

# Analysis of Tensile Test of Mild Steel using Finite Element Method

Edgar Jose

*Department of Mechanical Engineering,  
Federal Institute of Science & Technology,  
Kochi, India*

Tom Anto

*Department of Mechanical Engineering,  
Federal Institute of Science & Technology,  
Kochi, India*

**Abstract**—Mechanical testing shows an important role in finding the fundamental properties of engineering materials as well as in generating new materials and in managing the quality of materials for use in design and construction. If a material is used as a part of an engineering structure, it is important to know that the material is strong and rigid enough to overcome the loads that it will experience in service. As a result, engineers have developed a number of experimental techniques for mechanical testing of engineering materials. Tension, compression, bending or torsion loading is the techniques used for mechanical testing. The most common type of test used to measure the mechanical properties of a material is the Tension Test. Tension test is widely used to provide basic information on the strength of materials and is an acceptance test for the specification of materials. The major parameters that describe the stress-strain curve obtained during the tension test are the tensile strength (UTS), yield strength or yield point ( $\sigma_y$ ), elastic modulus (E), percent elongation ( $\Delta L\%$ ) and the reduction in area (RA%). Toughness, Resilience, Poisson's ratio ( $\nu$ ) can also be found by the use of this testing technique. In this paper, it is decided to do a tension test of Mild Steel using Abaqus CAE software.

**Keywords:** Finite Element Method, Mild Steel, Tensile Test

## I. INTRODUCTION

The tensile test is an engineering procedure used to characterize elastic and plastic deformations related to the mechanical behaviour of materials. Due to the non-uniform stress and strain existing at the neck of axial deformation, it has been recognized that significant changes in the geometric configuration of the specimen must be considered in order to describe the material response during the deformation process up to the fracture stage.

Although in the engineering applications, the design of structural parts is restricted to the elastic deformation of the materials involved, the knowledge of their behaviour beyond the elastic limit is significant since plastic effects with usually large deformations take place in the previous manufacturing procedures such as metal forming (traditionally classified into

- a. bulk-forming operations: forging, drawing, extrusion and rolling
- b. sheet forming operations: deep drawing, magnetic-pulse forming, tube

Crashworthiness, impact problems, inelastic buckling of thin-walled structures, superplastic forming, etc are some of the other important applications of elastoplastic models for metals [5].

The tension test is one of the most commonly used tests for evaluating materials. In its simplest form, the tension test is carried out by holding opposite ends of a test item within the load frame of a test machine. A tensile force is applied by the machine, which results in the gradual elongation and eventual fracture of the test item. During this process, force-extension data are monitored and recorded. When properly conducted, the tension test provides force extension data that can quantitate several important mechanical properties of a material. These mechanical properties determined from tension tests are as follows:

- Elastic deformation properties, such as the modulus of elasticity (Young's modulus) and Poisson's ratio
- Yield strength and ultimate tensile strength
- Ductility properties, such as elongation and reduction in area

- Strain-hardening characteristics

These material characteristics from this test are used for quality control in production, for ranking performance of structural materials, for evaluation of newly developed alloys, and for dealing with the static strength requirements of design.

The basic principle of the tension test is very simple, but numerous variables affect results. Usual sources of changes or variation in mechanical-test results include several factors involving materials, namely, methodology, human factors, equipment, and ambient conditions.

In this study, a simulation of a tensile test, which is a representative material test, is performed using a computer program (Abaqus CAE). Tensile test using mild steel is performed and then analysed the stress–strain relationship using a computer program (Abaqus/CAE).

## II. AN OVERVIEW TO ABAQUS CAE

ABAQUS is a set of finite element analysis modules. The cores of ABAQUS are the analysis modules, ABAQUS/Standard and ABAQUS/Explicit, which are complementary and integrated analysis tools.

- ABAQUS/Standard is a general-purpose, finite element module.
- ABAQUS/Explicit is an explicit dynamics finite element module.
- ABAQUS/CAE incorporates the analysis modules into a Complete ABAQUS Environment for modeling, managing, and monitoring ABAQUS analysis and visualizing results.

The finite element program used in the FEA room is ABAQUS/CAE, which is an intuitive and consistent user interface throughout the system.

### A Steps to launch Abaqus

- 1) Click Start - All programs - ABAQUS 6.13-1 - ABAQUS CAE Command in Windows System. An ABAQUS Command window appears in DOS environment.
- 2) Use general commands in DOS system to move to the directory on the hard disk. For example, if a file is created in catalogue C:\Temp named ABAQUS WORK and wanted all the ABAQUS results be saved in this file, type command CD C:\Temp\ABAQUS WORK.
- 3) Run command ABAQUS CAE to enter ABAQUS/CAE user interface

### B Starting a job in Abaqus

After entering main interface of ABAQUS/CAE, there are several paths to start a job.

- Create Model Database to begin a new analysis.
- Open Database to open a previously saved model or output database file.
- Run Script to run a file containing ABAQUS/CAE commands.
- Start Tutorial helps to begin an introductory tutorial from the online documentation

### C Exiting Abaqus

When some work is done in the middle of an ABAQUS job and want to exit, save your finished work as a Model Database file (\*.cae).

- Click File – Save and input a file name. Normally in this step the file filter is default to be Model Database (\*.cae).
- Click File – Exit to exit the main interface of ABAQUS/CAE.
- Next time you run ABAQUS, open the saved file and continue with the job.

## III. EXPERIMENTATION DETAILS

The model to be created and its dimensions according to the ASTM E8 standards are given below as shown in figure 3.1. The table 3.1 shows the dimensions to be taken while modelling the sample. The dimensions include Gauge Length (G), Width (W), Thickness (T), Radius of Fillet (R), Overall Length (L), Length of Reduced Section (A), Length of Grip Section (B), and Width of Grip Section (C).

The procedure demonstrates how to generate components, material properties, boundary conditions and forces to create a Finite Element model using Abaqus CAE software.

1. Create the Part Model as per the Dimensions and Shape given above. In the Part Manager dialog box, click on Create. In the Create Part dialog box, give the name of the sample as Mild Steel. Select 3D, Deformable, Solid, Extrusion, and Approximate size as 500. Click Continue. Sketch the Part.
2. Give the properties of the Mild Steel material in the Material Manager option. The properties that have to be given are density, young's modulus, poisson's ratio, yield stress and plastic strain as shown in table 3.2.
3. In the Assembly Module, select the Create Instance tool. In the dialog box, select the Mild Steel, click the dependent option and then click OK.
4. In the Step Module, different steps have been created. Step Manager, Field Output Manager and History Output Manager are provided.
5. In the Load Module, one end of the sample is made fixed and the axial load is provided in the other end.
6. Proceed to Mesh Module. Click on the Seed Part icon. Accept the defaults in the Global Seeds dialogue box. Click done. Click on the Mesh Part Instance icon, at the bottom of the screen select yes.
7. In the Job Module, click on 'create job' icon. Name the job as Tensile Test. Click Continue.
8. Click on Job manager. Click 'Submit' to submit the job to the solver. Click Monitor to check the analysis progress.
9. Once the job is complete, click results to check the results. Select the icon to display contours. To examine different stresses etc, select Result, Field Output from the main menu bar.

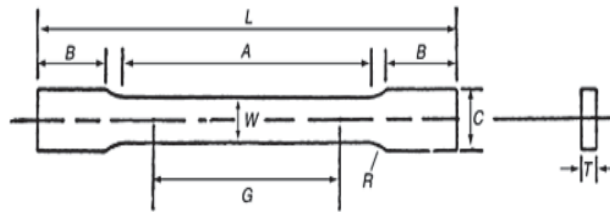


Fig 3.1 Sample to be created for the Tensile Test

TABLE I DIMENSIONS OF THE SAMPLE TO BE CREATED

Sl. No.	Description	Dimensions( mm)
1	Gauge Length (G)	203
2	Width (W)	35
3	Thickness (T)	10
4	Radius of Fillet (R)	25
5	Overall Length (L)	457
6	Length of Reduced Section (A)	229
7	Length of Grip Section (B)	76
8	Width of Grip Section (C)	51

TABLE II MECHANICAL PROPERTIES OF MILD STEEL

Density	7860 kg/m <sup>3</sup>
Young's Modulus	211 GPa
Poisson's Ratio	0.28
Yield Stress	247 MPa
Plastic Strain	0.02

#### IV. RESULTS AND DISCUSSION

The figure 4.1 shows the initial condition of the sample at the time of the application of the load. The Fig 4.2 shows the final condition of the sample, ie the sample is broken into two pieces and the Fig 4.3 shows the values of von mises stress that has obtained during the test.

By conducting the tensile test, we obtained the stress-strain graph. The plotted graphs are shown in Fig 4.4. Elastic deformation occurs in the initial portion of a stress-strain curve, where the stress-strain relationship is initially linear as shown in fig 6.3. In this region, the stress is proportional to strain. Mechanical behaviour in this region of stress-strain curve is defined by a basic physical property called the modulus of elasticity (abbreviated as  $E$ ). The modulus of elasticity is the slope of the stress-strain line in this linear region, and it is a basic physical property of all materials. It essentially represents the spring constant of a material. The modulus of elasticity is also called Hooke's modulus or Young's modulus.

The proportional limit (PL) is a point in the elastic region where the linear relationship between stress and strain begins to break down. At some point in the stress-strain curve, linearity ceases, and small increase in stress causes a proportionally larger increase in strain. This point is referred to as the proportional limit (PL) because up to this point, the stress and strain are proportional. If an applied force below the PL point is removed, the trace of the stress and strain points returns along the original line. If the force is reapplied, the trace of the stress and strain points increases along the original line.

The elastic limit (EL) is a very important property when performing a tension test. If the applied stresses are below the elastic limit, then the test can be stopped, the test piece unloaded, and the test restarted without damaging the test piece or adversely affecting the test results. For example, if it is observed that the extensometer is not recording, the force-elongation curve shows an increasing force, but no elongation. If the force has not exceeded the elastic limit, the test piece can be unloaded, adjustments made, and the test restarted without affecting the results of the test. However, if the test piece has been stressed above the EL, plastic deformation will have occurred, and there will be a permanent change in the stress-strain behaviour of the test piece in subsequent tension tests.

In the fig 4.4, the stress-strain curve is linear in nature in the beginning till  $0.17 \times 10^9$  stress value and 0.02 strain values. After that the curve began to lose its linearity. Then the curve reaches the proportional limit and later the elastic limit. When the sample is again loaded, it breaks.

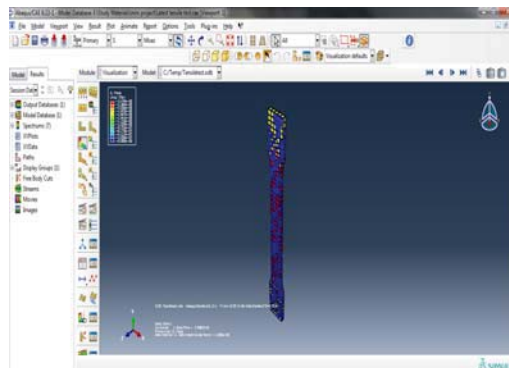


Fig 4.1 Initial Condition of the Sample

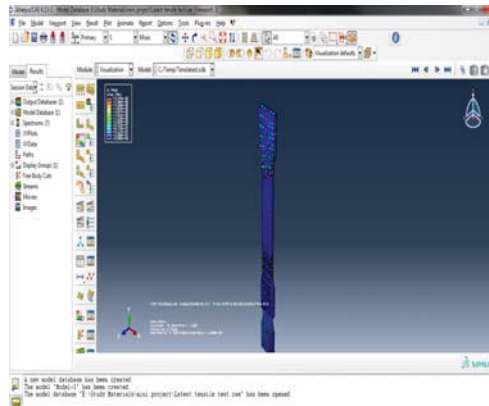


Fig 4.2 Final Condition of the Sample

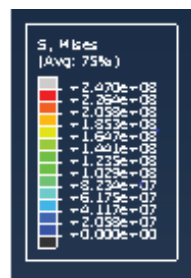


Fig 4.3 Values of on Mises Stress

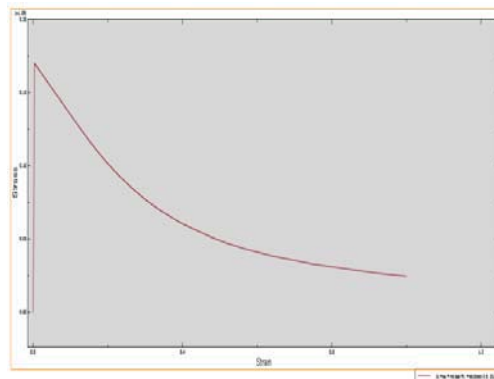


Fig 4.4 Stress-Strain Graph

## V.CONCLUSIONS

The Tensile Test Analysis of Mild Steel using finite element method has been conducted using a computer program called Abaqus/CAE. The results have been obtained. From the stress-strain graph it is understood that the mild steel follows the Hooke's Law i.e., stress is directly proportional to strain. After the linear region in the graph, there occurs necking on the sample and finally it breaks.

## REFERENCES

- [1] O.O. Joseph, F.I. Alo , "An Assessment of the Microstructure and Mechanical Properties of 0.26% Low Carbon Steel under Different Cooling Media: Analysis by one-way ANOVA", ISSN 2224-6096 (Paper) ISSN 2225-0581, Vol.4, No.7, 2014.
- [2] Eduardo E. Cabezas, Diego J. Celentano, "Experimental and numerical analysis of the tensile test using sheet specimens", Elsevier Science Publishers, *Finite Elements in Analysis and Design*, Vol 40 (2004) p.555–575
- [3] M. Prakash, S. Shekhar, A.P. Moon, K. Mondal, "Effect of machining configuration on the corrosion of mild steel", Elsevier Science Publishers, *J. of Materials Processing Technology*, Vol. 219 (2015), p.70–83.
- [4] George E. Dieter, "Mechanical Behavior Under Tensile and Compressive Loads", *Mechanical Metallurgy*, 3rd ed., McGraw-Hill, New York, 1986, p 275–295
- [5] D. Lewis, "Tensile Testing of Ceramics and Ceramic-Matrix Composites", *Tensile Testing*, P. Han, Ed, ASM International, 1992, p 147–182