Spring term 2015

Term paper

Project course (TGZC10)
Report number: LIU-IEI-TEK-G-15/00906--SE

Evaluation of Material Parameters of Cast Iron

Department of Management and Engineering
Division of Solid Mechanics

Submitted by:
Paula Márquez (Student-ID: 930731-T101)

Advisor:
Daniel Leidermark, Ass. Professor

ACKNOWLEDGMENT

I am deeply indebted to my supervisor Daniel Leidermark whose stimulating motivation and valuable ideas helped me to complete this work.

I would like to thank to my parents for their economical sacrifice to allow studying abroad and special thanks to my friends for their moral support and love, who were always standing by me in my hard times during this work.

ABSTRACT

The project is based on a previous experiment carry out by two PhD students. The first PhD student analysed how a specimen of cast-iron performs at different temperatures getting some results in the laboratory and the second one, the specimen has modelled using some programs to get the geometry and some necessary values to model it. Now, it is when we have to get conclusions about the two experiments before.

The aim of this project is to prove if the results of the experiments are correct. To do this, we are going to observe, evaluate and see all the experiment done before and to can appreciate in a correct way, we'll transfer in the known Stress-Strain graphic. Finally, we'll compare the graphics from the real experiments obtained by MATLAB with the graphics from ABAQUS and extract some conclusions.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

1. Introduction

1.1. Purpose

The objective of this master of engineering final project is to model the behaviour of a cast iron used for engine parts in Scania trucks. The parts are subjected to thermomechanical fatigue (TMF), which is both varying mechanical and temperature load. The behaviour due to an out-of-phase (OP) load, compression at maximum temperature, is to be characterised and modelled. The part will experience compression at maximum temperature and when unloaded/loaded cyclic it will experience tension at lower temperature. The material exhibit different yield limit and hardening behaviour in tension and compression. Obtaining values of all material parameters through an optimisation process using experimental curves. If enough time is present, model the creep behaviour.

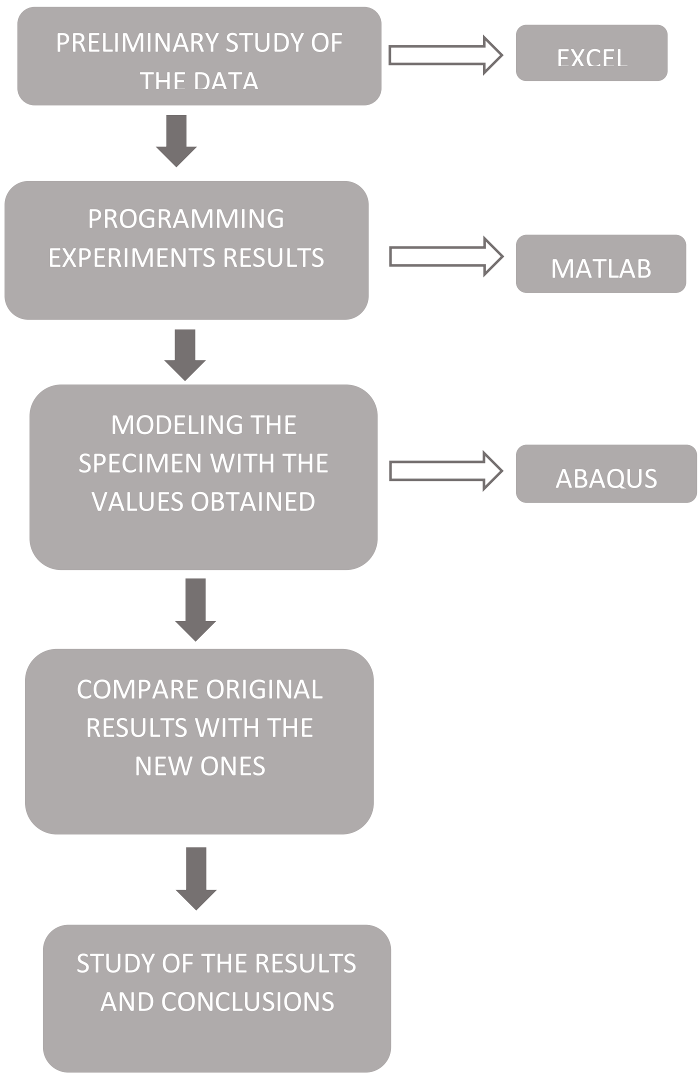
Things that have to be modelled:

• Hardening behaviour

• Material parametrisation

• Creep, if enough time

To be able to model these parts we have used some informatics tools as MATLAB and ABAQUS. A general overview of the project will be:



1.2. What is cast iron?

Cast iron is defined as an iron alloy with more than 2% carbon as the main alloying element. In addition to carbon, cast irons must also contain from 1 to 3% silicon which combined with the carbon give them excellent castability. Cast iron has a much lower melting temperature than steel and is more fluid and less reactive with molding materials. However, they do not have enough ductility to be rolled or forged (Engineers HandBook, 2004).

There are four basic types of cast iron:

- *White iron*: carbon is present in the form of iron carbide (Fe3C) which is hard and brittle. The presence of iron carbide increases hardness and makes it difficult to machine. Consequently these cast irons are abrasion resistant.

- *Grey iron*: carbon here is mainly in the form of graphite. This type of cast iron is inexpensive and has high compressive strength. Graphite is an excellent solid lubricant and this makes it easily machinable but brittle.

- *Ductile iron:* graphite is present in the form of spheres or nodules. They have high tensile strength and good elongation properties.

- *Malleable iron*: these are white cast irons rendered malleable by annealing. These are tougher than grey cast iron and they can be twisted or bent without fracture. They have excellent machining properties and are inexpensive.

Beneath we can see the range of compositions of the main types of cast iron, see Table 1.

Type of cast iron	Carbon	Silicon	Manganese	Sulfur	Phosphorus
White iron	1.8-3.6	0.5-1.9	0.25-0.8	0.03-0.2	0.06-0.2
Grey iron	2.5-4	1.0-3.0	0.2-1.0	0.02-0.25	0.02-1.0
Ductile iron	3.0-4.0	1.8-2.8	0.1-1.0	0.01-0.03	0.01-0.1
Malleable iron	2.0-2.9	0.9-1.9	0.15-1.2	0.02-0.2	0.02-0.2

Table1. Range of Compositions (values in percent (%)).

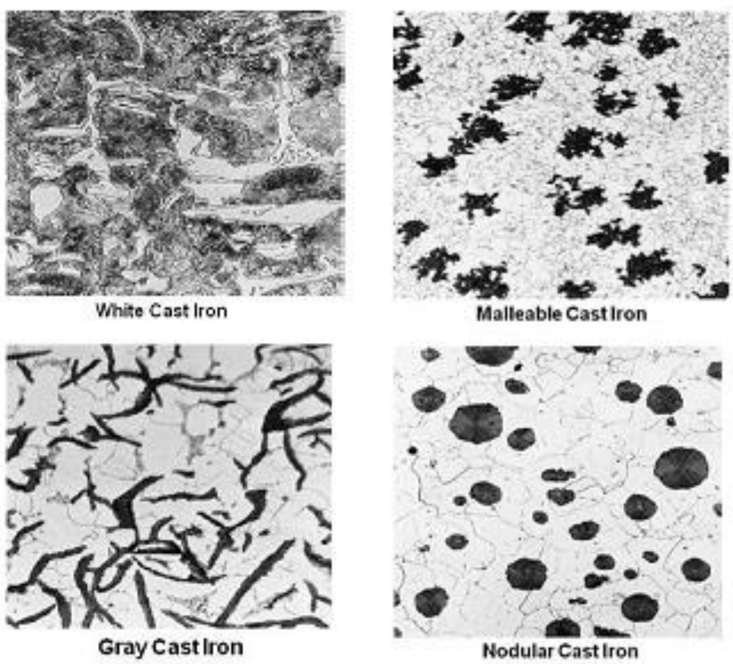


Figure 1. Microstructure of the main cast iron [4].

1.3. Elastic Deformation

Elastic deformation is defined as reversible alteration of the form or dimensions of a solid body under stress or strain. From Figure 2, many Stress-Strain curves for different materials are shown and to identify the elastic range, for the cooper, for example, it is the linear portion of the curve up to $50*10^3$ psi[1]. In this elastic range, the specimen is put under a load but it has the ability to return to its initial shape.

The elastic deformation is defined by Hookes Law in one dimension that states:

σ = E*Ɛ

where

σ = Stress

E = Young Modulus or Modulus of Elasticity

Ɛ = Strain

This formula is only valid in the elastic region which limit is the yield point (see Figure 2, the blue points are the yield limit) and when the yield point is reached, the material starts with the plastic deformation.

[1] psi: pounds per square inch that it is an unit of pressure or stress based on avoirdupois units

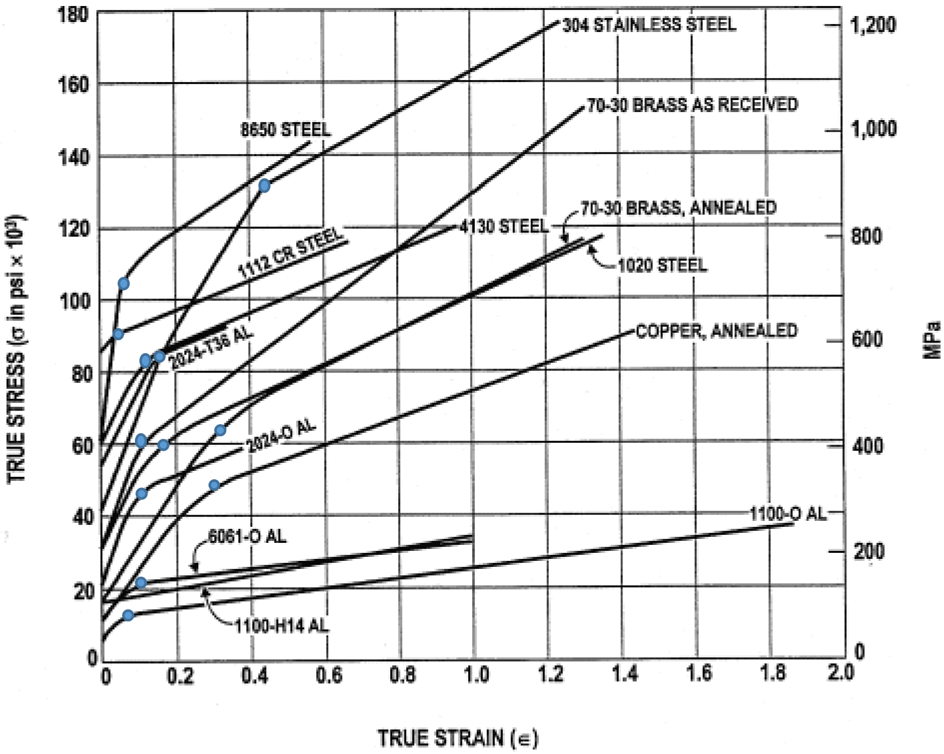


Figure 2.True stress-True strain curve

In ABAQUS, when we have to analyse specimens within the elastic range, only the Poisson ratio and Young Modulus are necessary. The analysis is assumed to be elastic until the yield point and as in the plastic region we don't specify anything, ABAQUS assume that in this part, the behaviour continues being elastic. So, if we only select the elastic option when a material is defined, we are assuming that the behaviour of the material is only elastic.

1.4. Elastic-Plastic

Plastic deformation is defined as a permanent change in shape or size of a solid body without fracture resulting from the application of sustained stress beyond the elastic limit.

If we take a look in Figure 2, for the same example that before, the plastic range starts after the $50*10^3$ psi value. In this plastic range, the specimen is put under a load but it hasn't the ability to return to its initial shape.

In ABAQUS, when we have to analyse specimens within the elastic-plastic range, the 'Poisson ratio', 'Young Modulus', 'Plastic strain' and of course, the 'Yield point' have to be defined. The analysis is assumed to be elastic until the yield point but after that, the analysis is in the plastic range because the yield point has been introduced.

Work or cold hardening is a consequence of plastic deformation and it consists in a permanent change in shape. Unlike the elastic deformation, the plastic deformation is not reversible. The materials usually exhibit both deformations being the work hardening most notably for ductile materials such as metals. Materials behave elastically until the deforming force increases beyond the elastic limit. At that point, the material is permanently deformed so there is a permanent change in shape when the force is removed. This phenomenon is called plastic deformation.

1.5. Programs

To get results and compare different data, two programs have been used: MATLAB and ABAQUS.

- Abaqus FEA or ABAQUS is a software application used for both the modelling and analysis of mechanical components and assemblies (pre-processing) and visualizing the finite element analysis result.

- MATLAB is a multi-paradigm numerical computing environment and fourth-generation programming language. Developed by MathWorks, MATLAB allows matrix manipulations, plotting of functions and data, implementation of algorithms, creation of user interfaces, and interfacing with programs written in other languages, including C, C++, Java, Fortran and Python.

2. Methodology and Approach

2.1. Methodology

The method for investigating the properties of the cast iron is to programme in MATLAB and then, to model in ABAQUS the specimen in order to check the values of the material provided from the experiments.

Prior to start analysing the data, an overview was done in Excel to get a previous idea about the results we should get.

The first analysis is programming in MATLAB the data files from the experiments to get the Young Modulus values with the aim of applying these values in ABAQUS in the second analysis.

2.2. Approach

This project will provide step by step direction to model a cast iron specimen in order to verify the performed experiments. The verification of the values is made through software tools as MATLAB and ABAQUS. First, we'll programme the experiments data in MATLAB to get the Young Modulus and then we'll model the specimen in ABAQUS inserting all the material properties and values obtained before. The end results of this project will show us if we are able to model an appropriate specimen according to the real cast-iron specimen used in the laboratory.

3. Material properties of cast iron

To implement properly the properties of the cast iron it is fundamental to have in mind that the behaviour of the material depends on temperature so this dependency must be shown in ABAQUS. It is also necessary to distinguish between the Elastic Region and Plastic Region.

For the *Elastic Region*, the known properties are the Young Modulus and the Poisson Ratio. The manner to implement them is selecting 'Use temperature-dependent data' and fill all the gaps in Edit Material.

Beneath we have the Young Modulus values obtain from MATLAB for each temperature.

Young Modulus (MPa)	Temperature (ºC)
175.606	25
167.045	100
135.833	400
126.785	450
113.333	500

Table 2. Reminder of the Young Modulus for each temperature

The Poisson Ratio must also be known to define the material. This value has been obtained by searching the net. (The Engineering Tool Box, s.f.)

Poisson Ratio (v)	0.211

Table 3. Poisson Ratio value

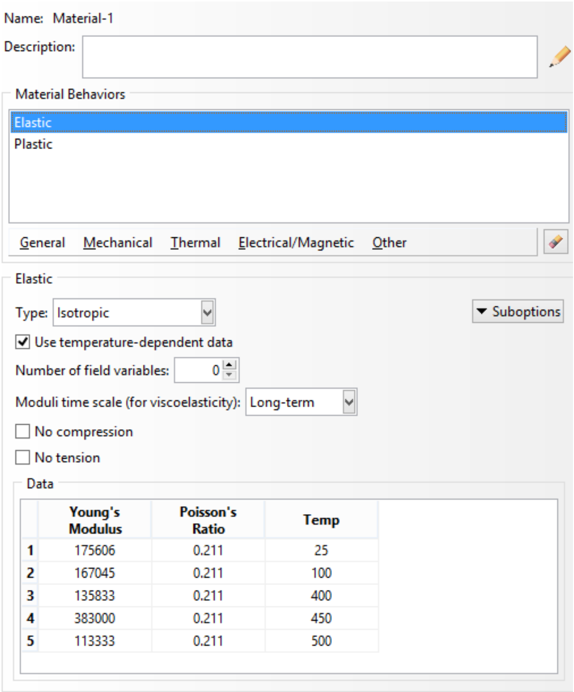


Figure 3. Elastic range in ABAQUS

Once studied the Elastic Region, it is time to study the Plastic Region to get the hardening parameters given that the Young Modulus and the Poisson Ratios are still known. The idea is to make graphs through the values from ABAQUS in order to compare them with the values provided from the laboratory.

This method is based on defining the hardening like “kinematic” in ABAQUS.

The Yield Point, defined as the stress level at which the material ceases to behave elastically, has to been calculated.

Beneath we can see in the graphic where the Yield Point and the Fracture are approximate.

BEHAVIOUR OF DUCTILE METAL UNDER TENSILE FORCE

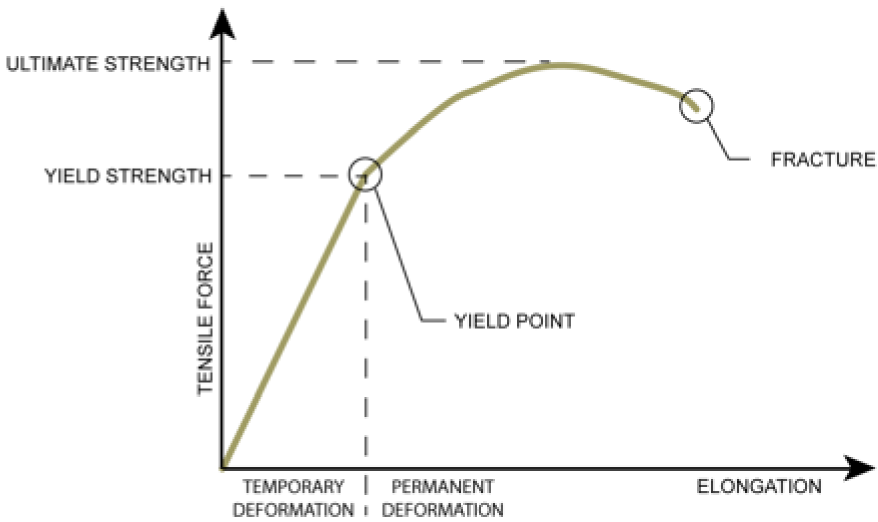


Figure 4. Behaviour of ductile metal under tensile force

The method used to calculate the Yield Point is the 0.2% offset method applying to Stress-Strain graphics obtained in MATLAB for each temperature. It consists in getting the value of the Yield Point drawing a line with slope equal to the Young Modulus at 0,2% Strain and the point when the line cuts, it is the Yield Point.

3.1. 25ºC

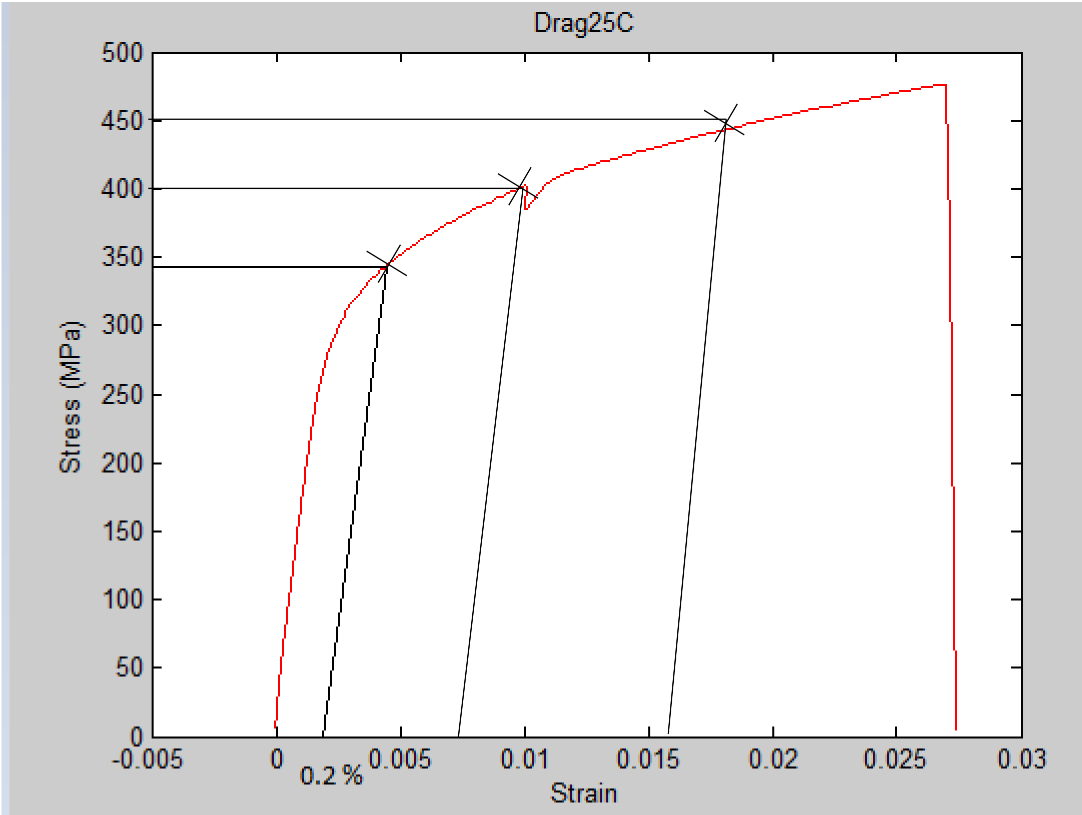


Figure . Calculation of Yield Point at 25ºC

To define the material in the Elastic-Plastic Region, at least, it is necessary to implement two values of Yield Point so we have decided to take three values to make it accurately.

Yield point (MPa)	Plastic Strain
345	0
400	0.0075
450	0.016

Table 4. Yield point and plastic strain at 25ºC

3.2. 100ºC

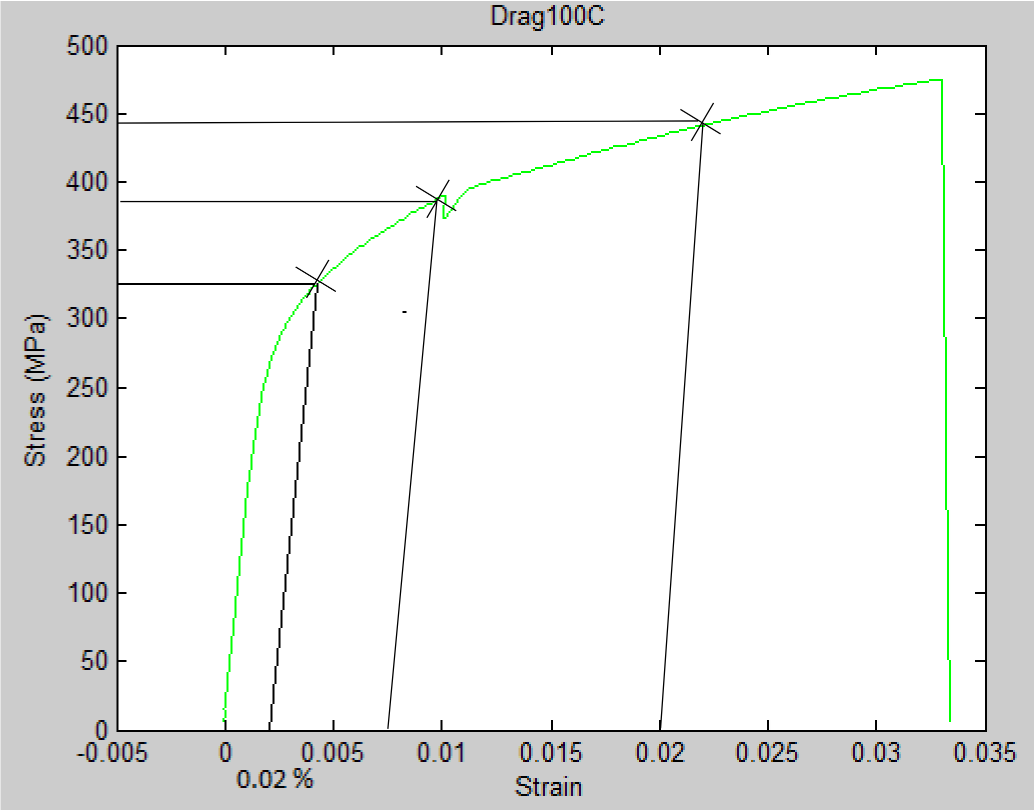


Figure 6. Calculation of Yield point at 100ºC

To define the material in the Elastic-Plastic Region, at least, it is necessary to implement two values of Yield Point so we have decided to take three values to make it accurately.

Yield point (MPa)	Plastic Strain
325	0
390	0.0075
445	0.02

Table 5. Yield point and plastic strain at 100ºC

3.3. 400ºC

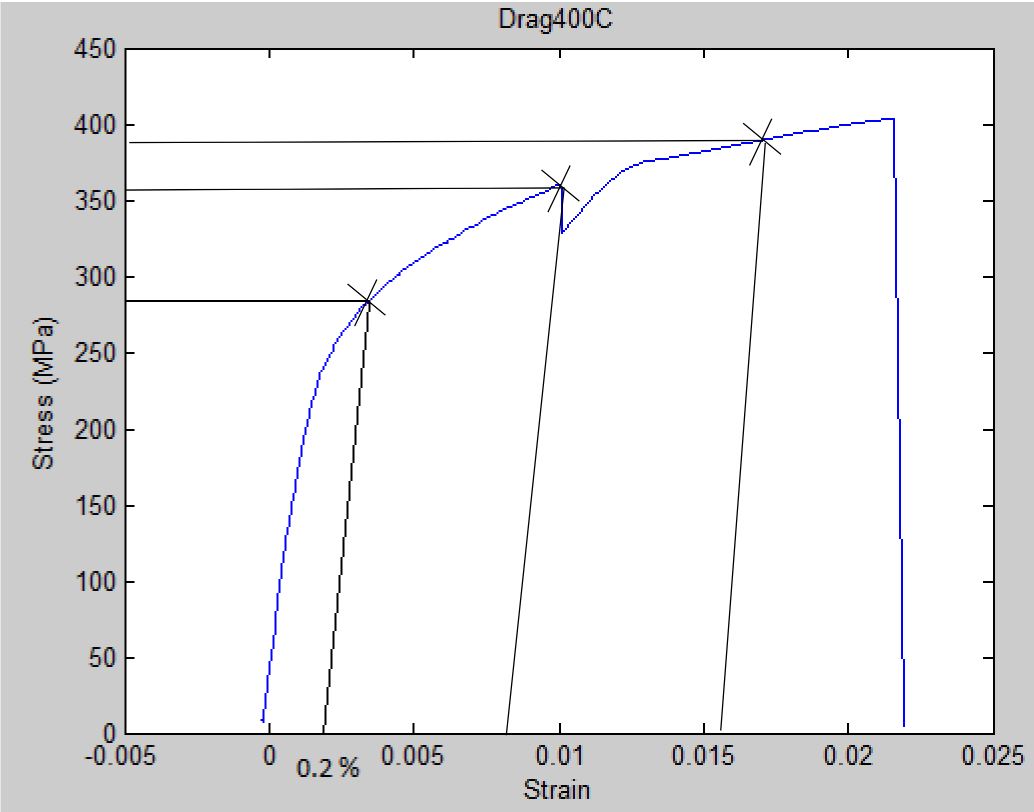


Figure . Calculation of Yield point at 400ºC

To define the material in the Elastic-Plastic Region, at least, it is necessary to implement two values of Yield Point so we have decided to take three values to make it accurately.

Yield point (MPa)	Plastic Strain
290	0
355	0.0085
390	0.016

Table 6. Yield point and plastic strain at 400ºC

3.4. 450ºC

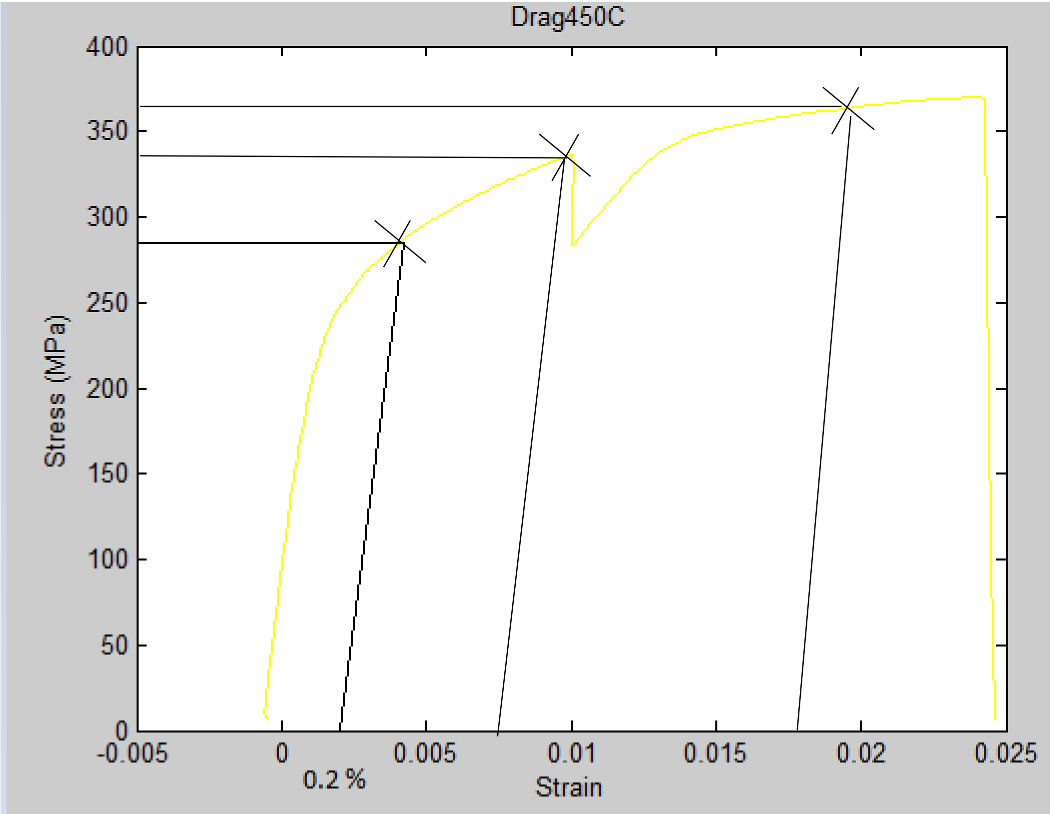


Figure 8. Calculation of yield point at 450ºC

To define the material in the Elastic-Plastic Region, at least, it is necessary to implement two values of Yield Point so we have decided to take three values to make it accurately.

Yield point (MPa)	Plastic strain
290	0
340	0.0075
360	0.0175

Table 7. Yield point and plastic strain at 450ºC

3.5. 500ºC

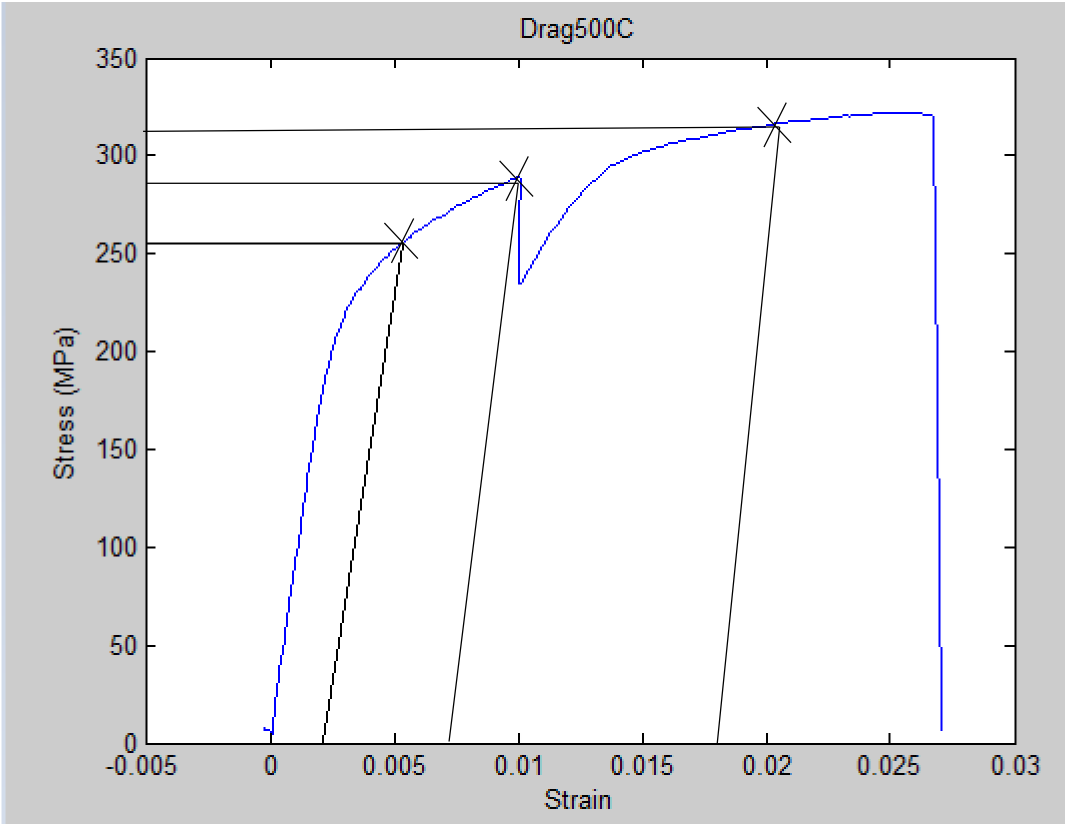


Figure 9. Calculation of yield point at 500ºC

To define the material in the Elastic-Plastic Region, at least, it is necessary to implement two values of Yield Point so we have decided to take three values to make it accurately.

Yield point (MPa)	Plastic strain
255	0
285	0.0075
310	0.019

Table 8. Yield point and plastic strain at 500ºC

We can define properly the material properties after knowing all these values.

4. Implementation of the material properties and results

Once implemented the properties, the mesh and the boundary conditions, it is time to run the program and see what the results are.

The established goal is comparing these results with the experiments results.

The chosen option to do that is making a graphic for the Elastic-Plastic Region of the cast iron at each temperature with the ABAQUS data in order to compare them with the MATLAB graphics.

When we have defined the Plastic Region in ABAQUS, we have selected the “Kinematic” option which means only two Yield Points and Plastic Strain can be defined for each temperature. We have three values so we have created two graphics with ABAQUS data.

To select the node to study, we have chosen a node in the middle of the specimen. This option has been chosen due to in the experiment done, the extensometer to get the data, was put in this region. So, when we see the data in the modelled specimen, we can see that there are a range with the same values so it is just to take on of the nodes of this range.

Once obtained these three graphics (one from the experiment and the two others from the ABAQUS), we have mixed them in an unique plot to see if they are similar which indicates if the approximation done in ABAQUS is good.

Beneath, we can see the three graphics together and as we can observed the approximation is quite good due to the curve from the experiments is pretty similar to the curve from ABAQUS. There is a little difference at the end of the elastic region, but this is probably due to there aren´t so many values in ABAQUS compared to MATLAB and instead of joining two points with a curve, ABAQUS join two values with a slope.

To create the following plots with three graphics at the same time, we have used Excel. The MATLAB data is the data from experiments so we only have to implement these values to create the graphic in Excel afterwards.

Making the graphics with ABAQUS data is a bit different. Firstly, two graphics have been created in ABAQUS with the aim of extract these values. Once extracted these values, this has been implemented in Excel to create the graphic.

Beneath, a legend is showed to understand from where it is each curve (it is the same for each temperature, the same colours).

(yellow)	Curve from MATLAB
(blue)	Curve from ABAQUS
(green)	Curve from ABAQUS

4.1. 25ºC

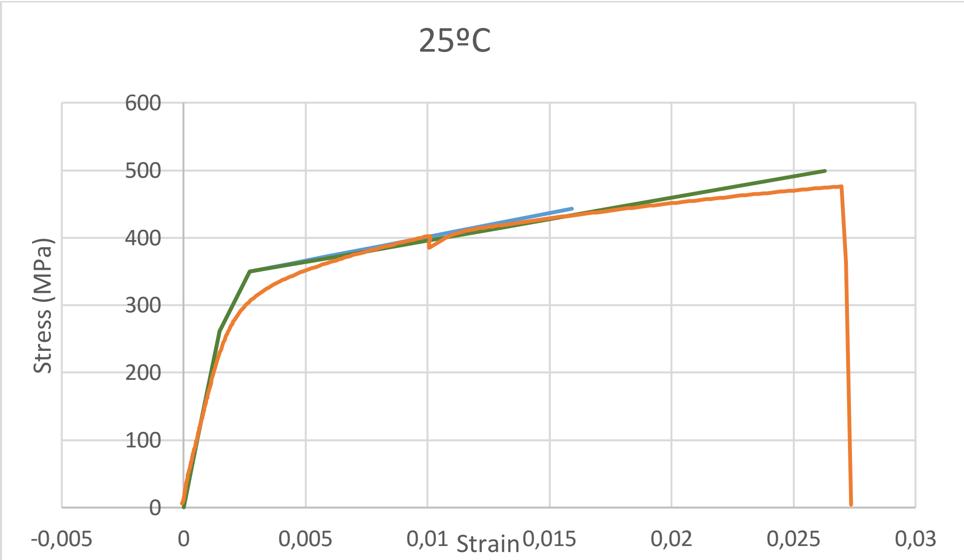


Figure 10. Comparison of the results at 25ºC

4.2. 100ºC

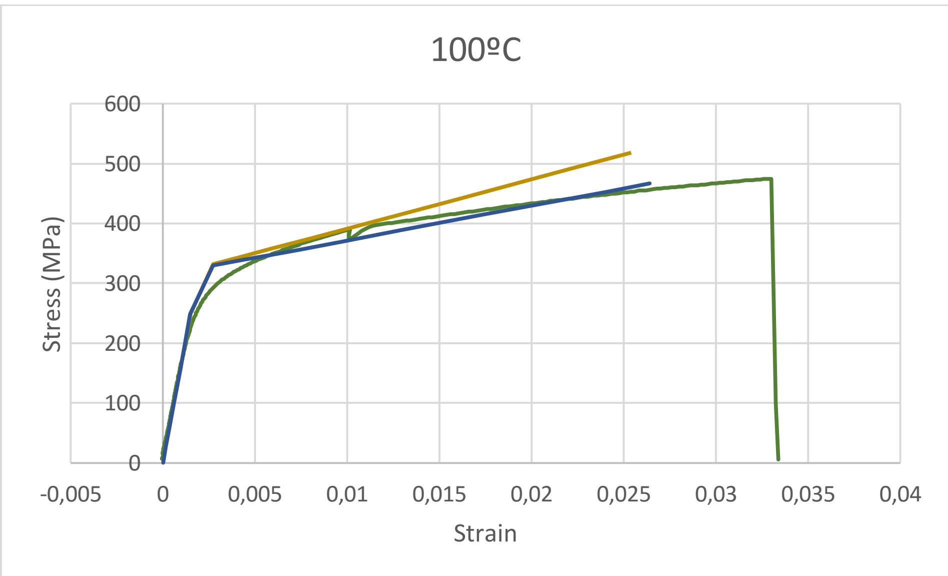


Figure 11. Comparison of the results at 100ºC

4.3. 400ºC

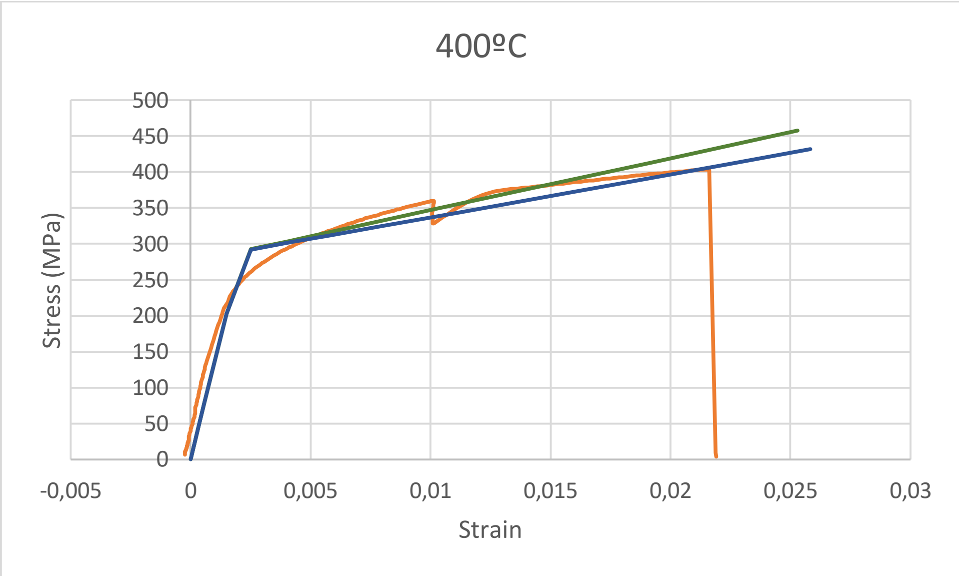


Figure 12. Comparison of the results at 400ºC

4.4. 450ºC

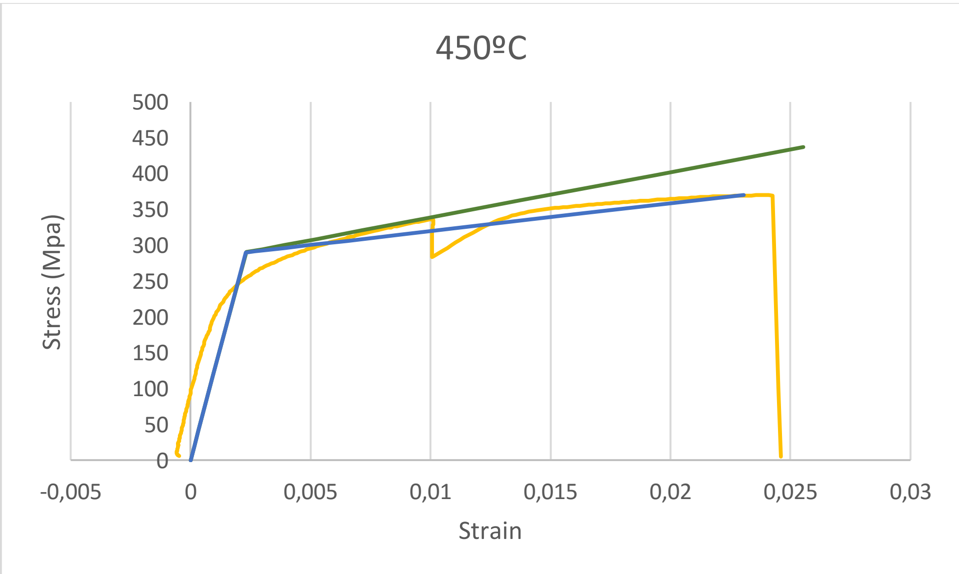


Figure 13. Comparison of the results at 450ºC

4.5. 500ºC

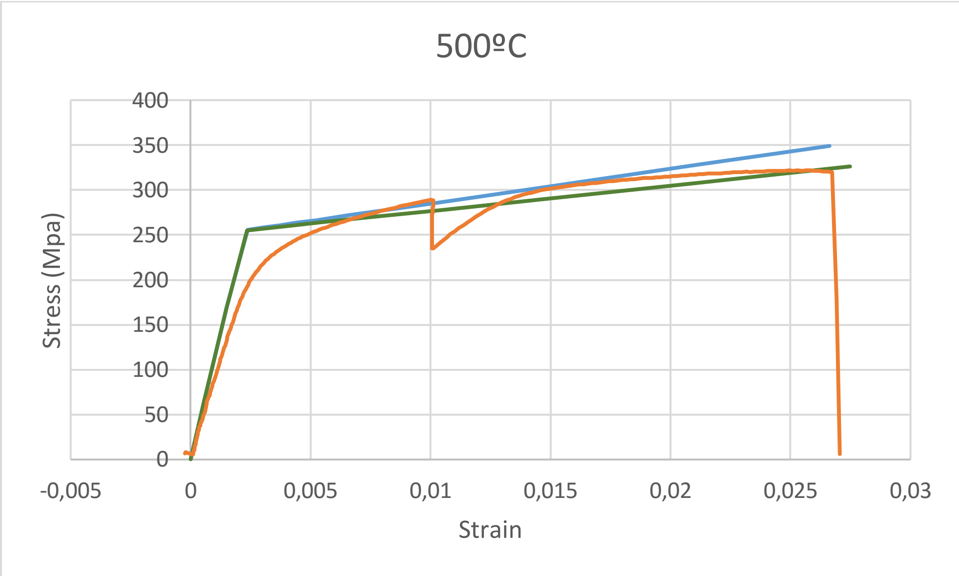


Figure 14. Comparison of the results at 500ºC

5. Hardening parameters

Now it is time to study the hardening parameters:

- *γ*: kinematic hardening modulus.
- *C*: rate at which hardening modulus decreases with plastic straining (σ_0, yield stress at zero plastic strain*)*.

Instead of selecting the kinematic hardening in ABAQUS, we have selected the combined hardening which requires the hardening parameters named before.

We need to get these parameters for each temperature. To obtain them we have used an equation which combines these two parameters.

$$\alpha_k = \frac{C_k}{\gamma_k}\left(1 - e^{-\gamma_k \varepsilon^{pl}}\right)$$

Figure 45. Formula to get the hardening parameters

What we know in this equation is the strain; the overall backstress α_k can be calculated through the stress with the following formula:

$$\alpha_i = \sigma_i - \sigma_i^0.$$

Figure 46. Overall backstress formula

We extract three values from each graphic in order to have two equations to be able to solve the unknown variables.

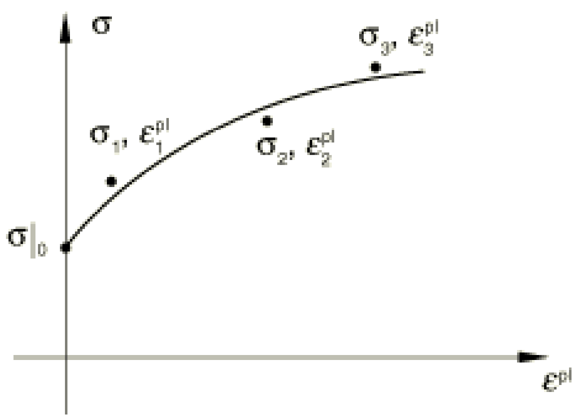


Figure 15. Plastic range of a stress-strain curve

5.1. 25 ºC

Remembering the values calculated by MATLAB, we have the following:

Yield Point (MPa)	Plastic Strain
345	0
400	0.0075
450	0.016

First, we have calculated the overall backstress:

α_1= 400-345=55 MPa

α_2= 450-345=105 MPa

Now, we have the equation system ready to solve (two equations and two variables):

$$55 = \frac{C}{\gamma}(1 - e^{-0.0075\,\gamma})$$

$$105 = \frac{C}{\gamma}(1 - e^{-0.016\,\gamma})$$

MATLAB has been used to solve the equation getting these results:

C	8119.167
γ	27.622

5.2. 100 ºC

Remembering the values calculated by MATLAB, we have the following:

Yield Point (MPa)	Plastic Strain
325	0
390	0.0075
445	0.02

First, we have calculated the overall backstress:

α_1= 390-325=65 MPa

α_2= 445-325=120 MPa

Now, we have the equation system ready to solve (two equations and two variables):

$$65 = \frac{C}{\gamma}(1 - e^{-0.0075\,\gamma})$$

$$120 = \frac{C}{\gamma}(1 - e^{-0.02\,\gamma})$$

MATLAB has been used to solve the equation getting these results:

C	11132.223
γ	69.8007

5.3. 400 ºC

Remembering the values calculated by MATLAB, we have the following:

Yield Point (MPa)	Plastic Strain
290	0
355	0.0085
390	0.016

First, we have calculated the overall backstress:

α_1= 335-290=45 MPa

α_2= 390-290=100 MPa

Now, we have the equation system ready to solve (two equations and two variables):

$$45 = \frac{C}{\gamma}(1 - e^{-0.0085\,\gamma})$$

$$100 = \frac{C}{\gamma}(1 - e^{-0.016\,\gamma})$$

MATLAB has been used to solve the equation getting these results:

C	4427.715
γ	-40.8684

If we observe the results, we can appreciate that the gamma-1 value is negative and this is not possible because it has to be positive. We tried to solve the equation proving values but it wasn't possible. This is due to the equations are compatibles and linear independent so there is only one possible solution. For that reason, and with the aim of avoiding problems in ABAQUS, this value hasn't been considered when we do the modelling.

5.4. 450 ºC

Remembering the values calculated by MATLAB, we have the following:

Yield Point (MPa)	Plastic Strain
290	0
340	0.0075
360	0.0175

First, we have calculated the overall backstress:

α_1= 340-290=50 MPa

α_2= 360-290=70 MPa

Now, we have the equation system ready to solve (two equations and two variables):

$$50 = \frac{C}{\gamma}(1 - e^{-0.0075\,\gamma})$$

$$70 = \frac{C}{\gamma}(1 - e^{-0.0175\,\gamma})$$

The results have been obtained by MATLAB.

C	10824.229
γ	141.673

5.5. 500 ºC

Remembering the values calculated by MATLAB, we have the following:

Yield Point (MPa)	Plastic Strain
255	0
285	0.0075
310	0.019

First, we have calculated the overall backstress:

α_1= 285-255=30 MPa

α_2= 310-255=55 MPa

Now, we have the equation system ready to solve (two equations and two varaibles):

$$30 = \frac{C}{\gamma}(1 - e^{-0.0075\,\gamma})$$

$$55 = \frac{C}{\gamma}(1 - e^{-0.019\,\gamma})$$

The results have been obtained by MATLAB.

C	5064.55
γ	65.61

The next step is to introduce all these values in ABAQUS to generate a new stress-strain curve and see what happen and compare. To get it, we have to impose that the 'Hardening' is combined and the 'Data type' is with parameters.

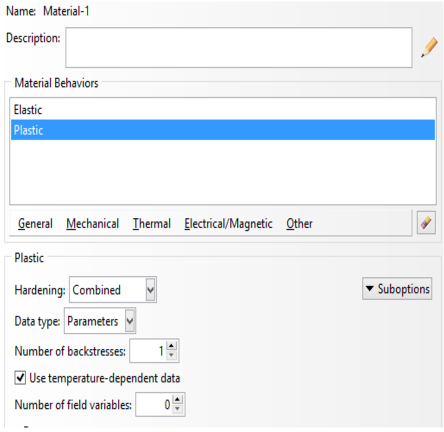


Figure 16. Screenshot of ABAQUS

Introducing all these data, the stress-strain curve for the cast iron is:

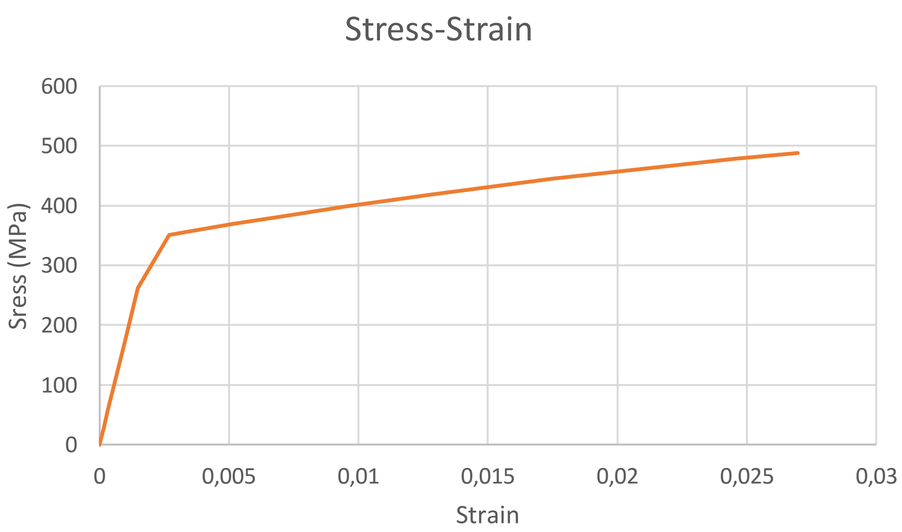


Figure 17. Stress-Strain curve with the hardening parameters

This curve is quite similar to the curves that were drawn before. It can be some differences between this one and the curves from MATLAB, but this is caused by the differences of the programs (the way to join two points, the amount of available data…) so a new model has been tried, using the same data and reducing the time step (initial time step was 0.01 and the new one is 10^{-6}) with the goal of getting more points and ABAQUS is able to draw a better curve. The drawn curve is:

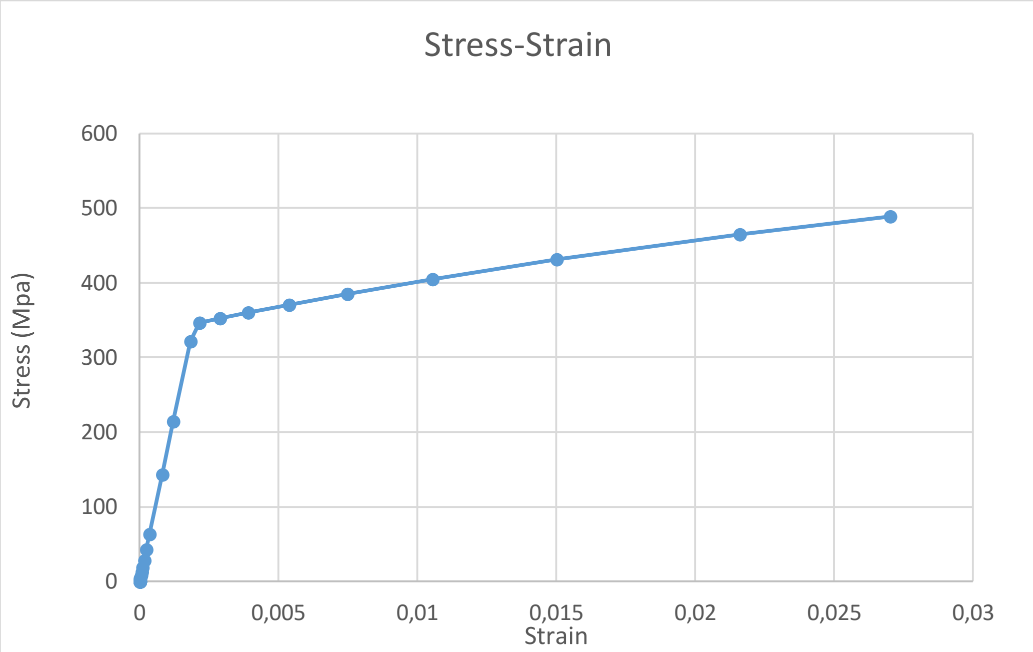


Figure 18. Stress-Strain curve with the new time step

As we can see above, the curve for the non-linear approximation is quite accurate but there is a little problem because it is not possible to draw the curve part in the plastic range even increasing the time step.

5.6. Special case: hardening parameters at 25ºC

In order to see what happen with the C and γ value if instead of using the 0.2 % offset to get the yield point, another approximation tool is used. For doing that, only one range of temperature has been selected and this is neccesary to appreciate the diference between both.

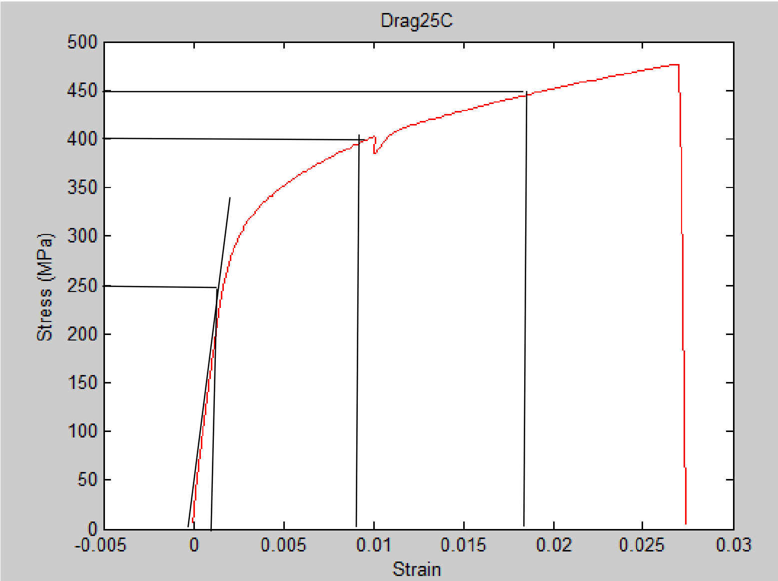


Figure 19. New yield point and plastic strain without using the 0.2% offset

Here, the new values are shown and the first difference comparing with the old one is that the yield points are the same but the plastic strains are a little bit high.

Yield point (MPa)	Plastic Strain
250	0
400	0.009
450	0.018

First, we have calculated the overall backstress:

α_1= 400-250=150 MPa

α_2= 450-250=200 MPa

Now, we have the equation system ready to solve (two equations and two variables):

$$150 = \frac{C}{\gamma}(1 - e^{-0.009\,\gamma})$$

$$200 = \frac{C}{\gamma}(1 - e^{-0.018\,\gamma})$$

MATLAB has been used to solve the equation getting these results:

C	11925
γ	53.0777

Introducing these values in ABAQUS in the same way we did before, a new curve is got and when we plot these values, the following curve appears:

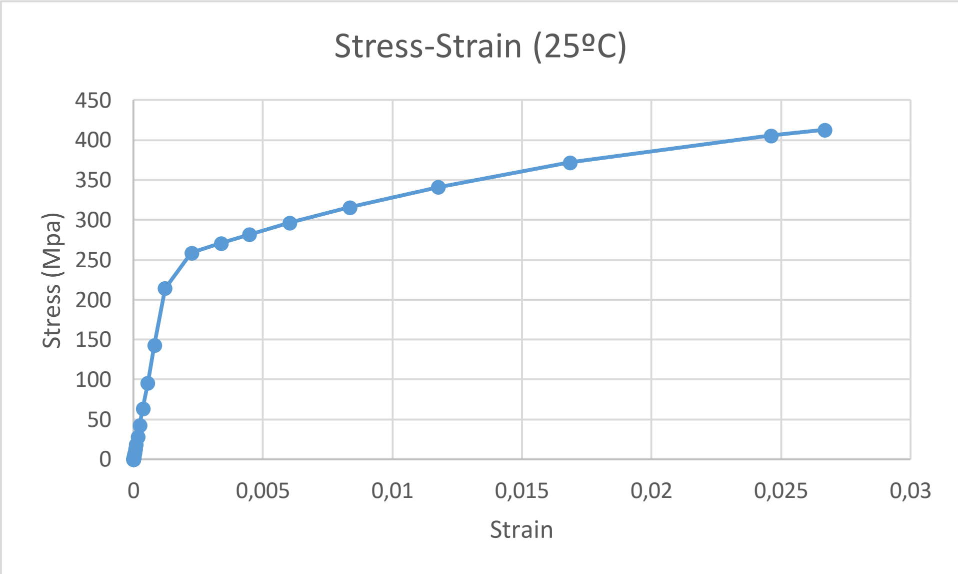


Figure 20. Stress-Strain curve with the new yield points

This is the comparison between the curve of the experiment (in orange) and the new curve (in blue). If we take a quick look, a big difference can be observed in the plastic range. The elastic range between both is pretty good but in the plastic range, it isn't. This can be due to we don't have enough values to get the new curve and ABAQUS can't draw a good curve because of that.

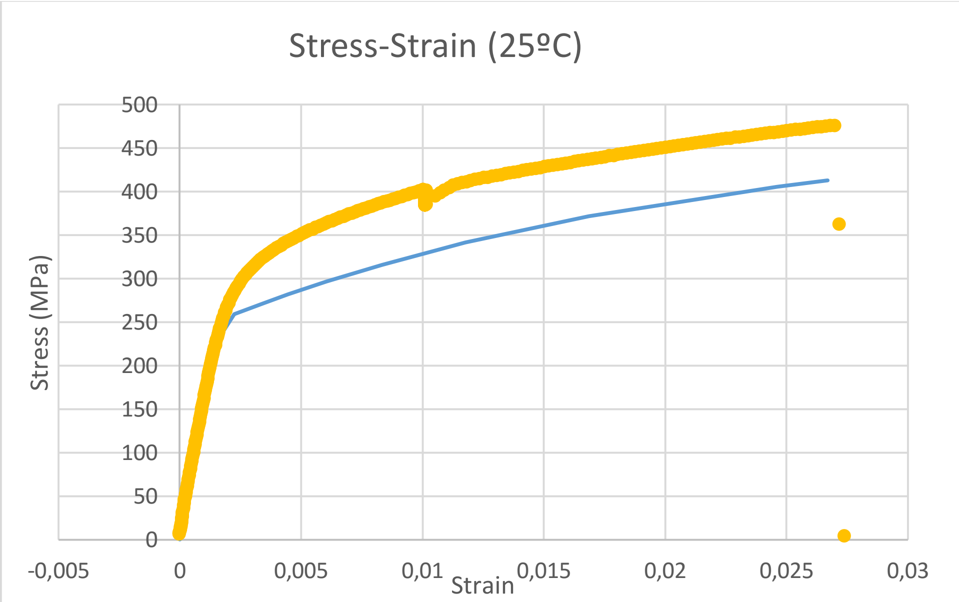


Figure 21. Comparison between 25ºC curves

6. Conclusion

This paper describes an approach to model the behaviour of a specimen of cast iron material through some software tools with the aim of check if the results provided from the experiments are the same as the results obtained from these software tools.

Firstly, we did a previous study of the data in Excel in order to get a preliminary idea about the values. This study gave us the idea to the results we will have to get once done the modelling and the study of the specimen in a software.

After the previous study, we got in MATLAB some Strain-Stress curves for the cast iron at different temperatures. The aim of these curves was obtained the Young Modulus value so later we implemented these properties in ABAQUS concurrently to model the specimen. Finally, we got the Stress-Strain curves from ABAQUS for the elastic and plastic region. To get an accurate approach of the plastic region we used two approximations: linear and non-linear hardening parameters.

Once obtained these graphics, we compared them with the first graphics and the conclusions is the approach done in ABAQUS and MATLAB for the cast-iron specimen has been pretty accurate. The new Stress-Strain curves obtained have been very similar comparing to the original curves from experiment results therefore the meaning is the properties obtained in the experiments for the cast iron are true due to we have been able to get almost the same values using some computer tools. We have to say there are some differences between the graphics but they are totally acceptable due to in the experiments we have lot of values so the curve is more accurate as more values you have.

To conclude, we can assure the properties of the cast iron from the experiments are well-aimed so we can work with them.

7. References

Engineers HandBook. (2004). Obtenido de http://www.engineershandbook.com/Materials/castiron.htm

J.Martin Sanjosé, M. M. (2004). *INGENIERÍA DE MATERIALES PARA INDUSTRIA Y CONSTRUCCIÓN.* Zaragoza: MIRA EDITORES, S.A.

Sever, M. (2014). *Properties of cast irons used in diesel engines for heavy duty trucks – Modelling the mechanical behavior of cast iron at high-temperature.*

Shah, K. P. (2006). *The Hand Book of Mechanical Maintenance*. Obtenido de http://practicalmaintenance.net/?p=1611

Szwaja, N. (2012). *Elastic and Elasto-Plastic Finite Element Analysis of a Tension Test.*

The Engineering Tool Box. (2010). Obtenido de http://www.engineeringtoolbox.com/poissons-ratio-d_1224.html

Total Materia. (August 2010). Obtenido de http://www.keytometals.com/Aluminum/page.aspx?ID=CheckArticle&site=kts&NM=280