



24th COBEM - 2017



24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-1106

PRELIMINARY STUDIES IN THE COMPUTATIONAL MODELING OF A SAFETY VALVE USED IN THE OFFSHORE INDUSTRY

Marcella Monnique Mello da Silva

Leonel Leonardo Delgado Morales

Federal District University Center – UDF, Brasília, Brazil

marcellamelloeng@gmail.com

leonel.morales@udf.edu.br

André Nepomuceno Trajano

University of Brasilia – UnB, Department of Mechanical Engineering, Brasília, Brazil

andrentrajano@gmail.com

Abstract. Blowout Preventer (BOP) valves were designed to trigger in emergencies situations by applying a compression force on the drill pipe, which will cause plastic deformation until its rupture. According to the report on the execution of Norske Veritas in 2011, the photographic records of the pipeline showed that the BOP have worked nevertheless did not act properly, it only carried out a crushing of the material instead of sealing the tube. As a result, there are indications failures of the safety device, which can be directly related to the way in which these devices were designed considering the mathematical models used when the failures were initiated. For that reason, it is necessary to launch investigations to propose a design criterion that fulfills the requirements of this type of device, considering that the pipes where the valves act have elastoplastic properties. The objective of this paper is to carry out a preliminary study concerning a computational modeling of measuring valve in the operating stage, simulating the drive and then analyzing the numerical results in the state of the pipe stresses. As an illustration, will be included the geometry of the riser and the rafters that constitute this mechanism, with their correspondents. And according to the preliminaries results, it is possible to evaluate the model used and, subsequently, to carry out necessary modifications to improve the study.

Keywords: Computational Modeling, BOP, Failure Criteria, Elastoplasticity

1. INTRODUCTION

Deep-sea oil extraction practice can be executed through floating or fixed to the ground platforms. The platforms have a drill column called Riser and a safety device designed to minimize damage from leaks, called Blowout Preventer (BOP). This device activates a valve with hydraulic drive, consisting of diametrically opposed rams, designed to apply a compressive force in the radial direction of the tube in order to plastically deform it until its rupture and, consequently, stop the leak.

But otherwise, in April 2010, when one of the largest environmental disasters in history occurred, the leak of 780 million liters of oil was reported, 11 workers were killed and 40 billion of dollars were lost. The accident occurred in the Gulf of Mexico caused by an internal pressure rise in the drill pipe generating a warning signal for the platform and the activation of safety devices. (Det Norske Veritas, 2011).

The extraction tube is characterized as 4340 alloy steel, high tensile steel composed of Chromium, Nickel and Molybdenum. This alloy combines strength and toughness, being able to work in the different types of application and levels of request.

The aim of this paper is to present the results of a preliminary study referring to von Mises and Johnsons-Cook models to evaluate the modeling of a safety valve and finally understand the effect of the operation on the cutting process of the tube in the first stage of the drive.

2. THE VON MISES MODEL

The fault criterion is based on the distortions caused by the strain energy. When deforming a material by external loading, it tends to store energy internally throughout its volume. It is called the energy per unit volume of the material the deformation energy density. If it is subjected to a uniaxial voltage, σ , we can write this density as:

$$\Phi(\sigma, \varepsilon^{-P}) = q - \sigma_y (\varepsilon^{-P}) \quad (1)$$

The term ε^{-P} is the equivalent plastic deformation, corresponding to the internal isotropic hardening variable. The component q in the equation 1 is called the Mises equivalent voltage, which represent a function of the deflection voltage (S):

$$q = \sqrt{\frac{3}{2} S : S} \quad (2)$$

The leakage criterion of the maximum distortion energy is indicated by the flow occurring in a ductile material when the distortion energy per unit volume of the material is equal to or greater than the distortion energy per unit volume of the same material subjected to a flow in a simple traction test.

3. JOHNSONS-COOK MODEL

The critical point within the engineering studies is the failure of a material. The fracture of material is characterized by the total inability to exert the role which it was designed. The failure process is relatively complex and difficult to predict the exact location that will occur. Many researchers seek to develop models capable of carrying out such prediction.

Therefore, the natural causes can be expected, many of failures criterion appear to predict and give parameters values that have to be controlled, such as stress in an specific area, the displacement at some point and/or the porosity level . The main purpose of any criterion is predict the exactly moment of failure. In continuum mechanics, the fracture involves the concept of internal voids.

It is inferred that every material have an initial level of voids. As soon as the strain increase, we have the growth of these voids, the form of new and the coalescence of voids. The process of failure of any material can be described as the evolution of a parameter called porosity. The total failure occurs when a determined value of porosity is reached. For this reason, the design of some component can be changed to achieve the requirements established in an earlier moment.

Here, the Johnson-Cook Constitutive Model is applied to evaluate the behavior of Blowout Preventer case. This model is completely empirical and its parameters are reached in laboratory tests. The Johnson-Cook model is one of the most common models used in dynamic cases of ductile materials and covers the mechanical behavior and the ductile damage parameter of the material that will be presented at the end of this section. And this model is well applied in situations of high strain rates, like the deformation in BOP and when fracture and failure of materials occurs. The yield stress for this model is given by Eq. 3:

$$\sigma_y = [A + B \varepsilon_n^p][1 + C \ln \varepsilon_p^*][1 - T_H^m] \quad (3)$$

where:

A, B and C = material constants determined in laboratory

- A = basic yield stress at low strains
- B = strain-hardening effects
- C = strain rate effects

ε_n^p = effective plastic strain

ε_p^* = normalized effective plastic strain rate

T_H^m = homologous temperature

4. CONTACT THEORY

According to Kikuchi and Oden (1988), the problem of contact between two bodies constitutes an important aspect of mechanics of solids, and it can be approached in several ways, among others, through variational theory. Contact mechanics have been studied over a long period of time and there are many engineering applications (WRIGGERS, 2016). The main methods used are the Lagrange method and the penalties method Hertz Law.

The theory of Heinrich Hertz is based on the classical theory of elasticity, continuum mechanics and Amontons laws. To determine a problem as Hertzian contact, four suppositions have to be achieved:

- Strains are small and inside the elastic region;
- Surfaces are continuous and non-conforming (contact area is smaller than entire dimensions of contacting bodies);
- Each body is considered an elastic half-space;
- The surfaces are frictionless.

The dynamic situation of BOP valve, the shear forces applied to cut the riser, the friction between rams and riser and level of related strains turns this contact problem a violation of Hertz Law. In order that, the simulation performed here are considered a non-Hertzian problem.

To understand the formulation of contact mechanics is considered a one-dimensional problem from the variational formulation (Wriggers, 2016). The solution presented for the Lagrange multipliers method gives an equation similar to the one used in the variational model, where an operator dependent function is established and the displacement in a determined direction. To fully understand the contact mechanical formulation it is considered one dimensional problem from variational formulation (WRIGGERS, 2016):

$$\Pi(u, \lambda) = \frac{1}{2}ku^2 - mgu + \lambda c(u) \quad (4)$$

The operator λ is equal to a reaction force (f_R). The derivative function in terms of u and λ results in two other equations:

$$ku\delta u - mg\delta u - \lambda\delta u = 0 \quad (5)$$

$$c(u)\delta\lambda = 0 \quad (6)$$

The first expression, Eq. 4, represents the equilibrium state of the mass point when its contact with the surface occurs. The Eq.5 represents the kinematic constraint for the contact. By giving the right conditions it is possible to prove the equality of the friction force and the Lagrange multiplier.

The penalty method consists basically in modifying the formulation by adding a penalty or restriction and using the formulation based on the Lagrange multiplier a new mathematical model is obtained which includes a penalty term, which constitutes a restriction:

$$\Pi(u, \lambda) = \frac{1}{2}ku^2 - mgu + \frac{1}{2}\varepsilon[c(u)]^2 \quad (7)$$

where: $\varepsilon > 0$.

By taking the equation 7, it is possible to obtain an expression for the constraint that follows:

$$cu = kh - mgk + c\varepsilon$$

Observing the expression above it is possible to verify that when $mg > kh$, a penetration of the point on the contact surface will occur, this penetration depends on the parameter of penalty.

As for the penalty, two possibilities can be considered:

- i) When $\varepsilon \rightarrow \infty \Rightarrow u - h \rightarrow 0$, we are faced with a small penetration;
- ii) When $\varepsilon \rightarrow 0$, it comes to high penetration.

Another aspect to be observed is that the solution allows the reaction force to be obtained and can be written such as:

$$\lambda = F_R = \varepsilon c(u) = \frac{\varepsilon}{k+c}(kh - mg) \quad (8)$$

We can notice that when $\varepsilon \rightarrow 0$, the penalty method is reduced to the Lagrange method.

From the solution point of view, the penalties method becomes cheaper and efficiently equivalent to the Lagrange method. In this paper we will use the penalty method because, despite of not providing the exactly result, the prediction of values is quite approximated and compensate the computational cost.

5. VALVE BLOWOUT PREVENTER – BOP

The valve has a height of approximated 26 meters, 400 tons and 4 stages of operation. At the bottom the valve has two rams in charge of executing the cut of the riser by means of tangential forces. On Fig. 1 shows the Deep-water Horizon BOP Port Side.

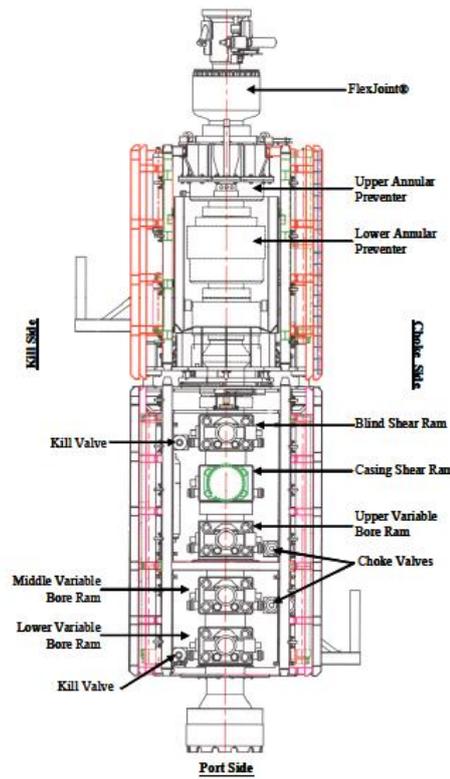


Figure 1. Blowout Preventer Device.

Based on the data provided by the failure analysis report (Det Norske Veritas, 2011) it has the geometry and dimensions of the parts to which were simulated numerically, according to Fig 2, Fig 3 and Fig 4.

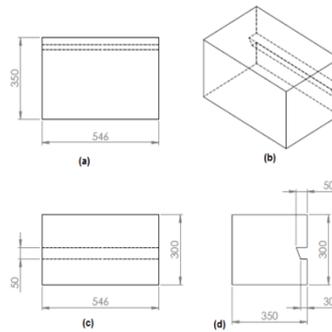


Figure 2. Views of the female ram: a) upper; b) isometric; c) frontal; d) right lateral.

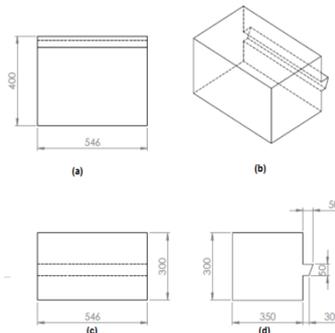


Figure 3. Views of the ram: a) upper; b) isometric; c) frontal; d) right lateral.

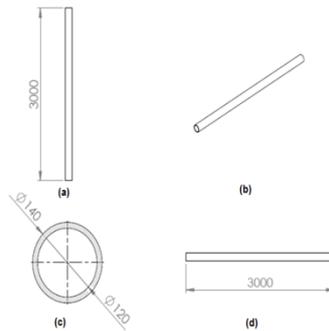


Figure 4. Tube views: a) upper; b) isometric; c) frontal; d) right lateral.

5.1 The Computational Modeling

The geometry of the valve was modeled considering the actual characteristics of the device and from this geometry the computational simulation was designed, using ABAQUS, commercial software. The main components of the system were the tube and two battering rams (male and female), according to Fig 5.

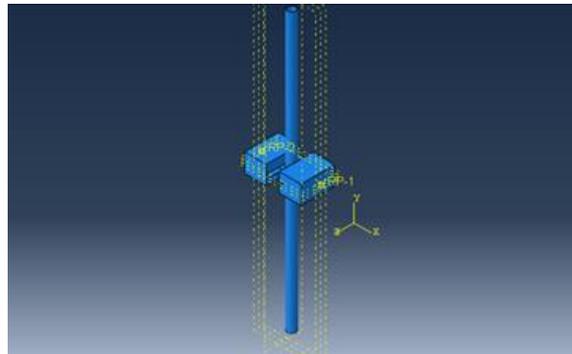


Figure 5. Positioning system

At the figures 6 and 7 this modeling can be seen only with three boundary conditions. The first one is that the both ends of the riser are fixed. The second and third contour conditions refer to the displacement applied to simulate valve operation and to be able to extract information resulting stresses in the riser.

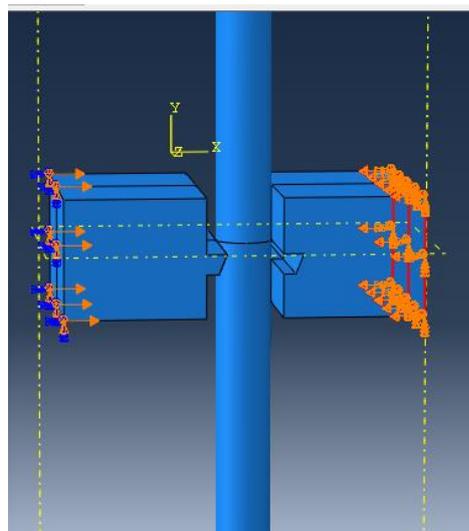


Figure 6. Boundary Conditions - displacement

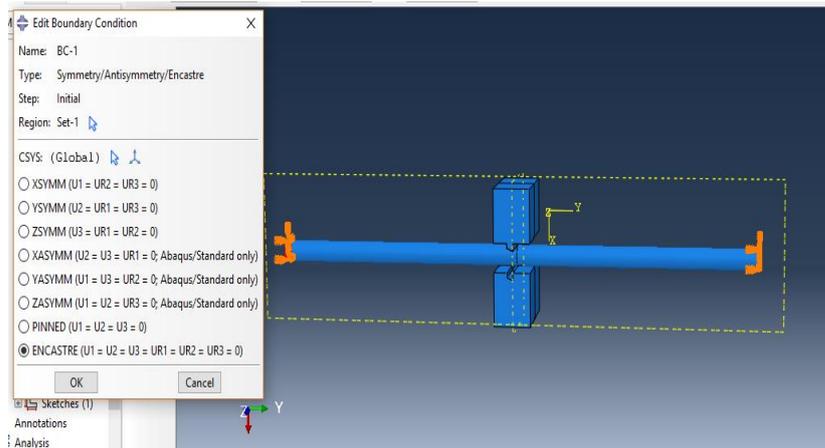


Figure 7. Boundary Conditions – riser ends set

The construction of the mesh in the three components was executed by considering the most interesting points, where the mesh should be more refined to seek results closer to the real. The element applied on this 3D case should have six well defined and connected faces. And to perform the mesh on the riser, an explicit linear hexadecimal element (8 nodes - C3DR in the ABAQUS) was selected in the central and hexahedral linear pattern in other regions. Then, the table 1 shows the mesh applied in each piece of the system.

Table 1. Mesh characteristics

| Components | Size of the element (mm) | Number of elements | Number of nodes |
|------------|--------------------------|--------------------|-----------------|
| Male ram | 20 | 8400 | 9802 |
| Female ram | 40 | 1204 | 1590 |
| Riser | 20 | 4972 | 9307 |

The two rams also received refinement in the region that is in contact with the riser, with the same objective of increasing the points of contact between the parts (Fig 8 and 9). While the main object of analysis is the riser, the mesh in the rigid bodies is not so relevant, the greater number of points of contact in the region of interest is more relevant.

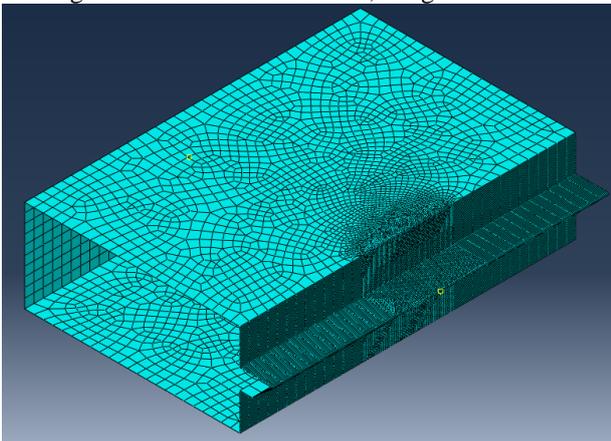


Figure 8. Mesh refinement in Male ram

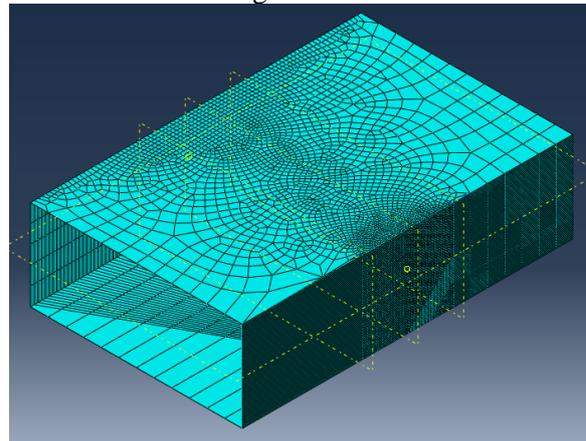


Figure 9. Mesh refinement in Female ram

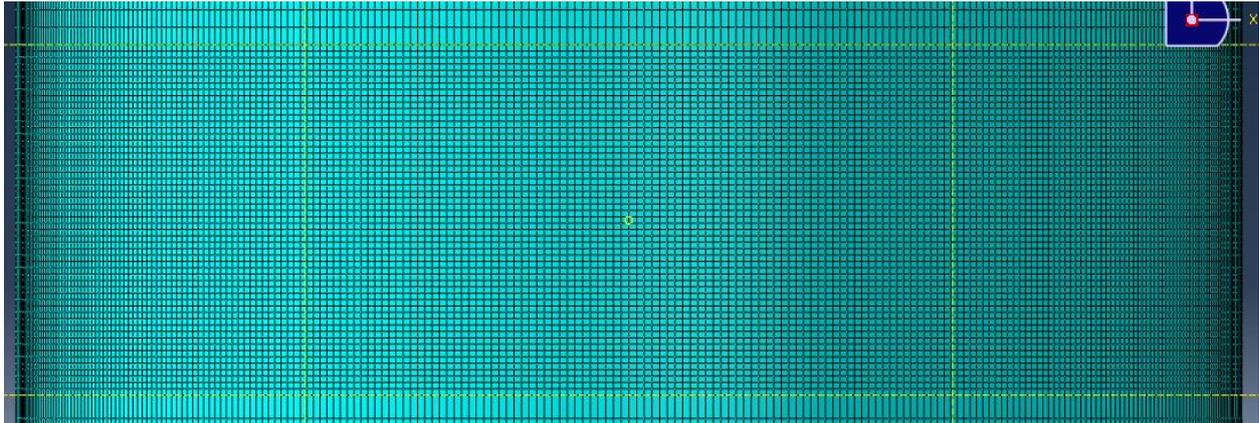


Figure 10. Mesh refinement in central region of tube

In the case of an elastoplastic problem, the hardening curve of the material should also be inserted. So, consequently, the model was tested with a hardening curve obtained from literature of the alloy steel 4340, Tab 2.

Table 2. Hardening properties

| | Yield Stress | Plastic Strain |
|---|--------------|----------------|
| 1 | 830 | 0 |
| 2 | 847.94 | 0.00499 |
| 3 | 870.27 | 0.01223 |
| 4 | 889.57 | 0.01969 |
| 5 | 906.92 | 0.0277 |
| 6 | 923.23 | 0.03673 |
| 7 | 938.76 | 0.04709 |
| 8 | 954 | 0.06005 |

6. RESULTS

This paper, initially, involved the use of two cutting geometries for the male ram and displacement was imposed on both battering rams instead of force. Applying a 50 mm displacement on both rams, it can be notice that the higher stresses in the problem are located in the center of the raiser caused by the high riser length.

The tensions found by the von Mises criterion resulted in 1042 MPa with rounded geometry (Fig 12). And with the most severe geometry, the von Mises maximum stress was 954.7 MPa (Fig. 13). It was noted that the material does not fail with the sharp geometry, but with the rounded geometry the maximum tension found is above the maximum hardening stress. Therefore, the rupture of the material can be considered.

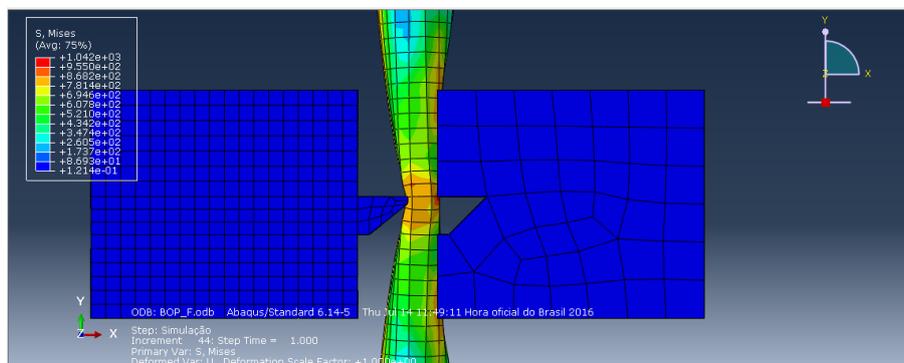


Figure 12. von Mises stress with rounded geometry

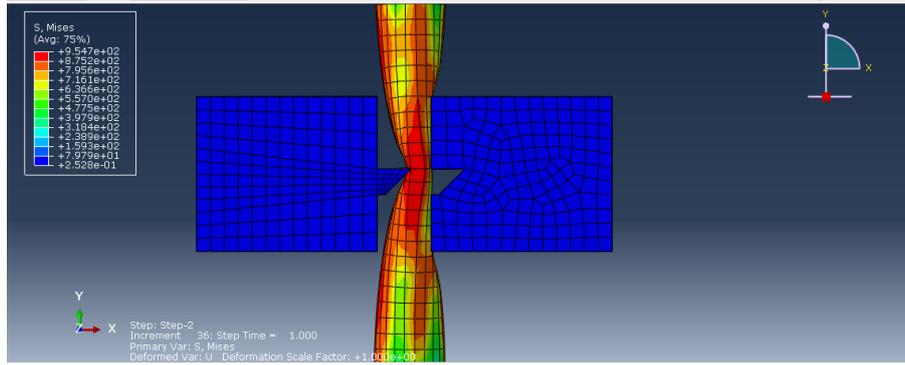


Figure 13. von Mises stress with sharp geometry

The hardening curve inserted on the Abaqus has a maximum stress strain equal to the second von Mises stress obtained. So, the material fails with the sharp geometry, but with it more rounded the maximum tension found is above the maximum stress of hardening.

The sequence of Figures 14a-14f shows the evolution of von Mises equivalent stress as the ram advanced in the cut. A tension distribution with the highest tension as expected is observed in the contact region. While the ram advances, the contact surface increases and the riser deforms in the shape of the curvature of the riser tip. In the first images, a crushing of the tube is observed until Fig. 14d. The elements began to be deleted until, according to Fig. 14f, the total separation of the tube.

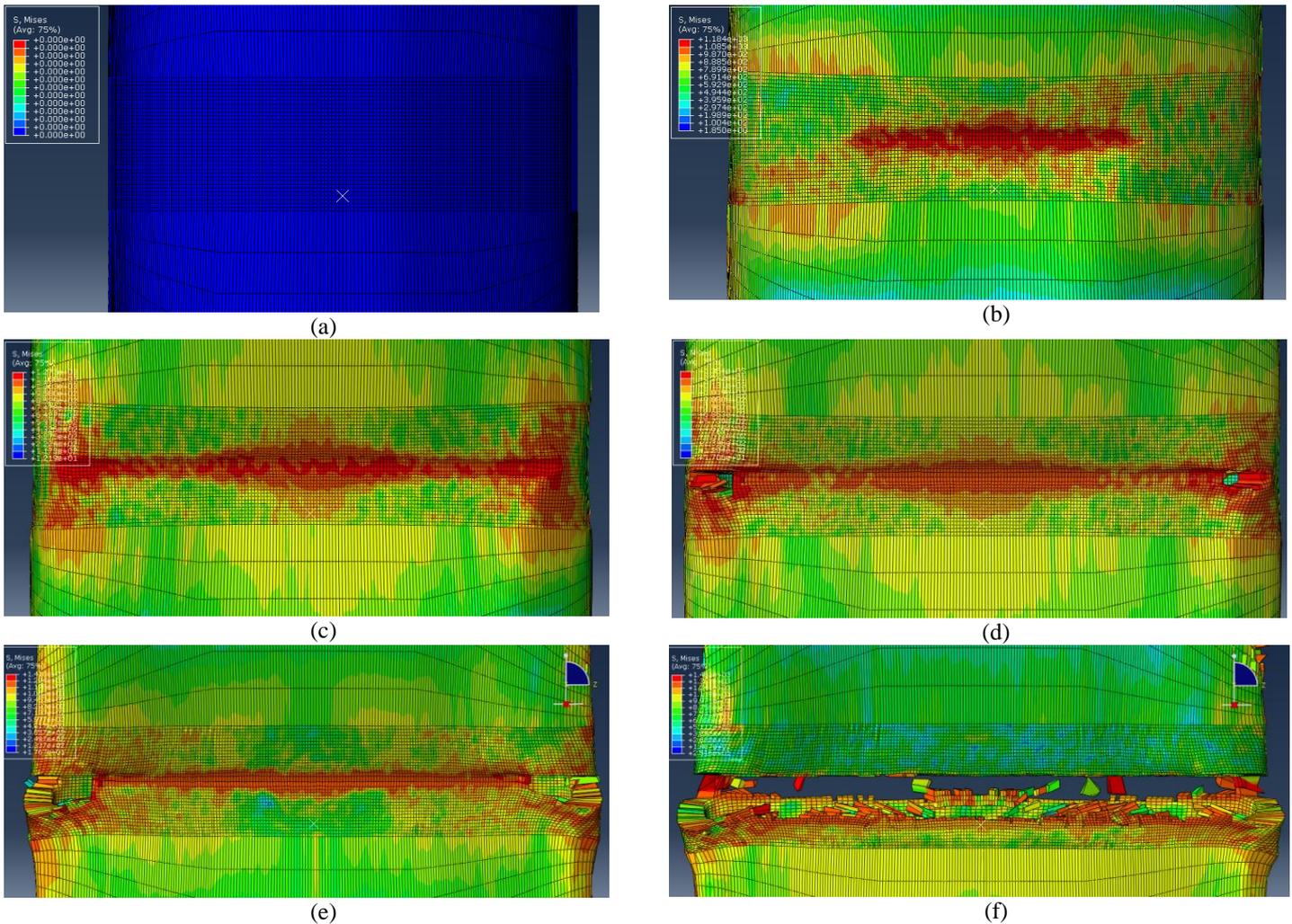


Figure 14 - Evolution of shear cut

Just as the displacement increases, it is possible to note the effectiveness of the cutting of the constituent elements of the riser. Although of having been quite refined and improved the mesh in the cut region we can see some elements that present characteristics of tension concentration and is not eliminated, this can be improved by seeking an optimization of both the geometric forms of the more homogeneous elements and the mesh. Certainly, this means a greater computational cost.

Some simplifications were adopted in the problem, such as the application of displacements instead of forces to initiate the simulations and increase the problem until a satisfactory representation of the model was obtained. The contact properties should still be intensified for more reliable results, as well as the properties of the Steel 4340.

7. CONCLUSION

Initially it is important to mention that this work provided the possibility of dealing with one of the most important and complex areas of mechanics, the computational contact is undoubtedly a field that still requires a lot of study and research. To represent the operation of a BOP valve stage in a finite element program, several parameters must be defined and inserted in the program so that the simulation can evolve and generate quality results. A very important set of data that directly influences the behavior of the material is that referring to the plastic part of the 4340 steel and the damage parameters, all based on the Johnson-Cook study. It was noticed during the evolution of the work that the components are very sensitive to the inserted material parameters and, therefore, the determination of them becomes quite relevant.

There were several attempts to structure the mesh and refinement as the results did not seem consistent, it was necessary to modify and improve the geometric configuration of the model, partitions and the contact region until satisfactory results were obtained. Certainly, the region where the contact between the ram and the riser takes place is a very delicate region, because that is where the cutting process begins, so there was a lot of effort in studying and finding discretization solutions in order to obtain a suitable behavior of the contact. The geometric shape of the contact surfaces is of fundamental importance as well as the configuration of the mesh, which implies a high computational cost. It is observed that to the extent that this contact area is more refined and geometrically homogeneous, there is greater possibility of convergence and quality results.

Even though it has not yet been possible to perform a convergence analysis, it was verified that the mesh with a greater degree of refinement presents smaller distortions in the elements and the cut of the tube manages to evolve. It is advisable to carry out such a convergence analysis to find an adequate balance between quality of result and computational cost. In addition, the evolution of the cut is still not represented in the best way represented. It was possible to observe that the elements were first deleted at the radial end of the tube and later advanced to the elements of the center of the tube. Physically the elements should be deleted right after the start when the ram's tip begins to crush the tube wall and is able to perform the cut.

The results obtained until that moment are preliminary, considering that this particular problem is treated for the first time from the point of view of computational contact, since it is a three-dimensional modeling problem, it becomes highly challenging to seek to improve modelling to achieve satisfactory results and replicate the effects of the real system. In order to improve modeling, it would be interesting, in future works, to perform other studies of mesh configurations, in the contact region, to obtain geometrically more homogeneous elements and a mesh convergence analysis, in order to found a good balance between computational cost and quality results.

8. ACKNOWLEDGEMENTS

We thank to Professor Lucival Malcher of the Department of Mechanical Engineering of Brasília University (UnB) for his instructions and for sharing his knowledge. We also thank to the Scientific Initiation Program of the Federal District University Center - UDF, for the encouragement and financial support to carry out this research.

9. REFERENCES

- Det Norske Veritas (2011). Report No. EP030842
Kikuchi, N. And Oden, J.T.(1988). Contact problems in Elasticity Studies in Applied Mathematics (SIAM).
Wriggers, P. (2006) Computational Contact Mechanics, Springer-Verlag

10. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.