Linköping University

Spring term 2015

Term paper

Project course (TGZC10)

Report number: LIU-IEI-TEK-G-15/00906--SE

Modelling the Constitutive Behaviour of Cast Iron

Department of Management and Engineering

Division of Solid Mechanics

Submitted by:

Sara Márquez (Student-ID: 930731-T085)

Advisor:

Daniel Leidermark, Ass. Professor

ACKNOWLEDGEMENT

I wish to express my sincere thanks to Daniel Leidermark, my project coordinator, for providing me with all the necessary facilities.

I place on record, my sincere gratitude to my parents for their unceasing encouragement and support and thanks to one and all who, directly or indirectly, have lent their helping hand in this venture.

ABSTRACT

The project is based on a previous experiment carry out by a PhD student. This student analysed how a specimen of cast-iron performs at different temperatures getting some results that we will analyse.

The aim of this project is to prove if the results of the experiments are correct. To do this, we are going to do a preliminary study of the data to have a first review of these values, then we'll programme in MATLAB the experiment values in order to get the Young Modulus properly and finally, we'll model the cast-iron specimen in ABAQUS implementing the properties obtained steps.

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:

A general overview of the project will be:

```
┌─────────────────────┐
│  ANALYSE AND        │
│  UNDERSTAND THE     │
│  DATA FROM THE      │
│  EXPERIMENT         │
└─────────────────────┘
          │
          ▼
┌─────────────────────┐         ┌──────────┐
│  PRELIMINARY STUDY  │ ══════▷ │  EXCEL   │
│  OF THE DATA        │         └──────────┘
└─────────────────────┘
          │
          ▼
┌─────────────────────┐         ┌──────────┐
│  PROGRAMMING        │ ══════▷ │  MATLAB  │
│  EXPERIMENTS        │         └──────────┘
│  RESULTS            │
└─────────────────────┘
          │
          ▼
┌─────────────────────┐         ┌──────────┐
│  MODELING THE       │ ══════▷ │  ABAQUS  │
│  SPECIMEN WITH      │         └──────────┘
│  THE VALUES         │
│  OBTAINED           │
└─────────────────────┘
```
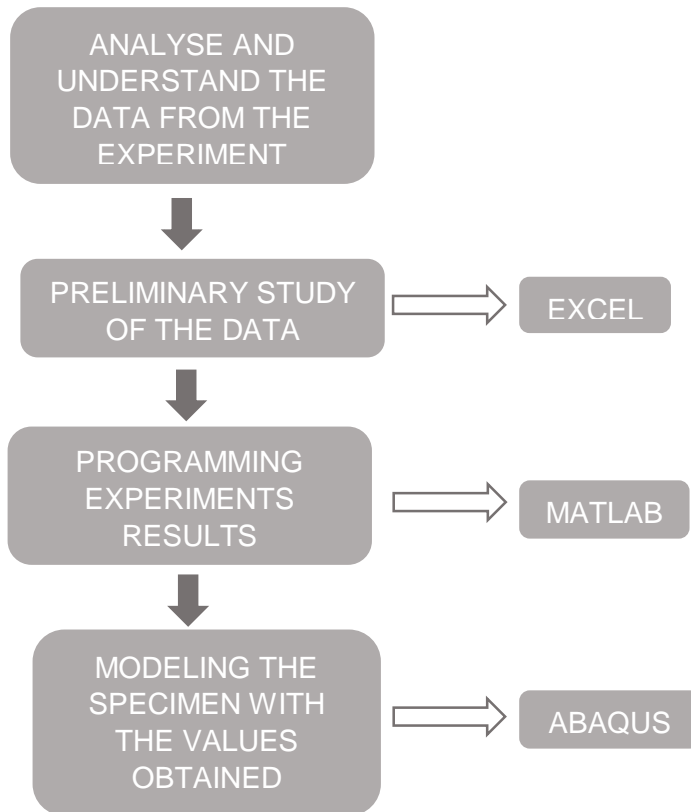
1.2. What is cast iron?

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.

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

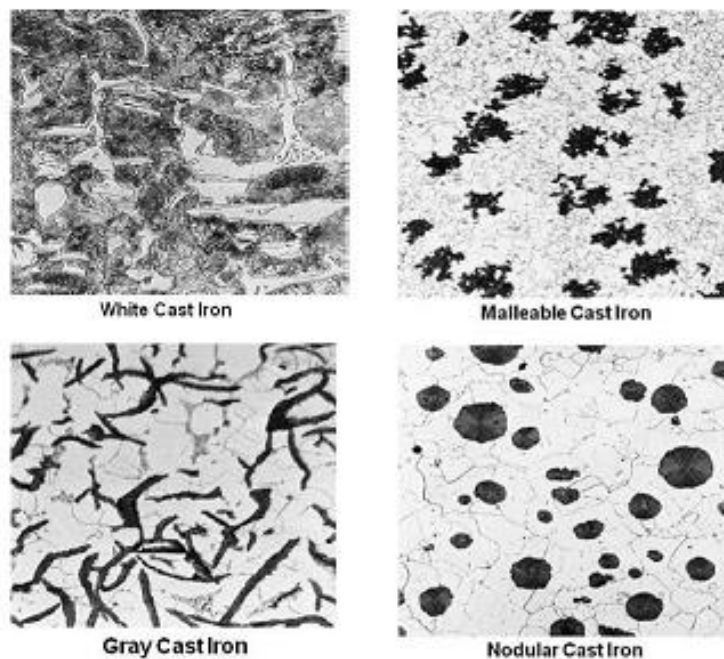Table1. Range of Compositions (values in percent (%)). (Engineers HandBook, 2004)

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 showed 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 that states:

$$\sigma = E*\varepsilon$$

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) and when the yield point is got, the material starts with the plastic deformation.

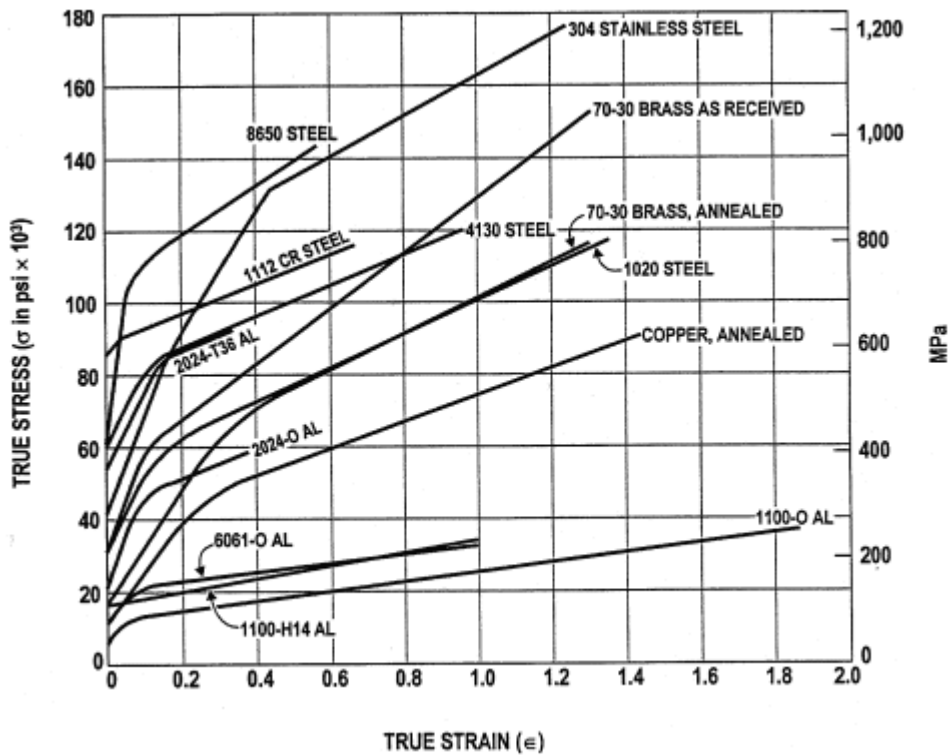[1] psi: pounds per square inch that it is an unit of pressure or stress based on avoirdupois units

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 but after that, as well. 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 than 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.

1.5. Programs

To get results and can 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 get the same experiment but in this case, with informatics tools.

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. 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.

3. Preliminary study of the data

The data files from the laboratory experiments were given, hence the first step was to analyse and study these values.

The experiments consist, in working of several specimens made by cast-iron at different temperatures.

The files can be divided in two groups: "Cyclic" and "Tension". The difference between them is in the first one you study the behaviour of the material in a cycle. When we started studying the files we noticed that the first values of the strain were negative which is a bit strange but it has an explanation: in experiments where you apply traction in a specimen it's pretty normal the values of the strain are negative at first due to the tensile machine testing, it needs few time to work properly.

After studying the files with the data, some files in Excel were done to have an overview about the results will have to get. It was not possible making a study in Excel with the "Cyclic" files because the Excel capacity is not enough.

Below, we can see the graphics received in Excel when I the values of stress and strain were selected. The graphics, as we can see below in Figure 4-8, are the well-known Stress-Strain Curves. One of the purpose of this project is achieve a value of Young Modulus (E) for this material so these graphics are a way to do a first approximation.
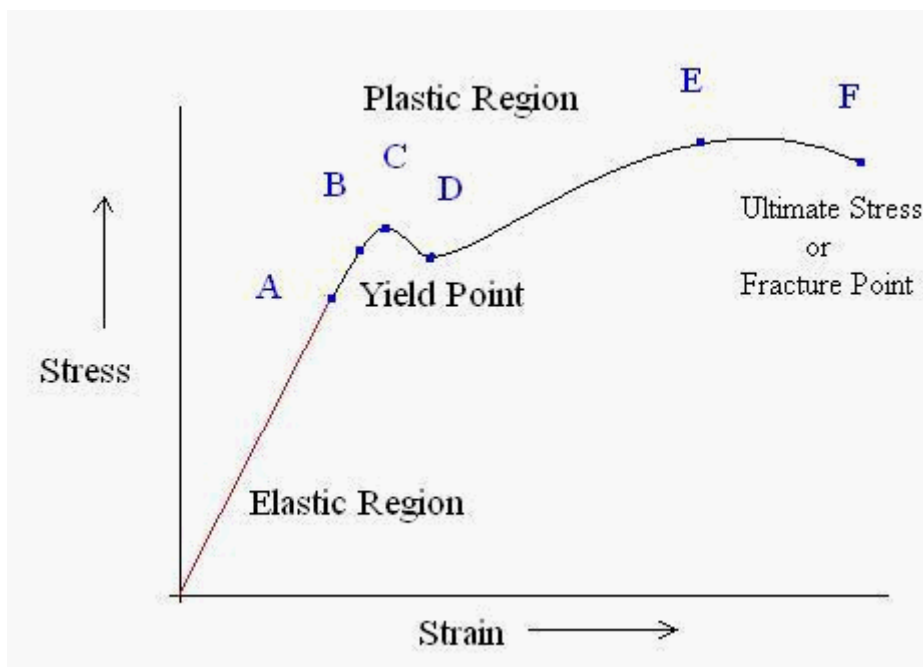

Figure 3. Typical Strain-Stress Curve (Total Materia, 2010)

This figure is the Strain-Stress Curve presumably to get with the data in Excel or at least, kind of the same curve.

And now you can see the curves from Excel for each temperature (25ºC, 100ºC, 400ºC, 450ºC, 500ºC).
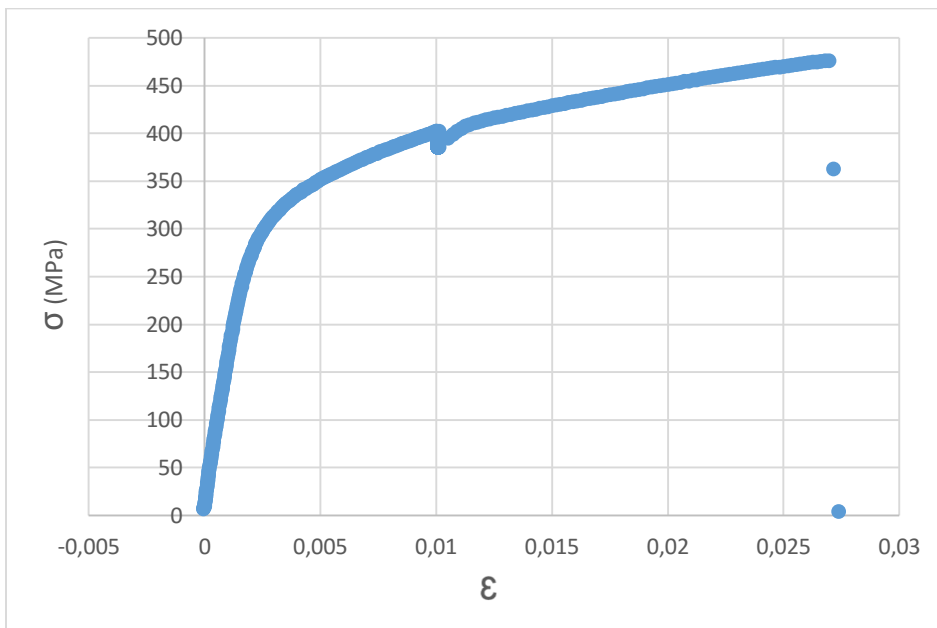
3.1. 25 ºC

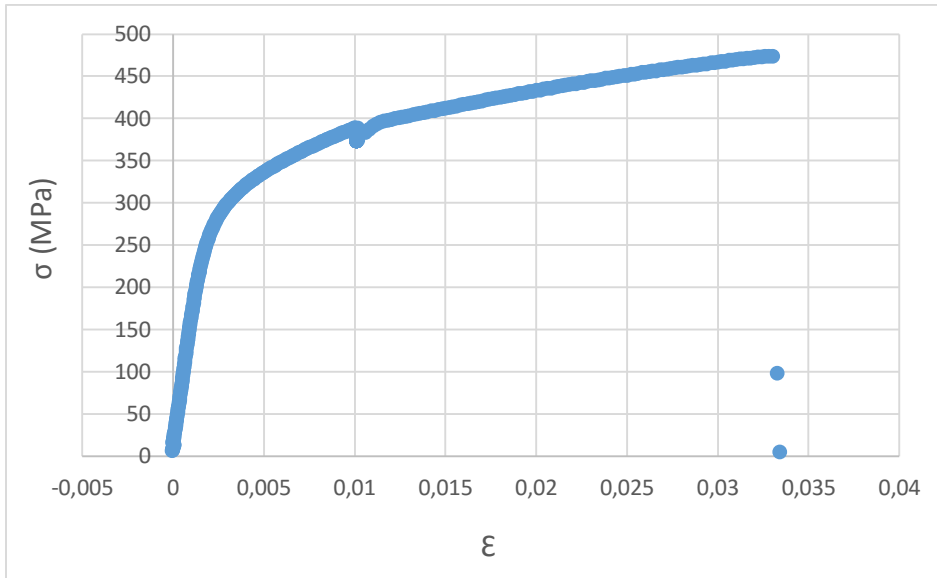Figure 4. Strain-Stress curve at 25ºC (Excel)

3.2. 100ºC

Figure 5. Strain-Stress curve at 100ºC (Excel)
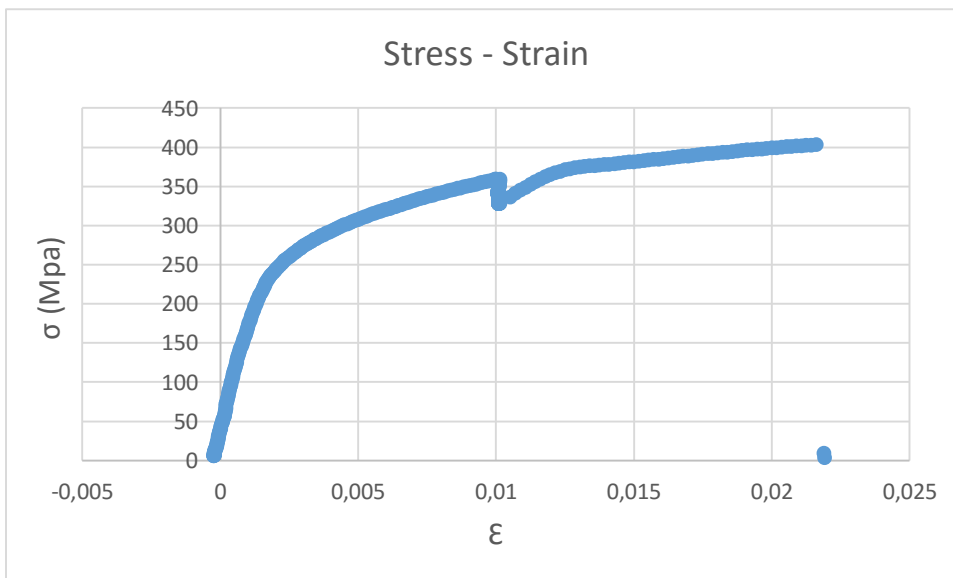
3.3. 400 ºC

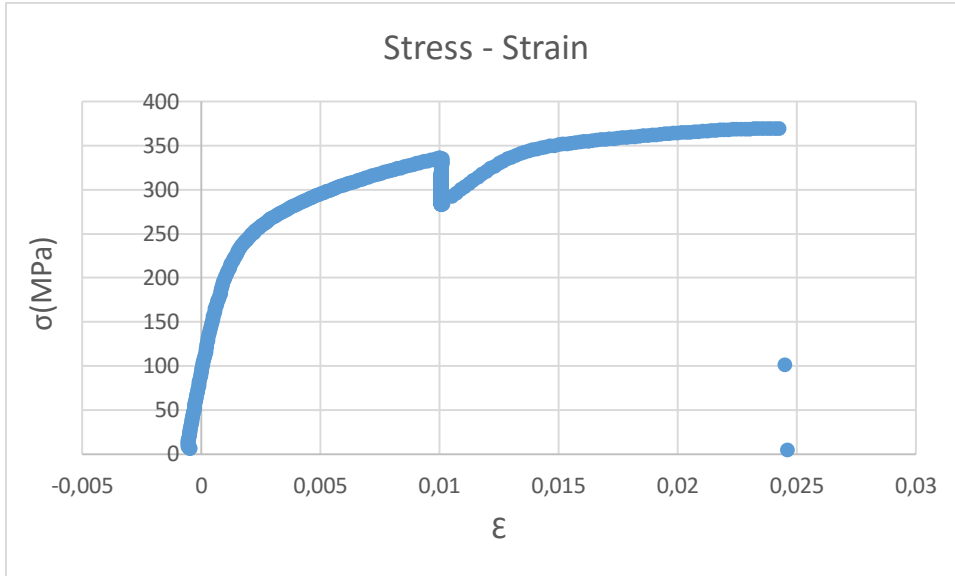Figure 6. Strain-Stress curve at 400ºC (Excel)

3.4. 450 ⁰C

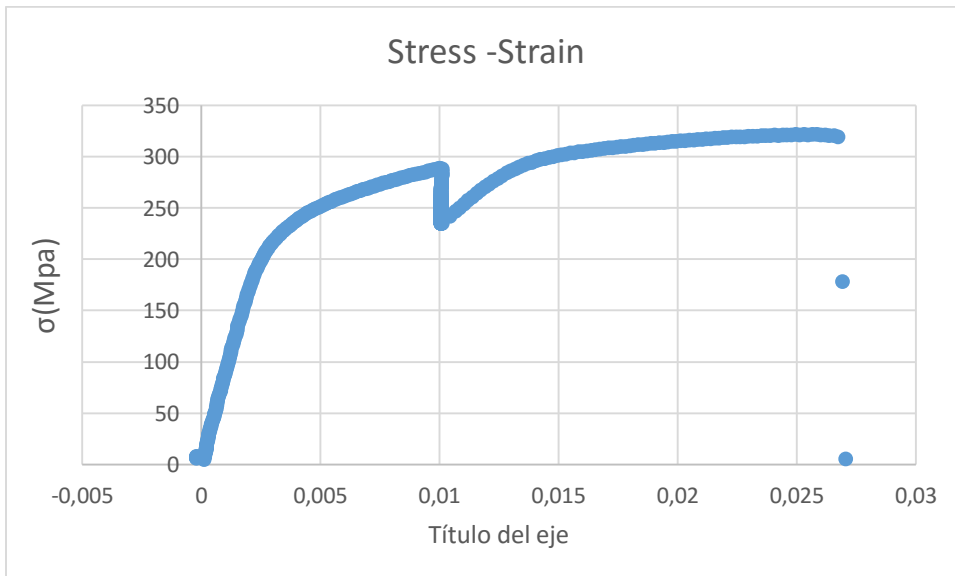Figure 7. Strain-Stress curve at 450ºC (Excel)

3.5. 500 ⁰C

Figure 8. Strain-Stress curve at 500ºC (Excel)

Once done the graphics in Excel it is evident that the curves obtained are like the real Stress-Strain Curves. We can see the range of the Stress and Strain values for each temperature is almost the same: the range for Stress is between 0-500 MPa and it is between 0-0.03 for the Stress. Likewise we can see some outliers in the graphics, whose meaning is there are some values out of range surely due to a bad performance of the tensile machine testing.

4. Programming in MATLAB

Once done a first overview about the data, it is time to study deeper how to obtain the exact values of the Young Modulus through the Strain and Stress.

The aim of this step is to get the Young Modulus value (E) for each temperature.

Briefly, the selected method to obtain the Young Modulus values has been:

1) Draw the Stress-Strain plot for each temperature.
2) Get the Young Modules values from the slopes of these graphics by hand.

You can use different tools to calculate these values, MATLAB has been our choice due to it is a powerful software. The reason why we have chosen this one is because of the amount of data collected so we have decided to work with a tool which let us save time and MATLAB is ideal because you can work with a lot of data.

The following procedure was implemented in order to use all the values of each file in MATLAB in order to keep the data in a powerful software which let us work faster with all the data to get the graphic Stress-Strain. The procedure has been programmed for Figure 9-17 and Table 2-11 all the files in MATLAB in order to be able to handle these values in this software to create the Stress-Strain graphics directly in MATLAB.

Following, we can see the charts for each temperature:

4.1. 25ºC

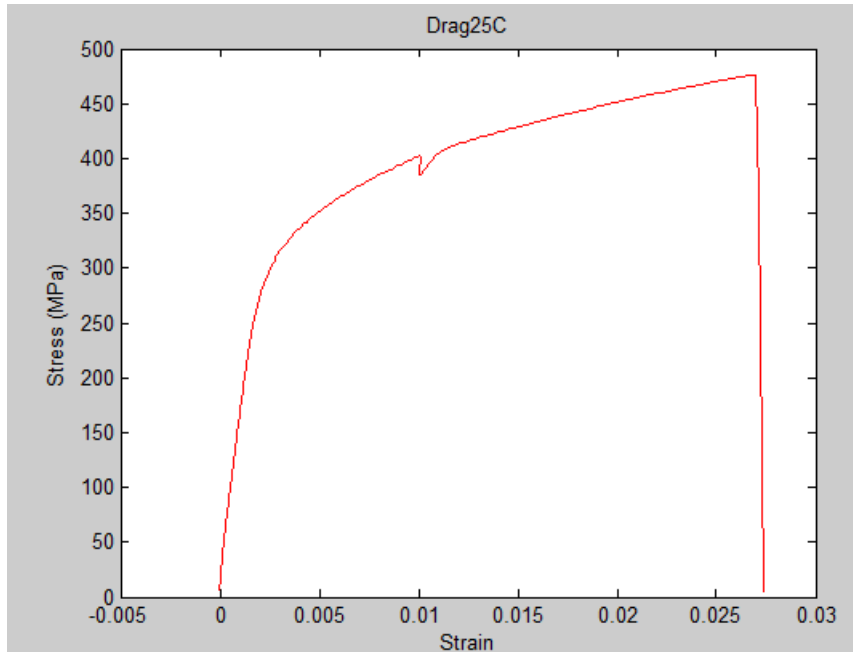For 25ºC the Stress-Strain Graphic is:

Figure 9. Strain-Stress curve at 25ºC (Matlab)

To get the Young Modulus (E) value, three values of strain and three values of stress were selected, and we have to operate to get only one Young Modulus with the following equation:

$$\sigma = \varepsilon * E$$

[2]

There are three possible values of Young Modulus so in order to get only one, an average of these three values was done.

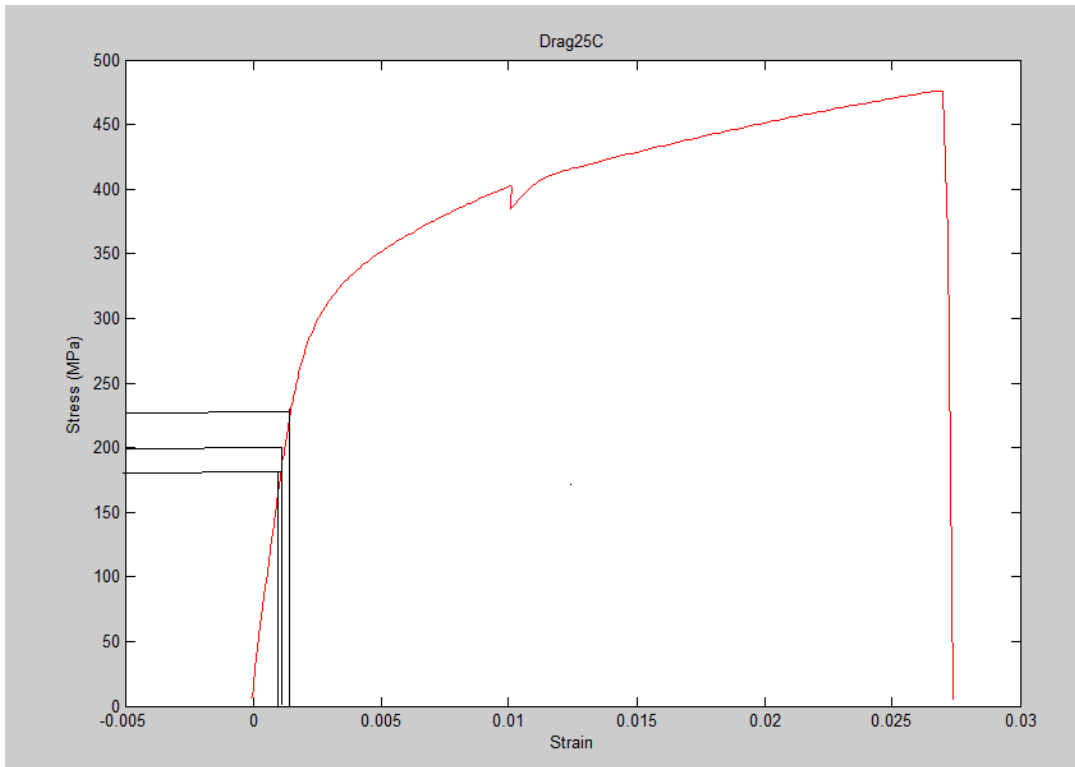[2] See the point 1.4 to have a more specified idea about the Hookes Law.

Figure 10. Strain-Stress curve at 25ºC with different values of strain and stress (Matlab)

Strain	Stress	Young Modulus
0,0015	240	160.000 MPa
0,0011	200	181.818,1818 MPa
0,001	185	185.000 MPa

Table 2. Young Modulus values at 25ºC

Doing the average between the three Young Modulus values in the chart, we get:

Young Modulus	175.606,0606 MPa

Table 3. Average of Young Modulus at 25ºC

4.2. 100 ºC

For 100 ºC the Stress-Strain Graphic is:

Figure 11. Strain-Stress curve at 100ºC with different values of strain and stress (Matlab)

Young Modulus is obtained in the same way done for the previous temperature.

Strain	Stress	Young Modulus
0,0012	165	137.500 MPa
0,0011	180	163.636,3636 MPa
0,001	200	200.000 MPa

Table 4. Young Modulus values at 100ºC

Doing the average between the three Young Modulus values in the chart, we get:

Young Modulus	167.045,4545 MPa

Table 5. Average of Young Modulus at 100ºC

4.3.   400 ⁰C

For 400 ⁰C the Stress-Strain Graphic is:

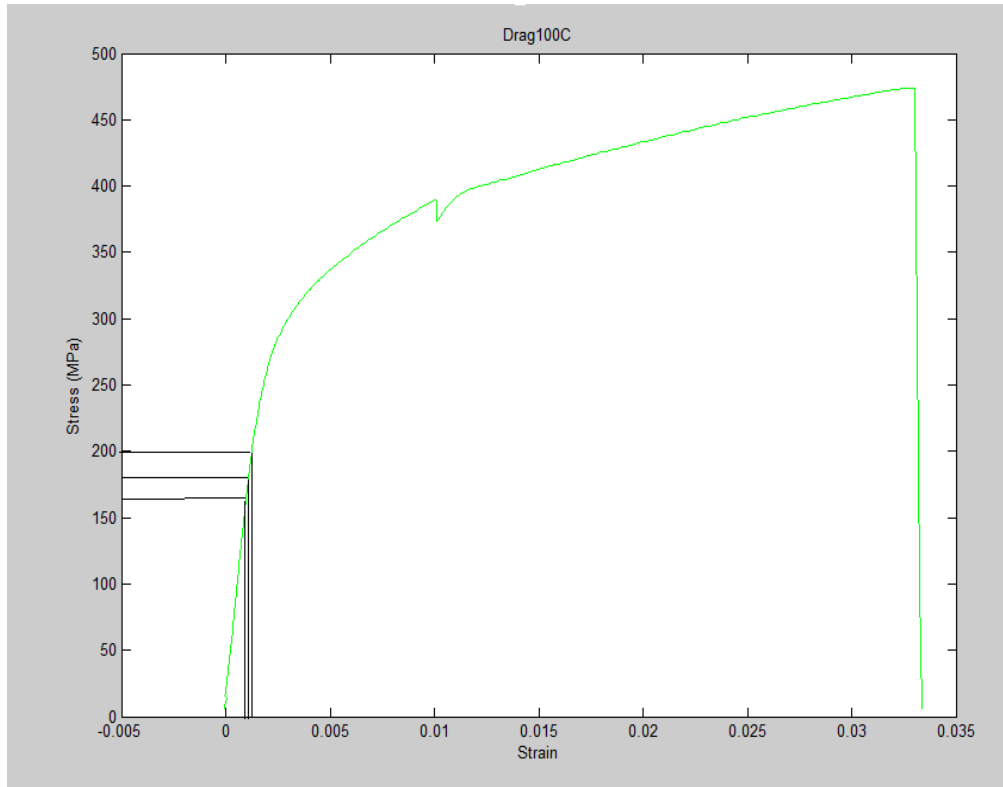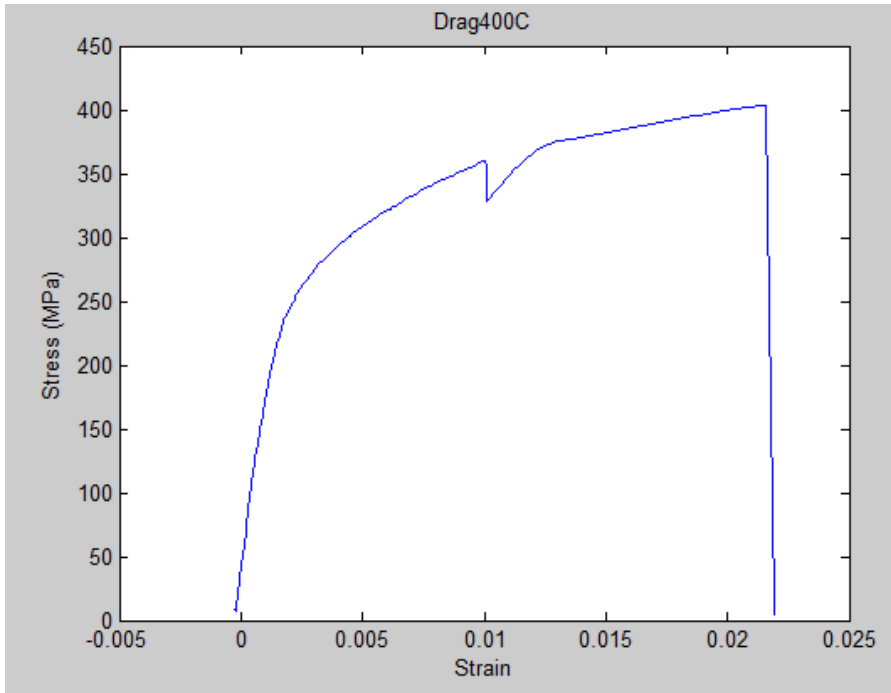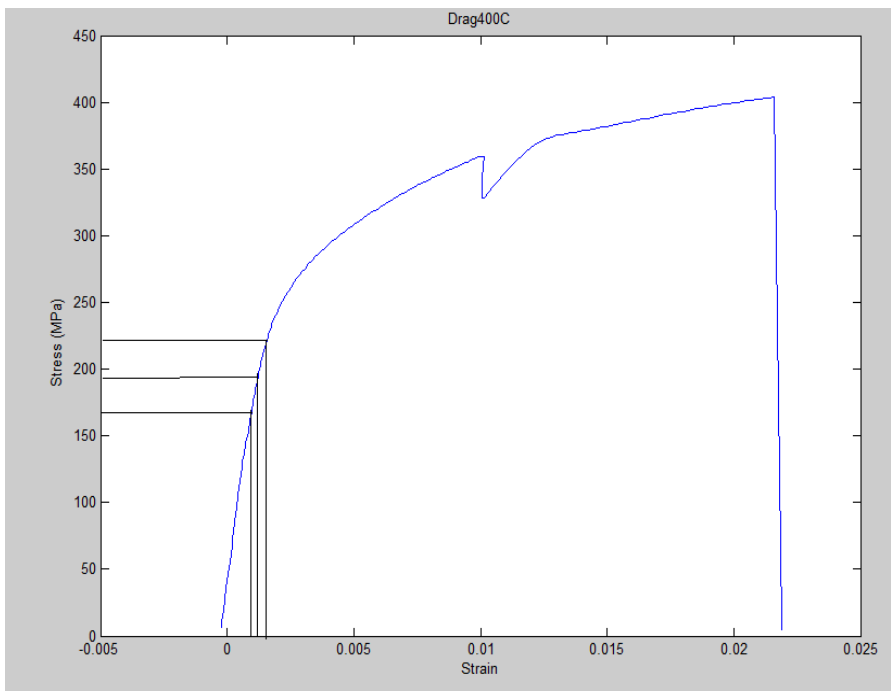Figure 12. Strain-Stress curve at 400⁰C (Matlab)

Figure 13. Strain-Stress curve at 400⁰C with different values of strain and stress (Matlab)

Young Modulus is obtained in the same way done for the previous temperature.

Strain	Stress	Young Modulus
0,002	225	112.500 MPa
0,0015	195	130.000 MPa
0,001	165	165.000 MPa

Table 6. Young Modulus values at 400ºC

Doing the average between the three Young Modulus values in the chart, we get:

Young Modulus	135.833,3333 MPa

Table 7. Average of Young Modulus at 400ºC

4.4. 450 ºC
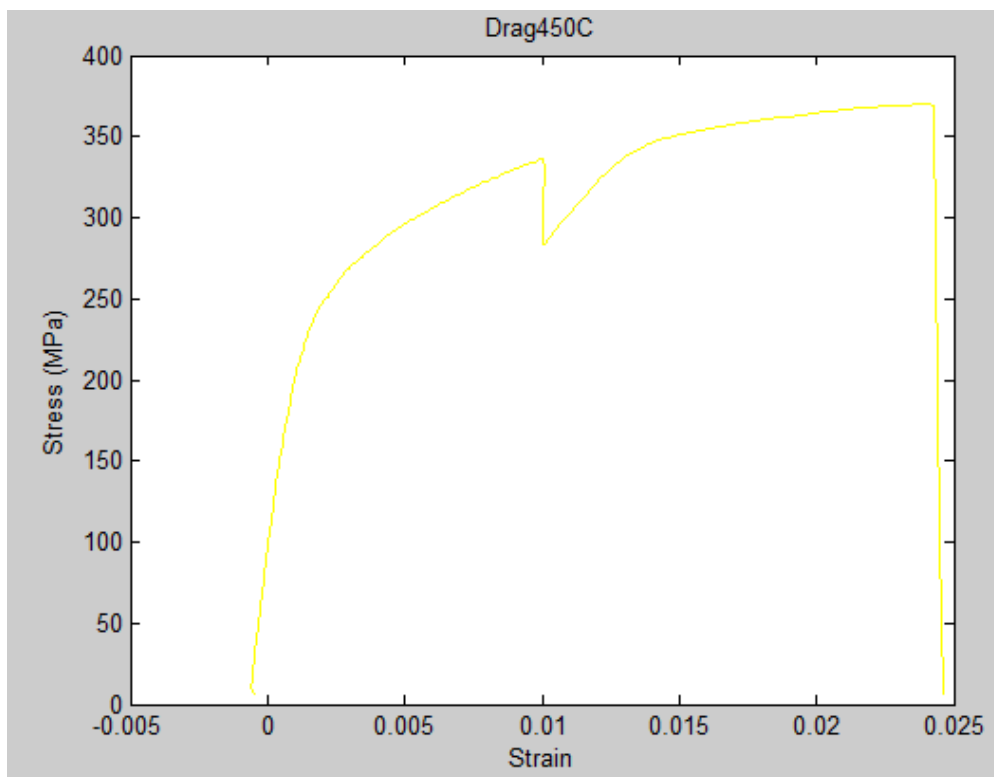
For 450 ºC, the Stress-Strain Graphic is:

Figure 14. Strain-Stress curve at 450ºC (Matlab)

Young Modulus is obtained in the same way done for the previous temperature.
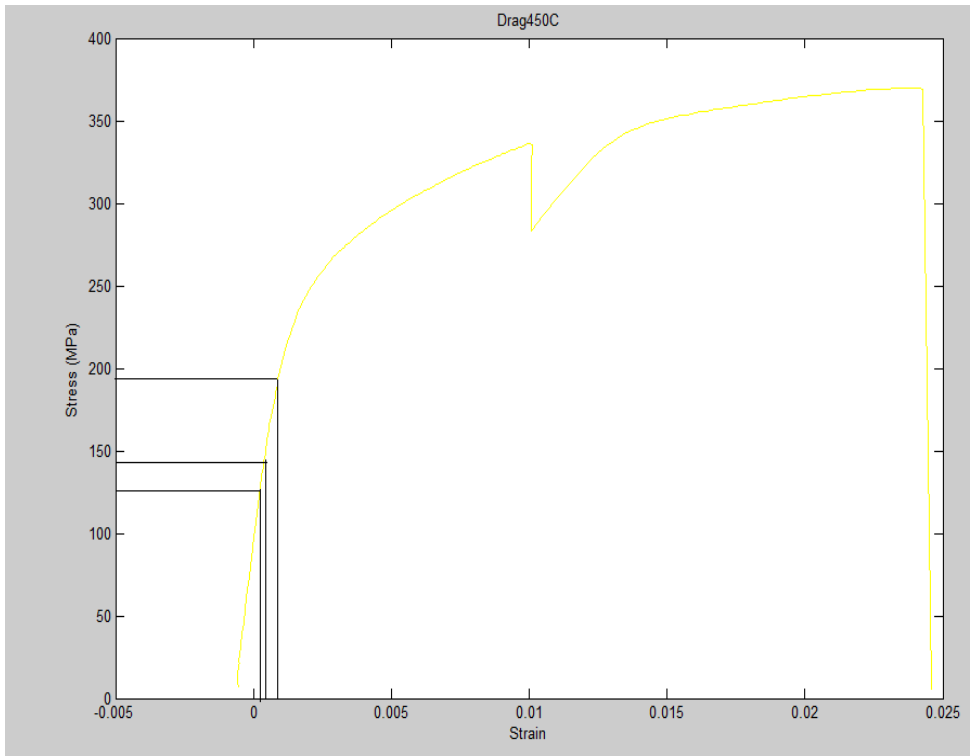
Figure 15. Strain-Stress curve at 450ºC with different values of strain and stress (Matlab)

Strain	Stress	Young Modulus
0,007	135	192.857,1429 MPa
0,0016	140	87.500 MPa
0,0019	190	100.000 MPa

Table 8. Young Modulus value at 450ºC

Doing the average between the three Young Modulus values in the chart, we get:

Young Modulus	126.785,7143 MPa

Table 9. Average of Young Modulus value at 450ºC

4.5.   500 ºC

For 500 ºC the Stress-Strain curve is:

Figure 16. Strain-Stress curve at 500ºC (Matlab)

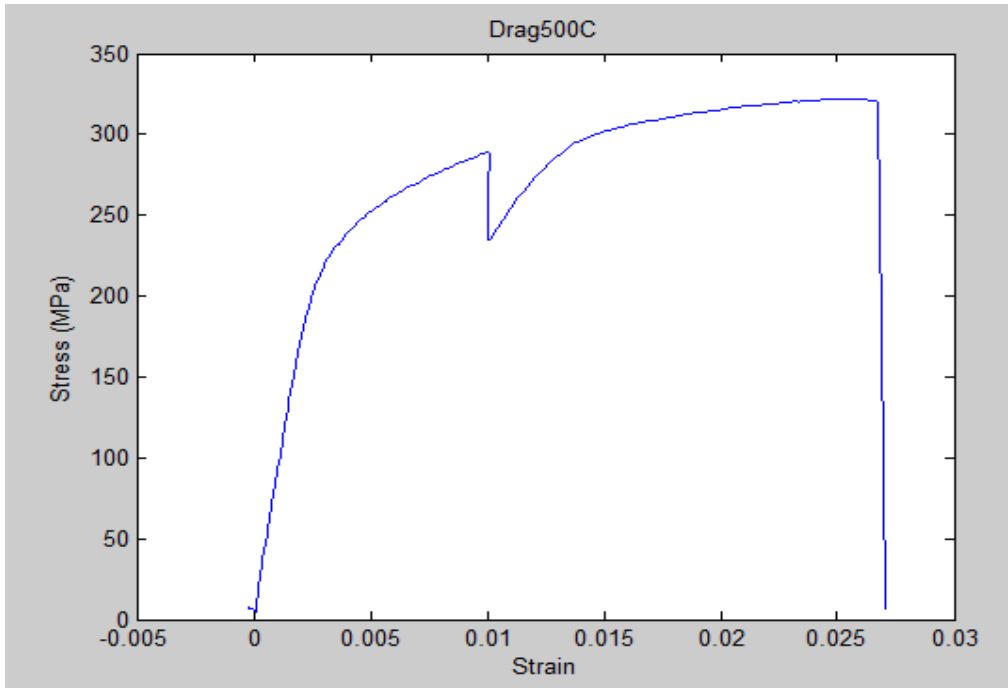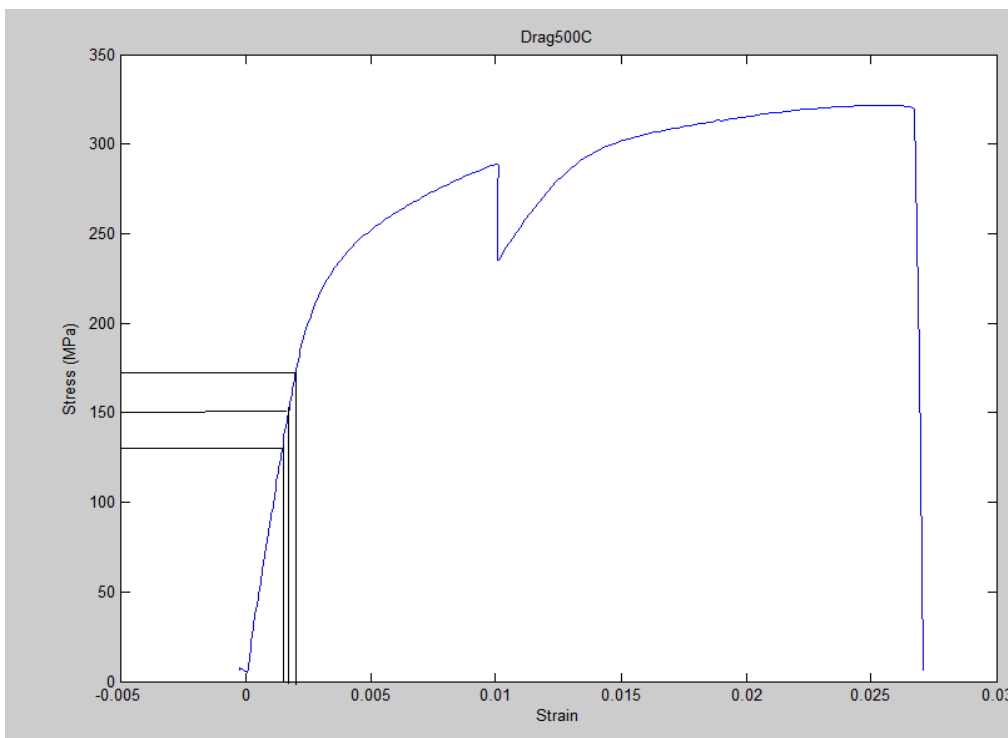

Figure 17. Strain-Stress curve at 500ºC with different values of strain and stress (Matlab)

Young Modulus is obtained in the same way done for the previous temperature.

Strain	Stress	Young Modulus
0,0015	170	113.333,3333 MPa
0,0016	150	93.750 MPa
0,0019	140	73.684,21053 MPa

Table 10. Young Modulus values at 500ºC

Doing the average between the three Young Modulus values in the chart, we get:

Young Modulus	113.333,3333 MPa

Table 11. Average of Young Modulus at 500ºC

Underneath we can see a chart for the Young Modulus values at each temperature. The best way to see how is the relation between the Young Modulus and the temperature is through a graphic.

Young Modulus (GPa)	Temperature (ºC)
175,606	25
167,045	100
135,833	400
126.785	450
113,333	500

Table 12. Summary of previous values
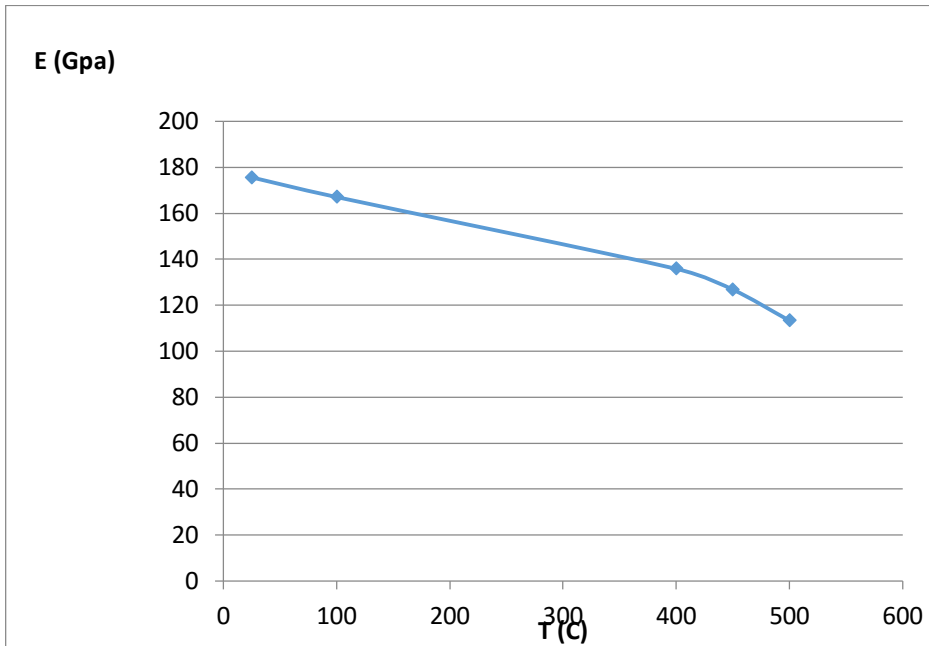
Figure 18. Temperature- Young Modulus curve

As we can see in the graphic above, the relation between the Young Modulus and the temperatures is pretty linear descendant, it means, at the same time the Young Modulus decreases, the temperature increases.

To get the equation of the Young Modulus-Temperature graphic, a linear approximation has been done to get it (the black line). The result is:

y = -0,1215x + 179,56
R² = 0,9794

Figure 19. Temperature-Young Modulus curve with the linear approximation

$$E\,(T) = -0{,}1215T + 179{,}56 \text{ MPa}$$
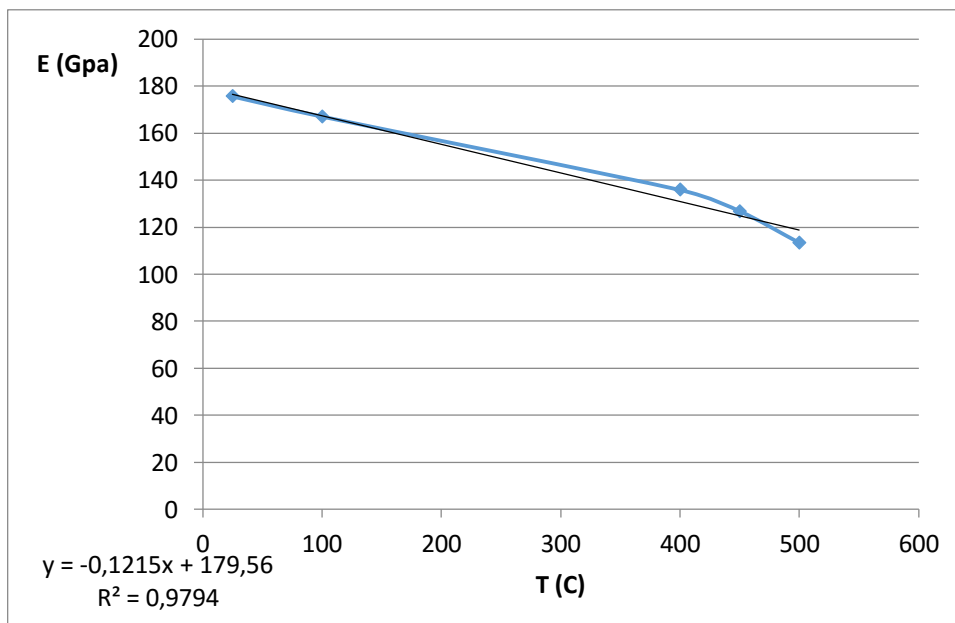
The R-value shown in the graphic says how close the equation is to the curve. The more close to one the R-value is, more perfect is the approximation hence our approximation is pretty good (0.9794).

4.6.   Load cycle

In the following graphics we can see the OP TMF cycle for the specimen at each temperature. It starts compressing but then there is a cycle of compression-traction. Beneath we can see the graphics about how the cast iron works at different temperatures (100ºC, 400ºC, 500ºC)
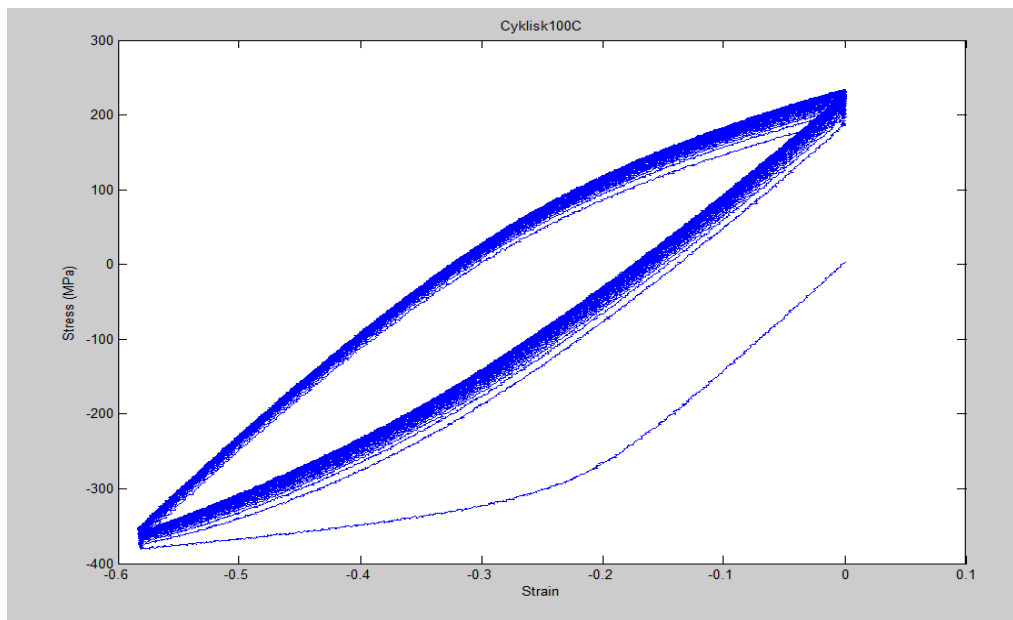
4.6.1. 100ºC

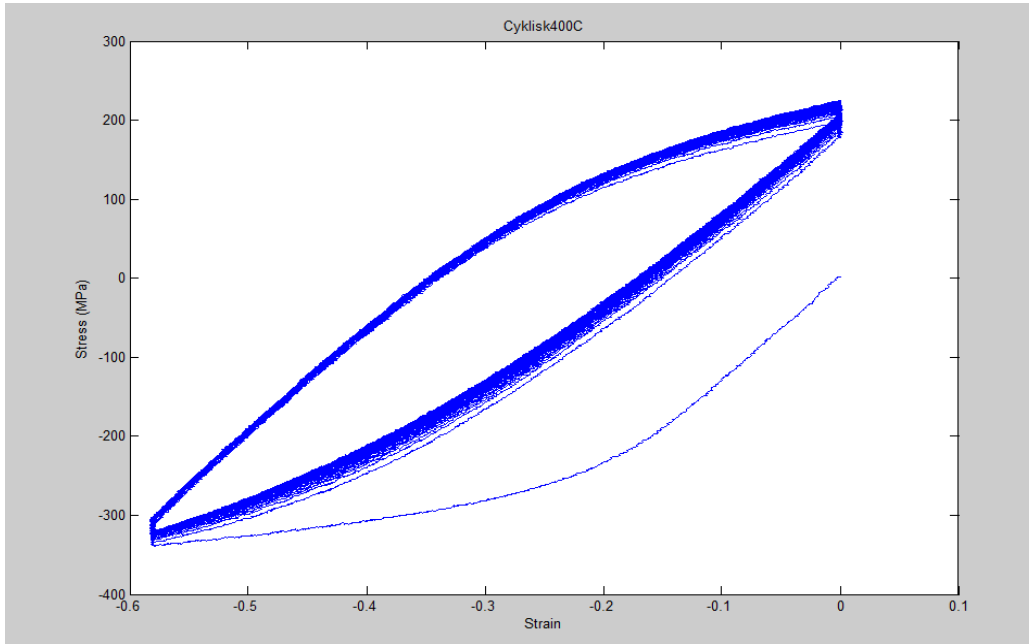Figure 20. Load cycle at 100ºC

4.6.2. 400ºC

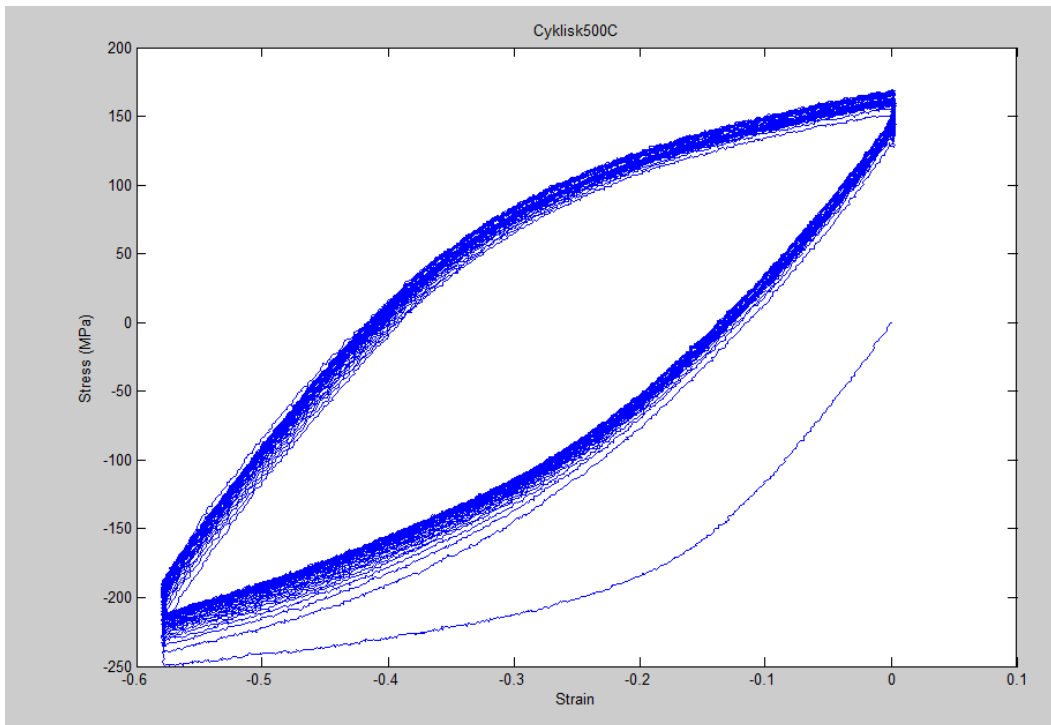Figure 21. Load cycle at 400ºC

4.6.3. 500ºC

Figure 22. Load cycle at 500ºC

What we can see with these graphics is the working cycle of the specimen at each temperature. The specimen starts compressing to later enter in a traction cycle until it breaks.

5. Modelling in ABAQUS

Modeling in ABAQUS that is another used tool which let us know how the specimen performs. If everything is correct, the results obtained in ABAQUS will be the same as in the real experiments. The established goal is getting graphics to compare them with the real values from the experiments.

The first step is to create the test specimen. Underneath we can see the measures of the specimen.
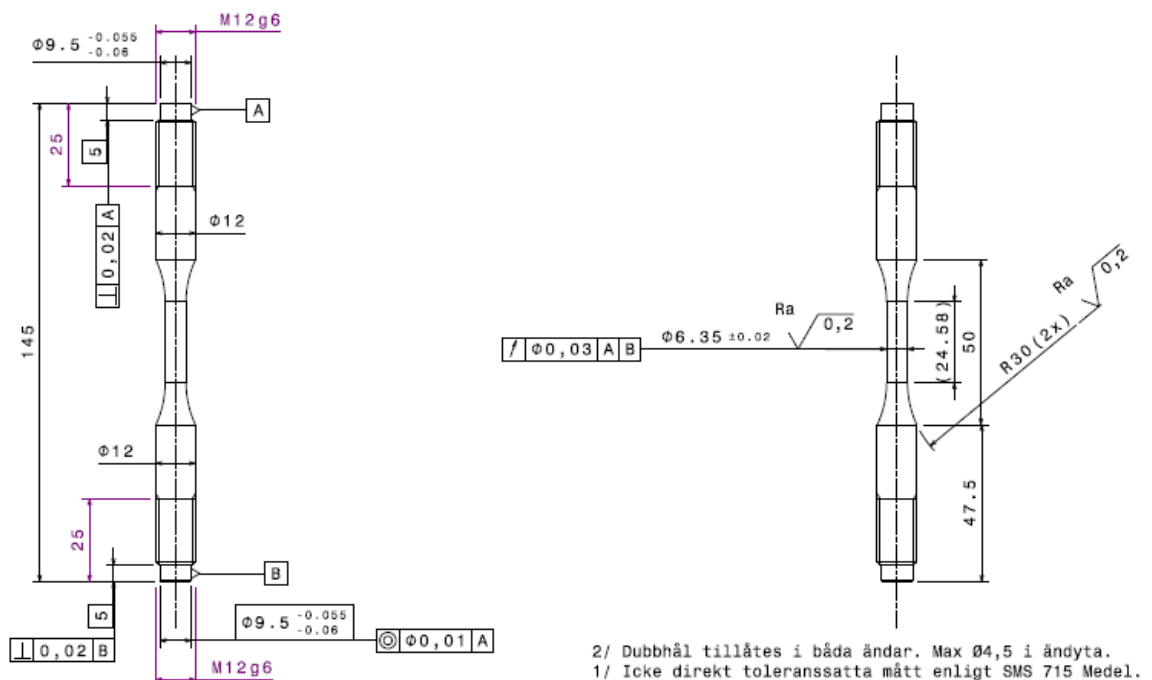

Figure 23. Specimen measures

The specimen in ABAQUS has been modelled like an Axisymmetric model. It could be modelled in a 3D way but it is more appropriated to design like the way which will be show below but first some explanations about the selected way of modelling.

- What does Axisymmetric mean?

 Exhibiting symmetry around an axis.

- Why have it been decided to model the specimen in this way?

If there is symmetry around an axis, it is always convenient to use an axisymmetric model. This is due to reduce a lot the needed time to solve the problem comparing with a 3D model and in general, the results that you get are much better. The desired section has to be modelled in the XY plane, being the X axis the radial axis and the Y axis, the symmetry axis of the model Figure 3. (Total Materia, 2010)

As said before, the first step is just to model the specimen in ABAQUS according to the option selected before. In the following chart, this is showed:
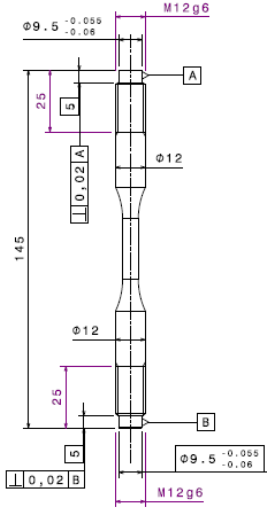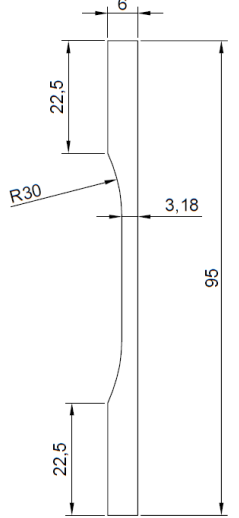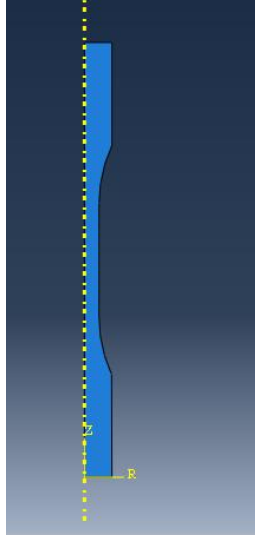
Initial specimen's geometry model [3i]	Specimen's geometry draw in ABAQUS	Final specimen showed in ABAQUS

Table 13. Comparative between the real geometry and the applied geometry in ABAQUS

If we take a look between both geometries, there is a big difference but this is due to the Axisymmetric model we have applied to model the specimen.

Once is created the specimen, it is time to implement the material properties for it.

---

[3] In this draw, the specimen's clamps are also showed but when it is time to model, you don't have to model them because it is just a way to hold the specimen while the experiment is working

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

8. Appendix

8.1. Implemented code in MATLAB

In the following pages, all the code implemented in MATLAB is shown. Five codes have been created in order to get the data for each temperature (25ºC-100ºC-400ºC-450ºC-500ºC). The files with the data are the results of the experiment done by other student.

25ºC

#Clear all the things written before

clear all

#Define the format of the numbers

format long

#Call to the field where the data is

name = 'drag_25C_v1.txt';

fileID = fopen (name);

#Define the number of columns we have in the file and the type of data that the numbers are

formatspec = '%f %f %f %f %f %f;

#Create a loop with the goal of reading all the data of the file

for i = 1:620

    value25 (i;:) = fscanf (fileID, formatspec [1,6]);

end

#Call to the variables we want to get specifying the number of column where they are

Strain25 = Value25 (:,4)

Stress25 = Value25 (:,5)

 #Close the file

fclose (fileID);

#Plotting a curve with the data read before

plot (Strain25, Stress25, '-r')

100ºC

#Clear all the things written before

clear all

#Define the format of the numbers

format long

#Call to the field where the data is

name = 'drag_100C_v1.txt';

fileID = fopen (name);

#Define the number of columns we have in the file and the type of data that the numbers are

formatspec = '%f %f %f %f %f %f;

#Create a loop with the goal of reading all the data of the file

for i = 1:584

    value100 (i;:) = fscanf (fileID, formatspec [1,6]);

end

#Call to the variables we want to get specifying the number of column where they are

Strain100 = Value100 (:,4)

Stress100 = Value100 (:,5)

#Close the file

fclose (fileID);

#Plotting a curve with the data read before

plot (Strain100, Stress100, '-r')

400ºC

#Clear all the things written before

clear all

#Define the format of the numbers

format long

#Call to the field where the data is

name = 'drag_400C_v1.txt';

```
fileID = fopen (name);
```

#Define the number of columns we have in the file and the type of data that the numbers are

```
formatspec = '%f %f %f %f %f %f;
```

#Create a loop with the goal of reading all the data of the file

```
for i = 1:549

    value400 (i;: ) = fscanf (fileID, formatspec [1,6]);

end
```

#Call to the variables we want to get specifying the number of column where they are

```
Strain400 = Value400 (:,4)

Stress400 = Value400 (:,5)
```

#Close the file

```
fclose (fileID);
```

#Plotting a curve with the data read before

```
plot (Strain400, Stress400, '-r')
```

450ºC

#Clear all the things written before

```
clear all
```

#Define the format of the numbers

```
format long
```

#Call to the field where the data is

```
name = 'drag_450C_v1.txt';

fileID = fopen (name);
```

#Define the number of columns we have in the file and the type of data that the numbers are

```
formatspec = '%f %f %f %f %f %f;
```

#Create a loop with the goal of reading all the data of the file

```
for i = 1:683

    value450 (i;: ) = fscanf (fileID, formatspec [1,6]);
```

end

#Call to the variables we want to get specifying the number of column where they are

Strain450 = Value450 (:,4)

Stress450 = Value450 (:,5)

#Close the file

fclose (fileID);

#Plotting a curve with the data read before

plot (Strain450, Stress450, '-r')

500ºC

#Clear all the things written before

clear all

#Define the format of the numbers

format long

#Call to the field where the data is

name = 'drag_500C_v1.txt';

fileID = fopen (name);

#Define the number of columns we have in the file and the type of data that the numbers are

formatspec = '%f %f %f %f %f %f;

#Create a loop with the goal of reading all the data of the file

for i = 1:533

    value500 (i;:) = fscanf (fileID, formatspec [1,6]);

end

#Call to the variables we want to get specifying the number of column where they are

Strain500 = Value500 (:,4)

Stress500 = Value500 (:,5)

#Close the file

fclose (fileID);

#Plotting a curve with the data read before

plot (Strain500, Stress500, '-r')