Computational modeling of the small punch test using FEM

https://doi.org/10.56238/interdiinovationscrese-066

Wilfredo Javier Valdivia Rojas
MsC., Cesar Vallejo University – UCV - Huaraz, Perú
E-mail: wvaldiviaro@ucvvirtual.edu.pe

Heber Elcano Chanducas Tantaleán
MsC., Cesar Vallejo University – UCV - Huaraz, Perú
E-mail: hchanducasta@ucvvirtual.edu.pe

Luis Alberto Rabanal Ramirez
Dr., Fluminense Federal University – ICT-UFF- Rio das Ostras / RJ, Brasil.
E-mail: lramirez@id.uff.br

Sandro Pereira Vilela
Dr.
E-mail: sandro.vilela@gmail.com

Cláudio Márcio de Freitas da Silva
Doctoral, Technological Institute of Aeronautics – ITA - São José dos Campos/SP - Brasil.
E-mail: claudiocmfs@ita.br

Yovana Torres Gonzales
Dra., Continental University, Huancayo - Perú
E-mail: ytorresg@continental.edu.pe

Manuel Emiliano Esteves Pairazamán
Dr., National University Toribio Rodríguez de Mendoza – UNTRM - Amazonas, Perú
E-mail: manuel.esteves@untrm.edu.pe

Jesús Johan Huaney Carranza
Mag., Cesar Vallejo University – UCV - Huaraz, Perú
E-mail: jehuaneyca@ucvvirtual.edu.pe

Javier Alejandro Manrique Catalán
Mag., Technological University of Peru – UTP, Perú
E-mail: c24432@utp.edu.pe

ABSTRACT
This work uses the material characterization technique known in the bibliography as the "Small Punch Test" (SPT), considered non-destructive and low-cost, due to the fact that, for its application, small samples of material are used, compared with others. sizes of samples used in other trials, both the technique and the computational model are presented together due to their interrelationship. The simulation results are highly correlated with the published ones. After the point and calibration of the model, optimized results of the characterization of the API 5L-X70 steel used in gas pipelines are presented, the main objective of this work.

Keywords: Small Punch Test, Finite elements, Characterization of metals.

1 INTRODUCTION

The pipes used in the transport of oil through the different regions of Peru are subject to operating conditions that introduce temporary degradation phenomena, with large variations in temperature, humidity, corrosive environments, high voltage states, etc. Therefore, giving an opinion on the state of the useful life of these and critical parts (such as weld areas) is not an easy task, since there are no direct parameters that characterize the level of degradation. Sometimes, due to lack of information, these ducts are replaced prematurely because of the risk they represent.

Material characterization techniques are classified into destructive and non-destructive testing. Usually, in destructive testing, dimensionally large samples of material are used, which after the test are unusable. In the case of these samples, there is difficulty in performing the mechanical characterization of small regions, implying the waste of the material.

On the other hand, there are non-destructive tests, these tests are carried out on an object without causing any type of alteration, implying imperceptible or no damage. Examples of non-destructive testing are: liquid penetrants, magnetic particles, ultrasonic testing, radiography, eddy currents, thermal infrared, and acoustic emission tests (Bray and McBride, 1992).

In addition to non-destructive testing, computational mathematical modeling allows the mechanical process to be simulated with certain degrees of freedom, in particular, to simulate the characterization of materials with a high degree of fidelity, it is also possible to repeat the process a number of times, as many times as necessary, in order to analyze phases with greater richness of detail. greater control than if the test were carried out without modeling. This better understanding reduces the so-called "ignorance factor" and thus allows for a lower "safety factor" without sacrificing product reliability (Bray and McBride, 1992). It should also be noted that non-destructive testing, in addition to reducing costs, makes it possible to eliminate the dangers inherent in destructive testing.

In order to achieve the proposed objectives, this work was organized as follows: in section 1 an introduction to the subject is made, as well as to the objective of the work, in section 2 the theoretical framework is presented, a review of the theoretical foundations related to this thesis is made. Section 3 presents the theory of Finite Elements together with the particularities of the implementation of the Small Punch Test. Section 4 presents the results of the implementation of the Small Punch Test model in the ABAQUS© 2019 software. Section 5 presents the conclusions and future work indicating possible lines of research in order to give continuity to this work, and finally in section 6 the Bibliography is presented.

2 THEORETICAL FRAMEWORK

Conventional mechanical testing requires a relatively large volume of material to be tested, and removing it from operational components can affect system integrity. In these situations, mechanical testing based on small sample testing techniques is considered efficient solutions for characterizing the mechanical properties of components. The most important aspect for the creation of a miniature sample is the size of the equivalent reference length (EGL), which must be high enough in relation to the metallurgical characteristics of the material (e.g. grain size, Fig 1) so that the properties obtained are representative of the material and not only the properties of the sample in question but also allow for easy repair of the component.

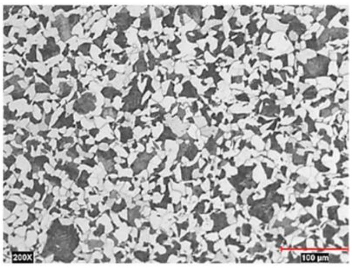
The set of miniature tests includes, for example: the impression creep test (EIC) (Chu and Lee, 1977), the mini "Charpy" test (Kurishita, 1994), the instrumented indentation test (Byun and Hong, 1977), the "Small Ring" test (ESR) (Hyde et. al, 2009), the tensile test on miniaturized specimens

(Kumar et. al. 2014) and the spherical punching test known as Small Punch Test (SPT), etc. The latter is the object of study in this work.

2.1 SMALL PUNCH TEST (SPT)

SPT is a semi-destructive technique used to evaluate the mechanical strength of materials. Following the recommendations of the standards (ASTM 3205, 2020), BS EN 10371:2021 (EN 10371, 2021), it generally involves applying a repetitive cyclic load to a small disc (or square sheet) of material, with a diameter of approximately 8-10 mm and a thickness of 0.5 mm, using a specialized testing device. The load is applied by means of a spherical punch with a diameter of 1 mm that is pressed against the center of the disc. These small samples taken from in-service components not only keep the structural integrity of the part intact, or at least repairable, but also allow for the location of critical component zones (areas most susceptible to stress and damage).

Figure 1. API 5L X70 steel micro structure.



SPT is primarily used to evaluate the fracture toughness and creep strength of materials, especially in situations where available samples are limited or when rapid evaluation is required. In addition, this technique can also be used to evaluate the ductility, fatigue strength, and impact resistance of materials. SPT has the potential to calculate most of the mechanical properties of materials (Hyde and Brett, 2009). The characterization of the mechanical behavior of structural materials, except for their hardness, is by definition destructive. For a better description of the test to be followed, we present the components of the device (Fig 2) and their main characteristics.

Figure 2. Components and their arrangement in the SPT device adapted from (CWA 15627, 2008).

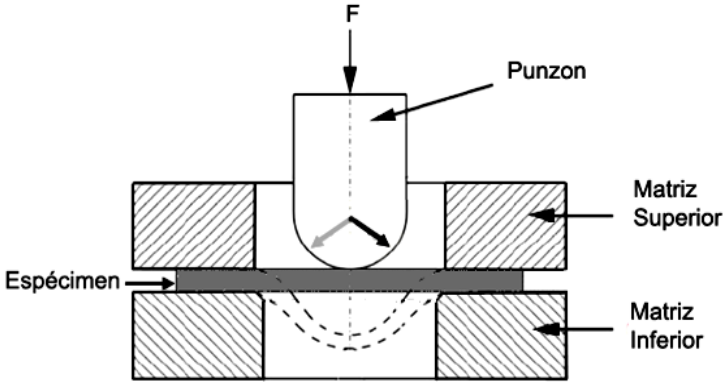


Fig 2 illustrates the shearing process of the punch (cutting tool). The process is caused by the constant pressure exerted by the tool on the surface to be cut specimen (sample, specimen). Separation occurs when the pressure exceeds the maximum shear stress of the material. The punch usually has a hemispherical head or a single sphere, in the latter case it is easier and more practical to replace the sphere, in case it suffers any damage. The specimens, specimens also called specimens are usually of two types, discs of 10 mm in diameter and 0.5 mm thick (Torres and Nunes, 2021), rectangular solids usually 10x10 mm2 in section and 0.5 mm thick. The biggest difference between circular and rectangular geometries is the contact area between the specimen and the matrices. The rectangular section has a larger contact area, making it easier to attach to the SPT test set. The dies correspond to fastening solids, it is important to note that it is important to consider the friction of the components in contact.

2.2 FORCE VS DISPLACEMENT

The SPT allows the determination of the curve of force vs displacement, the force is that which the punch applies on a central point of a small sample, initially it is a point as the punch called indenter moves until it crosses the sample completely. SPT tests can be performed at both ambient and elevated temperatures, evaluating mechanical resistance at different temperatures is especially important in industry, where materials can be exposed to different temperature conditions, ABAQUS makes it possible to create these temperature conditions, in order to obtain fracture properties and elastic-plastic creep, more realistic.

Generally, a force-displacement curve obtained from SPT at room temperature can be divided into four regions (see Fig 3). From this curve, mechanical properties are calculated, such as: yield strength, strength limit, fracture toughness and creep properties (the latter being the most explored). SPT has the potential to calculate most of the mechanical properties of materials (Hyde and Brett, 2009). The characterization of the mechanical behavior of structural materials, except for their hardness, is by definition destructive.

Figure 3. Curve Applied force vs displacement of the center point of the specimen.

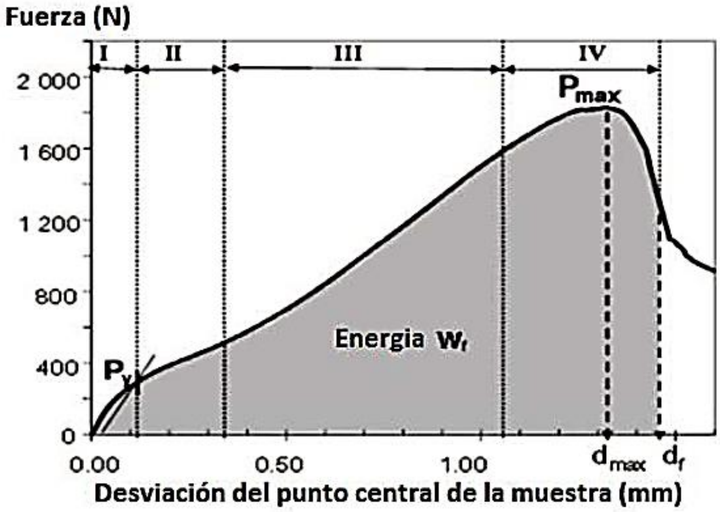


Fig 3 illustrates the force-displacement curve obtained by SPT, in this curve four different zones can be distinguished: zone I elastic deformation, in this zone a slight influence is observed at the initial contact of the punch with the specimen, zone II elastoplastic transition, in some parts of this zone specifically those that are in contact with the periphery of the punch, Zone III is the result of generalized plastic deformation, zone IV characterized by plastic instability and fracture.

This technique has practical application in different areas, but the oil and nuclear industries are the ones that took advantage of this technology the most and made it possible to study and develop it.

2.3 API 5L-X70 STEEL

API 5L-X70 steel is a type of high-strength steel used in the manufacture of pipes for the oil and gas industry. This steel is characterized by its high mechanical strength, which makes it ideal for withstanding high pressures and extreme temperatures.

The importance of API 5L-X70 steel lies in its ability to ensure the safety and reliability of piping systems used in oil and gas exploration and production. In addition, its high corrosion and impact resistance make it a reliable choice for critical infrastructure projects.

Pipelines used in the transport of petroleum products require materials with an exceptional combination of mechanical properties, including high mechanical strength, toughness and fatigue resistance. API 5L X70 Steel is presented as one of the best options, its study and characterization is the focus of this work.

API steels, with high strength characteristics, combined with good weldability, low level of inclusions and good surface quality, are specified by the American Petroleum Institute (API) in the case of APISPEC 5L 2007. The main guide for pipe welding is an API 1104 standard, which provides the necessary data to obtain good quality welded joints. The joints in the pipes are welded in the field and only on the external side. As the tube is fixed, welding must be performed in all positions, welded joints must be addressed in the scope of characterization of this steel.

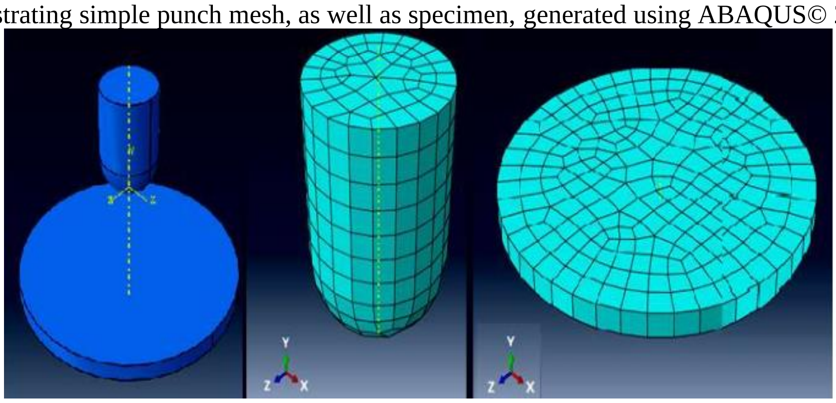
For API class 5L, chemical composition and mechanical strength grades vary from API 5L A25 to API 5L X80. In designations consisting of the letters A or X, the last two digits specify the minimum flow limit values for the material in ksi. Thus, API 5L X80 steels have a minimum flow value of 80 ksi (-550 MPa). According to the requirements of the pipes, API-5L-A and B grades are used in low-pressure pipes, while API-5L X42, X46, X52, X60 and X-70 grades are used in high-pressure pipes.

2.4 FINITE ELEMENT METHOD

The FEM simulation of the characterization technique "Small Punch Test" is important because it allows to simulate the conditions close to the real ones and to improve the analysis of the results, since it is possible to repeat the process n times, choose parts of the simulation to perform a more detailed temporal analysis. In addition, FEM can be used to model the mechanical behavior of the material, virtual testing can be performed before testing physical samples, reducing the cost and time of developing new materials.

In this context, FEM appears as a numerical approach that solves partial differential equations approximately. From this, the main idea of the WEF is the division of the problem domain into regions with simpler geometry (see figure 3). These geometries can be triangular, cubic, square, among others, depending on the type and dimension of the problem. The simplest forms are connected by nodes, and by this arrangement it is possible to obtain an approximate solution. This is called finite element mesh, and the process of making meshes is called mesh generation. Each component is a continuous structure, using polynomial functions in conjunction with matrix operations, the continuous behavior of each element is implemented as a function of the geometric properties and the properties of the material. Loads can be applied within each element (gravity, dynamics, thermal, etc.), either on the surface of the element or on its nodes.

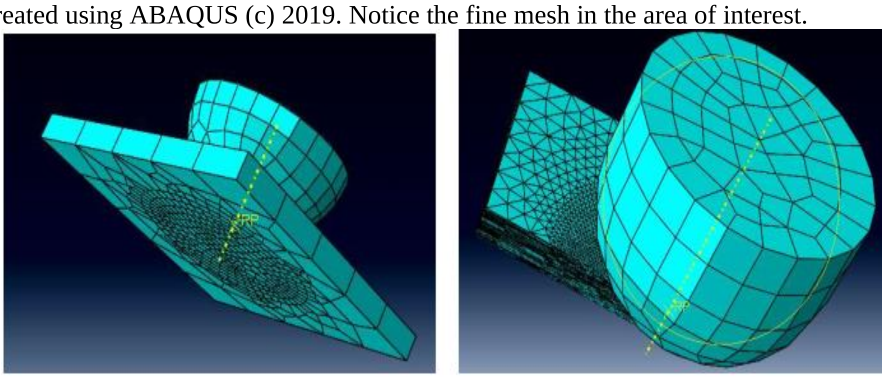
Figure 3. Illustrating simple punch mesh, as well as specimen, generated using ABAQUS© 2019 software.



The nodes of the elements are their fundamental governing entities, since it is in them that the element interconnects with other elements and where the elastic properties of the element are finally established, boundary conditions are assigned, and finally the forces (contact or body) are determined. The degrees of freedom of the nodes are the independent motions of rotation and translation that can occur. Each node can have a maximum of three degrees of freedom of translation and three degrees of rotation.

Once each element within a structure has been locally defined in the form of an array, the elements are assembled (tied) globally by their common nodes in a global array of the system. The applied loads and boundary conditions are then specified, and through matrix operations, the values of all unknown degrees of freedom of displacement are determined. Knowing the displacements, the deformations and stresses can be calculated using the equations of continuous mechanics and the constitutive laws of materials.

Figure 4. Finite element types (a) complete model and (b) simplified model, taking advantage of the symmetry of the problem. Meshes created using ABAQUS (c) 2019. Notice the fine mesh in the area of interest.



In FEM, the accuracy of the method depends on several factors present in the mesh, such as the number of nodes, type, size, and number of elements. The solution tends to converge to the exact solution as the mesh is refined (examples in Figure 4), i.e., the greater the number of elements, the greater the accuracy of the results.

The analysis carried out by the FEM is divided into three distinct parts, namely: pre-processing, processing and post-processing. In the first step, you need to define the geometry, analysis type, mesh, material properties, and boundary conditions. In processing or analysis, the desired type of analysis (using linear or nonlinear equations, among other configurations) must be defined to obtain the nodal displacements. In the post-processing step, the results are presented, such as voltages, heat flow, convergence, safety factors, etc. This sequence of steps is followed by most FEM software, among the most well-known ABAQUS, ANSYS©©, FEMLAB © etc. From this point on, when we refer to the software used, we are referring to ABAQUS©.

FEM modeling is also important because it can help identify critical regions of the sample where failures will occur during testing. By knowing these regions, it is possible to modify the geometry of the sample or adjust the test conditions to avoid premature failures. Overall, FEM modeling is a valuable tool for improving the efficiency and accuracy of the SPT characterization technique.

3 IMPLEMENTATION OF THE SPT USING ABAQUS-FEM

3.1 CHARACTERISTICS ABAQUS SOFTWARE

ABAQUS 2019 is a highly adaptable finite element software that allows you to create models with extremely high geometric complexity, it is used in this work to implement the SPT model and its shape adaptations to characterize API 5L x70 steel. Consequently, these models can be tested and examined for accuracy, with the possibility that their parameters can be modified in a way that optimizes the results.

Table 1. Units used in ABAQUS Software compared to SI units.

Magnitude	**IS**	**Units in ABAQUS**
Longitude	m	mm
Time	s	s
Mass	kg	ton
Strength	N	N
Pressure	Pa(N/m^2)	MPa (N/mm^2)
Velocity	m/s	Mm/s
Energy	J	mJ

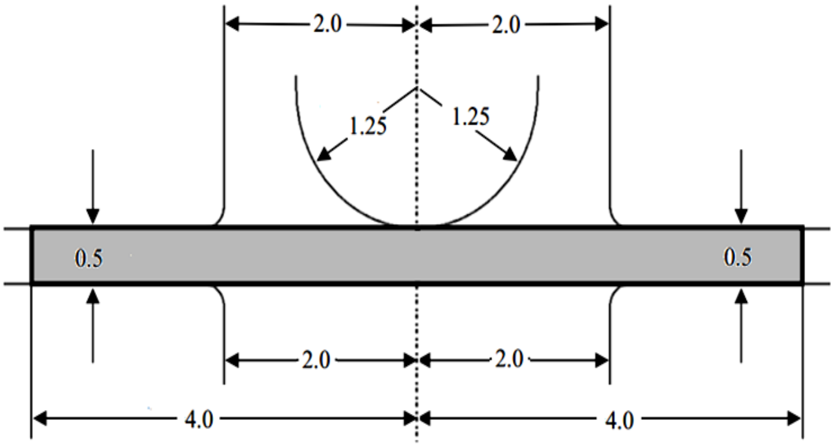
3.2 UNIT SYSTEMS USED IN ABAQUS

Before you begin modeling the problem, it's important to warn about the drive system used by the software. Numerous properties and measures were used in the implementation of the model. ABAQUS does not work with specific units. This is not a user-made choice but a software design choice. Therefore, the system of units must be consistent. In this case, the basic International System (IS) was not used. Two main arguments support the decision. Initially, it is not suitable for measurements, and second, the units of measurement do not match those used by the rest of the literature. Therefore, a variation with multiples and submultiples has been used, as detailed in Table 1.

3.3 SPT BASE MODEL

The cross-section of the SPT geometric model is shown in Figure 5. The model consists of an upper and lower die, a punch, and a sample. The latter is a thin circular disc with a diameter of 8 mm and a thickness of 0.5 mm, while the punch radius is 1.25 mm. The inner diameters of the upper and lower die are all 4mm. Both the upper die and the bottom die have a gunwale bend radius of 0.2 mm.

Figure 5. Cross-section of the SPT model, all units in mm.



3.4 ABAQUS MODULES

Solving a problem with any finite element software goes through three phases. Each phase with its corresponding modules as described below:

1. Pre-processing phase: Pre-processing consists of the definition of the problem itself. This is where the model is introduced into the program, taking into account the necessary simplifications. The following modules belong to this phase:

Part.- This module defines the geometries and basic parts that make up the model. This can be done in the ABAQUS editor itself, although there is also the possibility of importing geometries from other CAD programs in case of more complex geometries. The types of parts that can be created are: deformable, discrete rigid, analytical rigid or Eulerian.

Property.- Defines the materials of the parts created (Fig 6). Those materials are mapped to sections, and the sections are mapped to geometries created in the Part module.

Assembly.- The "assembly" module allows you to place the previously defined parts in their relative positions in space.

Step.- This module introduces the type of analysis to be performed. It also allows you to define how long the simulation will last.

Interaction.- after having placed the different pieces in space, it is necessary to establish the relationships between them, from joining geometries to establishing the type of contact between the pieces (Fig7).

Figure 6. Illustrated the definition of the properties of the sample.

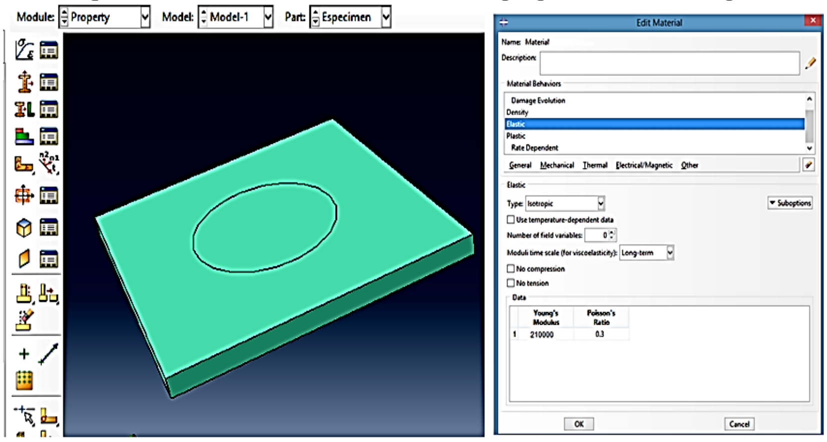
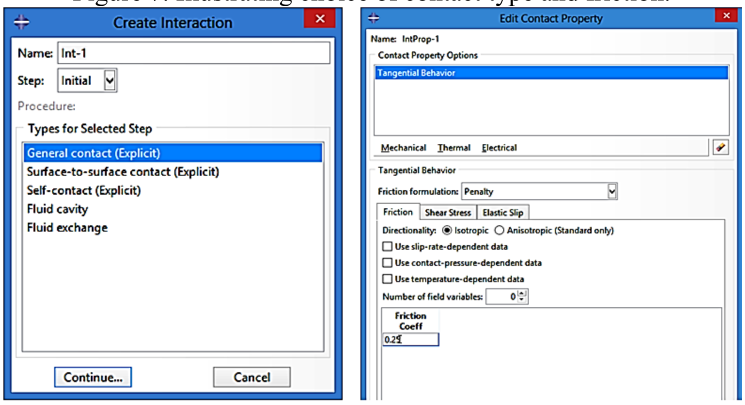
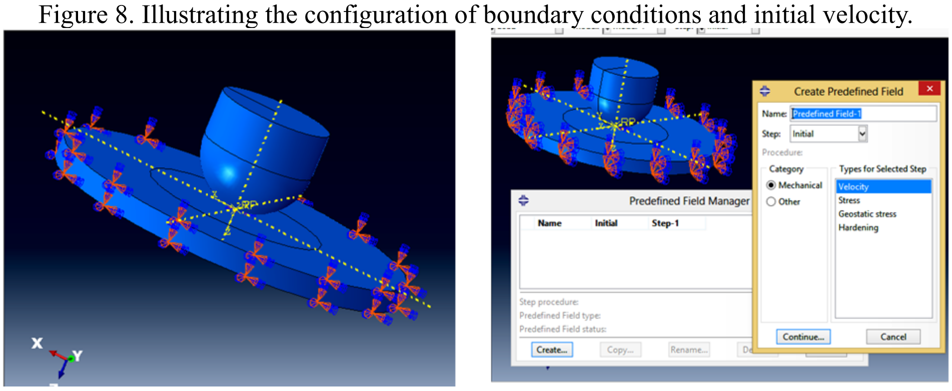


Figure 7. Illustrating choice of contact type and friction.



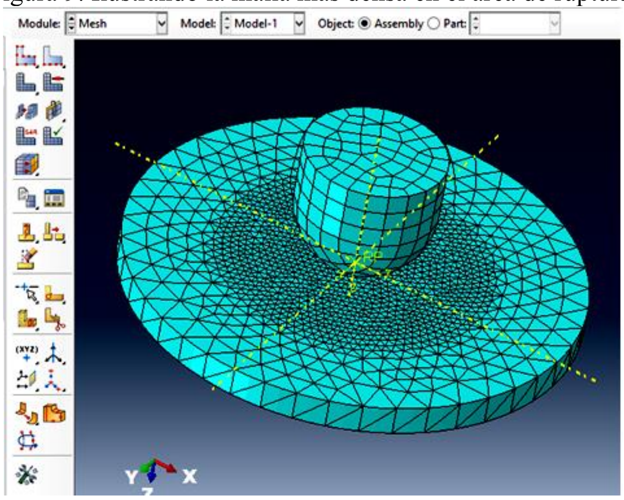
Load.- It allows you to enter the loads and boundary conditions of the parts, as well as the restrictions on initial movements and speeds (Fig 8).

Figure 8. Illustrating the configuration of boundary conditions and initial velocity.



Mesh.- Generates the meshes that discretize the model. A more refined mesh will contain a greater number of elements and therefore, the results will be more accurate. However, the computation time will increase considerably, making it necessary to balance the mesh resolution in relation to the tolerable margin of error and the available computation time (Fig 9).

Figura 9. Ilustrando la malla más densa en el área de ruptura.

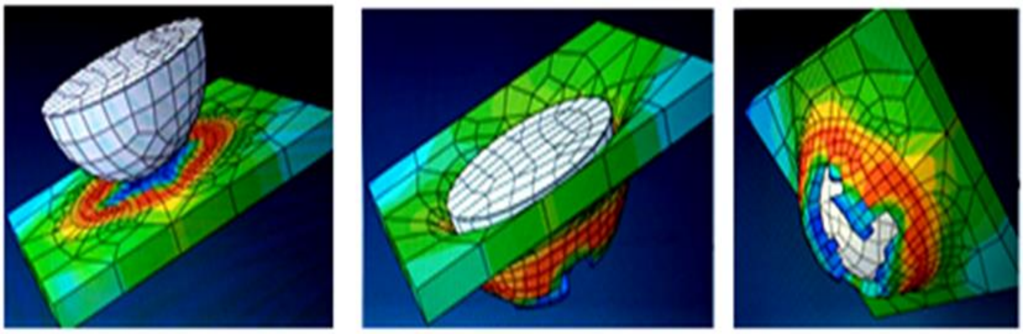


2. **Processing phase**: In this part of the process, the simulation of the introduced model is computed. It does not require any input from the user. The calculation time depends on the complexity of the problem and the resolution of the mesh. It consists only of the **Job module**, in which the number of cores (computer processors) is set. It is possible to monitor the evolution of the results and interrupt the process if necessary.
3. **Post-processing phase:** In the final phase, the user can view the results of the analysis and check the evolution of the variables of interest (**Visualization module).** This part will be presented in the next section,

4 RESULTS AND CONCLUSIONS

SPT simulations were performed using ABAQUS(c) 2019 software in order to evaluate the elastoplastic behavior of API-5L-X70 Steel, in terms of SPT, punches and dies are considered rigid elements because the deformation they undergo during the test is negligible compared to the deformation that occurs in the test specimens. Fig 10 illustrates the case that the rectangular specimen (10x10 mm2 and 0.5 mm thick) was fixed using the boundary conditions applicable using the software.

Figure 10. Simulation of the main punching stages, a) impact, b) penetration, c) burst. Figures obtained with the model being implemented.



The specimens have been considered as elastoplastic deformable bodies and have been modeled with elements of size 0.025 mm in the areas where they break.

As can be seen in Fig 11, the test consists of fixing the specimen (3 mm in diameter and 0.25 mm in thickness) by holding it firmly between two dies, and then deforming the specimen until it breaks using a hemispherical head punch (1.25 or 0.5mm radius respectively for the two types of specimens mentioned). Note that the specimen was represented as a quarter of its total shape in the geometric model taking advantage of the symmetry of the same, the dies and the punch, but the calculations were performed in their entirety, this was necessary due to the limitations imposed by the software (student version) but the results were not compromised.

Figure 11. Geometrically simplified model of the SPT (disabled to display the upper fixing matrix in order to better observe the details of the other components),

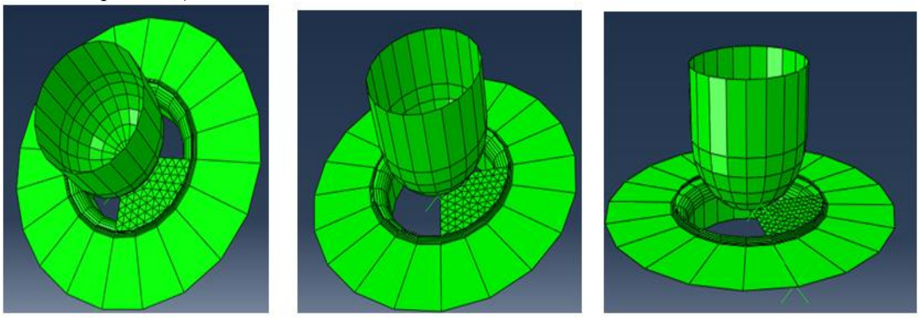
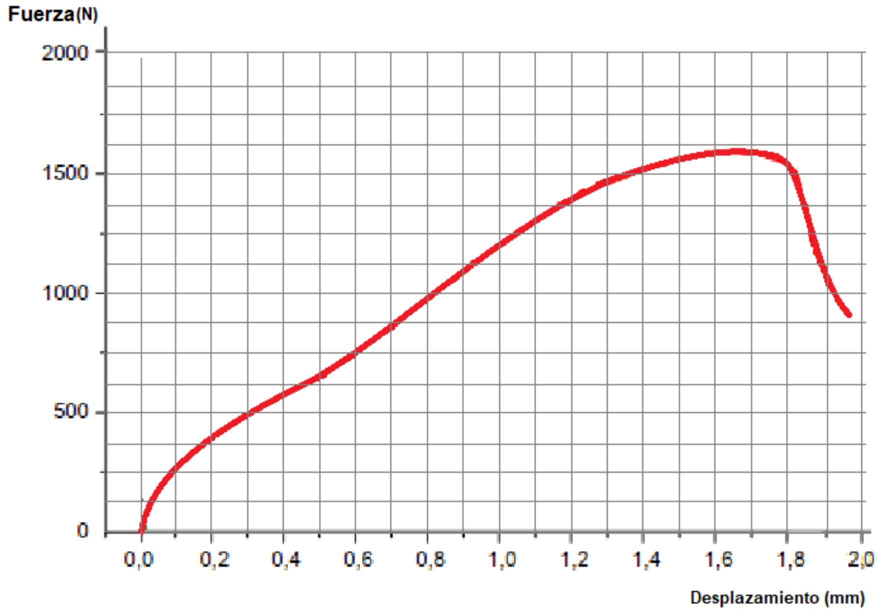
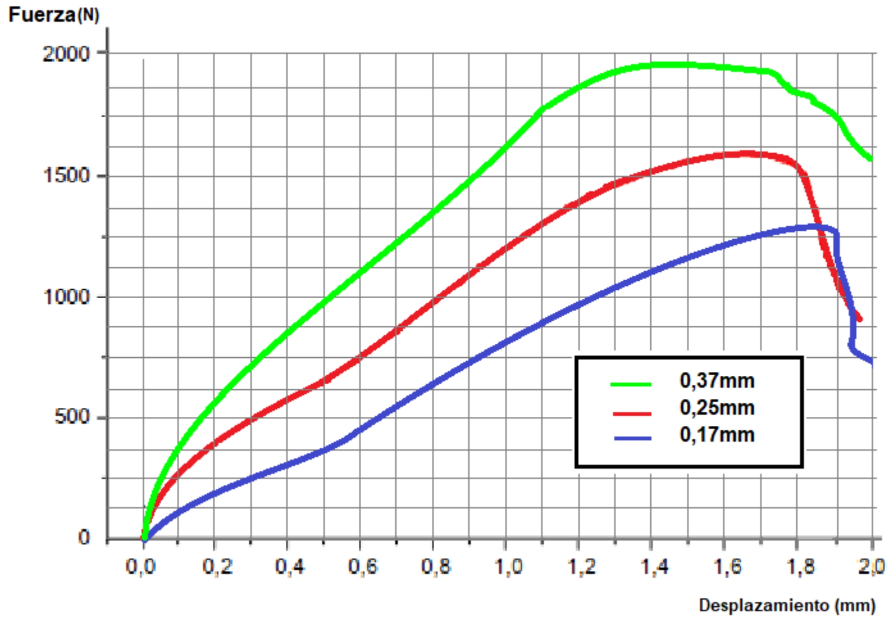


Fig 12.a illustrates the force-displacement curve obtained by simulating the SPT in ABAQUS (c) 2019, for the circular-shaped specimen attached to the fixing matrices, in this curve the four characteristic zones are observed: i.e. zone I where elastic deformation occurs, in the range of 0 to 0.2mm, in this area a slight influence is observed at the initial contact of the punch with the specimen, it is also possible to observe zone II in the range 0.2 to 0.5mm, called elastoplastic transition, which presents effects of contact with the periphery of the punch; In zone III, generalized plastic deformation occurs between 0.5 and 1.4 mm, finally we observe zone IV characterized by plastic instability and fracture, in the range 1.4 to 1.83 mm.

Figure 12. a) Curve Applied force vs displacement of the central point of the circular specimen, attached to the fixing matrices, coefficient of friction μ = 0.1; b) Force vs. displacement curves, for three thicknesses of the circular specimen 0.37; 0.25 and 0.17 mm, coefficient of friction μ = 0.1.



(a)



(b)

Of Fig 12.b, which represents the curves obtained with the same steel for specimens of 0.37, 0.25 and 0.17mm thickness, of the Force vs. displacement curves we observe the dependence of the thickness of the specimen used.

Finally, we can conclude that it was possible to model the "Small Punch Test" successfully, using the ABAQUS© 2019-FEM software, consequently the main objective of the work was achieved: "To model the small punch test using finite elements to obtain the characteristic stress-strain curve of API 5L-X70 steel". These results are the beginning of an ongoing work.

REFERENCES

Annual Book of ASTM Standards, (2020) American Society for Testing and Materials.

Bray, Don and E. McBride, (1993) Nondestructive testing techniques, Wiley. Volume9, Issue2, March/April 1993, Pages 159-159.

EN 10371 Metallic materials, (2021) Small punch test method. European Committee for Standardization (CEN).

Byun, T. S.; Hong, J. H.; Haggag, F. M. Farrell, K.; Lee, E. H. (1977), Measurement of Through-the-Thickness Variations of Mechanical Properties in SA508 Gr.3 Pressure Vessel Steels Using Ball Indentation Test Technique. International Journal of Pressure Vessels and Piping.

Chu, S. N. G., and Li, J. C. M. (1977). Impression creep; a new creep test. Journal of Materials Science, 12, 2200-2208.

CWA 15627, (2008) Small Punch Test for Metallic Materials, CEN Workshop Agreement, European Committee for Standardization.

Hyde, TH, Sun, W, Brett, SJ., (2009) Some recommendations on standardization of impression creep testing. In: Proceedings of ECCC conference on creep and fracture in high temperature components – design and life assessment (eds Shibli, IA, Holdsworth, SR), Zurich, Switzerland.

Kumar, K.; Pooleery, A.; Madhusoodanan, K., (2014) Use of Miniature Tensile Specimen and Video Extensometer for Measurement of Mechanical Properties. Bhabha Atomic Research Centre, Mumbai, India.

Kurishita, H., (1994) Current status of small specimen technology in Charpy impact testing. Journal of nuclear materials.

Torres-Castellanos, N., and Núñez-López, A. M., (2021) Tensile behavior of normal-strength steel-fiber green ultra-high-performance fiber-reinforced concrete. ACI Materials Journal.