the whole model in general. Particularly, the zones with the biggest decrease of stresses are the ones which had the highest tensions before creeping started. After 400 hours stresses in the threads are more homogeneous. Although this stress relaxation, the three parts of the model don’t lose contact between each other and the parts still remain joined. There is also some deformation and stress relaxation in the aluminum part, in particular in the zone under the head of the bolt.
4.3.2 Decrease of Temperature and Low Temperature Creep

Now the temperature is decreased from 120ºC to 20ºC. This cooling brings the model to a new stress state, which is presented in the following lines. The model remains 400 hours at this temperature before a next increase. Due to the high stresses, some creep also affects the parts.

Figure 4.29 shows the stress results after the cooling down and after 400h at room temperature.

After cooling down the model, the stresses increase in the whole part. There are some high stresses which may damage the model. These stresses are located in the lower parts of each thread and are caused by the bending of the threads. Despite this high stresses in very located points, the average stresses of the entire part are below 100 MPa.

After 400h, creep has affected to the highest stresses and some relaxation can be appreciated at these zones. Figure 4.30 shows this stress relaxation at room temperature. The blue curve plots the stresses at the threads of the AZ91D part just after the cooling down and the green one plots these stresses after 400h at 20ºC. As said, just high-stress peaks creep. The red line shows the stress distribution before the model is cooled down.

The aluminum part and the bolt do not change their state distribution in a critical manner and they behave correctly during this period of time at low temperatures. As was said in previous lines, this cooling step has bad effects on the stresses which may damage the model and leave it useless.

However, these high stresses may be caused by the way the simulation has been conducted.

The temperature is decreased instantly. This is unrealistic and brings the model to instantly shrink.

Figure 4.29 Stress distribution after cooling
4.3.3 End of the First Cycle

After the 400h at 20°C the first cycle out of four has finished. This is a good time to compare the stress state at this point with the stress state at the beginning, when the tightening bolt force was applied.

Figure 4.31 and Figure 4.32 show the stress states of the AZ91D part before and after the first cycle. The first figure shows the stress at the threads surface and let the reader see that during this first cycle there has been some stress relaxation. However, Figure 4.32 is more useful when trying to explain the results from this first cycle.

The color legend under the parts of Figure 4.32 shows that the maximum stresses have fallen during the first cycle at least 40%. Therefore, this stress decrease is only at the maximum stress zones: the peaks plotted on Figure 4.31. According to the rest of the part, stresses also decrease but they do it less than 40%. The stress gradient gets wider and stresses become more homogeneous along the whole part.
As a conclusion of this first cycle it can be said that a considerable relaxation on the maximum stresses has been found, together with a homogenization of the stresses on the whole part. There are some conflict areas which may present some trouble: the first threads zones, both from the bolt and the AZ91D part, are the ones which receive most of the stresses and, at some points, overcome the yielding values. However, contact does not disappear. The same happens with the tightening force: it remains joining the parts so the whole model does not lose its function.

---

Figure 4.32 Stress distribution on the AZ91D part before and after the first cycle
4.4 Next Cycles

The following pictures (Figure 4.33) shows the stress distribution on the AZ91D part at the end of each cycle of the simulation under the same scale. It is difficult to see the difference between the pictures of the ending of the cycles because they are very similar.

The reader can also take a look to the APPENDIX 1: Stress on AZ91D threads graphics, where several graphics plot the stress distribution along the threads of AZ91D during the whole simulation. In these graphics it is also difficult to observe differences. With these two evidences, it can be said that after the first cycle, there is no more observable stress relaxation.

Despite there is no more observable stress relaxation after the first cycle, there could be a failure due to fatigue of one of the parts depending on the number of cycles. This fatigue failure is not treated on this work.
Findings and analysis

Figure 4.33 Stress distribution on the AZ91D part before the simulation and after every cycle

Start of the first cycle
Only tightening force

End of the first cycle

End of the second cycle

End of the third cycle

End of the fourth cycle

S, Mises
(Avg: 75%)

- 238.060E+06
- 139.000E+06
- 127.420E+06
- 115.639E+06
- 104.289E+06
- 92.679E+06
- 81.098E+06
- 69.518E+06
- 57.938E+06
- 46.357E+06
- 34.777E+06
- 23.197E+06
- 11.516E+06
- 36.110E+03
4.4.1 Aluminum-Bolt Contact Surface

As the following graphic shows (Figure 4.34), stresses at the contact surface between aluminum and the head of the bolt also decrease during the first cycle but then remain almost the same. There is some notable reduction of stresses at the fourth cycle in the zone where the head of the bolt ends.

However, none of the different stress states found out during the four cycles is a problem for this contact zone and the joint remain fasten.

![Figure 4.34 Stress distribution on the Aluminum-Bolt Contact area](image)

### 4.4.1.1 Pressure between bolt and aluminum

It is interesting to observe the contact forces appearing between the aluminum and the head of the bolt. The variable CPRESS on Abaqus easily gives information on the normal compression developed at contact surfaces. In Figure 4.35 and Figure 4.36, the evolution of this contact pressure is shown. It is shown that stresses get reduced in the first cycle but then remain almost the same at the next cycles. Aluminum and the head of the bolt are always in contact and with enough force to assure the joint.

### 4.4.1.2 Pressure between aluminum and AZ91D part

In order to assess the contact between these parts, the pressure on the surface is plotted in Figure 4.37 and Figure 4.38. These plots confirm that the contact between the parts is always present. The stresses are much higher at the zone under the head of the bolt and get reduced along the surface. However, the parts are always in contact during the four cycles, assuring the joint.
Figure 4.35 Contact pressure between bolt and aluminum during the first cycle.

Figure 4.36 Contact pressure between bolt and aluminum at the end of each cycle.
Findings and analysis

Figure 4.37 Contact pressure between aluminum and AZ91D during the first cycle

Figure 4.38 Contact pressure between aluminum and AZ91D at the end of each cycle
5 Discussion and conclusions

5.1 Discussion of method

To sum up, the methods followed during this work could be grouped in three main areas: a literature survey to grow a theoretical basis, the software Abaqus 6.11 learning which is used for the third method: to simulate the model and to analyze the results.

The literature survey has the objective to catch all the information that will be needed in further stages of the project; in other words, to define the state of the art of the different subjects this work deals with. On the first place, a great number of articles and textbooks have been surveyed looking for magnesium and, more precisely, the AZ91D magnesium alloy.

Another big number of articles were used to understand and define the creep phenomenon. It must be said that it is quite complicated to find literature that gives a global vision of the AZ91D’s properties; most of the literature is very specific and deals with stress or temperature conditions or with some variations of the alloy that do not fit this project.

The presence of threads (screw fittings) in the model could not be avoided. The mechanics regarding bolts and nuts is known to be of a high complexity. That’s why this part of the project has needed quite an important time to correctly characterize the threaded hole that appears in the work. Besides, despite the complexity, a big work has been done to simplify this part and make it understandable and accurate.

The most important work done does not appear in the report; but its consequences do appear. It is the software learning, which began from zero and finished with the simulation done. Despite simplicity was a requirement, the complexity of the problem extended in great manner this stage of the project.

The main problem of this thesis is that it collects very specific properties, which makes quite difficult to find help in manuals and other literature. In particular, great efforts were needed to precisely model and apply the creep function and to calibrate it in order to behave as expected. Finally all these complexities were solved and in a way that makes everything seem simple and understandable, which was an objective of this work.

This learning stage has been carried out at the same time with the model design. Thus, it has been possible to spend more time that expected to the learning process, finishing the design of the model in a prudential time. Once the design was done (and the program features learnt and under control) it was time to run the simulation and spent the rest of the time analyzing the results presented in this report.

The use of Abaqus 6.11 has been essential to get the results. Consequently, it is justified this time dedicated to the correct learning of this software.
However, it must be said that this time spend on learning and getting experience with Abaqus may be time against the simulation. Maybe some other features could be used, or better precision, or numerical stability controls that would improve the reliability and precision of the results.

Nevertheless, the results are good and they answer to the objectives of the project. As everything, this work can be improved. The premise of this project was to go for simplicity and there it has gone, getting, however, rigorous, accurate and reliable results.

### 5.2 Conclusions

During the previous chapter (number 4) the results obtained from the simulation have been shown. Taking in consideration the research objectives at the beginning of this project the conclusions now are presented in a schematic way in order to facilitate the reading and the understanding.

- **Is there any stress relaxation at the screw fittings of the AZ91D part?**

  The first appreciable stress relaxation in AZ91D screw fittings happens right after the model is heated at 120°C when this part starts to deform by creep. This creep deformation runs during all the 400h but high relaxations take place during the first half of this time. At the end, stresses all over the AZ91D part are much lower and the stress distribution happens to be more homogeneous. During the second cycle, there is also a little bit more of stress relaxation. After this second cycle, there is no more observable stress relaxation and the following cycles are similar.

- **Is there any stress relaxation in anywhere else of the model?**

  Stress distribution varies along the timeline in all the different parts of the model. The bolt does not present observable stress relaxation. However, the aluminum part presents some relaxation of the stresses originated by the contact between the head of the bolt and the upper surface of the aluminum.

- **What is the cause of the stress relaxation?**

  The stress relaxation found in this work is originated by deformation due to creep at high temperature of the AZ91D. This deformation frees some stresses. As said in the report, this magnesium alloy is affected of creep with temperatures over 100°C.
A cycle without creep is carried out in APPENDIX 2: Simulation without creep. The stress results with a creep model implemented (section 4.3) are almost half of the stresses from the simulation without creep. This leaves no doubt on that creep affects in great manner to the relaxation of the stresses.

Thermal expansion modifies the model (makes it to expand) and frees some stresses present on the threads. However, this relaxation is very low compared to the creep relaxation.

- **If there is any stress relaxation, how much are the stresses reduced and what are the zones where this relaxation occurs?**

  Given the creep power law that has been used, the most stressed points are the ones that show more relaxation. Thus, the zone around the first threads is the area which reduces more the stresses due to creep deformation.

  The amount of stress reduction depends on the stresses present previous creep effects started so it is different on each point in the model. A good average would be to say that stresses decrease between 40 and 50% in the AZ91D part due to stress relaxation.

- **Does the stress relaxation affect to the usefulness of the model? Does it work after the stress reduction?**

  There are some areas which may present some trouble: the first thread zones, both from the bolt and the AZ91D part, receive most of the forces and, at some points, deform plastically (permanently). Contact between the parts does not disappear despite the deformation of the first threads. Given the results from the numerical simulation, the joint of the two parts is secure and, despite of the stress relaxation, the model is still useful.

### 5.3 Develop and improvements to do

These conclusions are the result of all the work done during five months and treated in this report. As the reader may have noticed there are a great number of hypotheses adopted in order to simplify the model. Some of them could be reviewed with the purpose of continue or develop the work done so far. In the lines below there are some suggestions on how to develop this project.

- **AZ91D’s properties could be obtained empirically.** Testing all the mechanical and physical properties in the laboratory, using specimens of the same material with the same characteristics as the material used in the model. Also creep test could be run with this material, at stress an temperature conditions similar to the ones found in the model.
Discussion and conclusions

These lab tests would be used to improve the materials definition on the simulation and make it even more reliable. Also contact interaction could be improved with actual data on friction coefficients of AZ91D.

- A new creep law could be developed in order to fit better the creep behavior of the AZ91D. In this work, the power law model is used, which fits but it is very general. Geometrics, temperature and stresses influence the creep and maybe a better model could be defined.

- The results of this work could be ratified or improved with a 3D model, with helical threads. This more-realistic geometry would make appear new forces originated by the threads inclination that may change the results a little. Unscrewing forces do not appear in 2D models.

- Vibrations are not considered in this work and they may be an important area to study of an engine. They can be the origin of failures due to fatigue.
6 References


CES Edupack 2012 [Software]. - [s.l.] : Granta Design.


Meshinchi Asl Kaveh Improving the Properties of Magnesium Alloys for High Temperature Applications [Book Section] // Magnesium Alloys - Design,


7 Appendices

7.1 APPENDIX I: Stress on AZ91D threads graphics

In the following pages, plots of each stress state on the threads of the different cycles of the simulation are shown.

7.1.1 FIRST CYCLE

![Graphs showing stress on AZ91D threads for different cycles: Apply bolt load, Increase Temperature up to 120, 400h of Creep at 120, Decrease to 20]
7.1.2 SECOND CYCLE

- Increase Temperature up to 120
- 400h of Creep at 120
- Decrease to 20
- 400h of creep at 20
7.1.3 THIRD CYCLE

Increase to 120

400h of Creep at 120

Decrease to 20

400h of creep at 20
7.1.4 FOURTH CYCLE

- **Increase to 120**

- **400h of Creep at 120**

- **Decrease to 20**

- **400h of creep at 20**
7.2 APPENDIX 2: Simulation without creep

In order to determine whether creep phenomenon affects or not the results of the simulation, one cycle is carried out without a creep law. All the remaining properties announced in this work still active; it is just the creep phenomenon that is been removed.

This cycle without creep consists of the bolt torqueing at room temperature (20 °C), the temperature increase up to 120 °C, and the decrease to room temperature. Given that the bolt load and the first temperature increase are carried out before creep is implemented, it is obvious that the first stages of this unique simulation do not differ from the results of the main simulation. The following pictures illustrate the stress distribution at the end of this simulation in order to compare.

Figure 7.1 General Mises Stress distribution after cooling down the model
Figure 7.2 Stress distribution on the bolt. From left to right: Von Misses, Stresses on the X-axis, and stresses on the Y-axis.
Figure 7.3 Stress distribution on the AZ91D part. From top to bottom: Von Misses Stresses, Stresses on the X-axis, and stresses on the Y-axis