COB-2023-2002
STRESS ANALYSIS VIA FINITE ELEMENTS IN STRAIGHT AND CURVED PIPELINES CONTAINING DENT DEFECTS

Maize Cibele de Lima Melo[a]
Paulo Roberto Maciel Lyra[b]
Departamento de Engenharia Mecânica. Universidade Federal de Pernambuco, Av. Acadêmico Hélio Ramos s/n: 50670-901, Recife, PE-Brazil
[a]maize.melo@ufpe.br
[b]paulo.lyra@ufpe.br

Juliana Von Schmalz Torres[c]
Núcleo de Tecnologia, Centro Acadêmico do Agreste, (NT-CAA). Universidade Federal de Pernambuco. Rodovia BR 104 KM 59 s/n: 55002-970, Caruaru, PE-Brazil
[c]juliana.von@ufpe.br

Adriano Dayvson Marques Ferreira[d]
Ramiro Brito Willmersdorf[e]
Departamento de Engenharia Mecânica. Universidade Federal de Pernambuco, Av. Acadêmico Hélio Ramos s/n: 50670-901, Recife, PE-Brazil
[d]adriano.mferreira@ufpe.br
[e]ramiro.willmersdorf@ufpe.br

Abstract. *The present work aims to evaluate how dent defects influence the structural integrity of straight and curved pipelines. Pipelines play a fundamental role in the transport of fluids over long distances and are essential to the energy industry, particularly the oil and gas industry. Most of the pipeline networks are buried and their integrity is affected by defects caused by corrosion, dents and other factors. Dent defects induce stress concentration that has been the cause of a significant number of pipeline failures (EGIG, 2018). This study proposes a computational methodology to assess the integrity of pipelines with dent defects and subjected to internal pressure. For this purpose, computational models are generated with the Finite Element Method (FEM) using Ansys software. The present work extends the approach to curved pipelines in order to evaluate the influence of the curvature on the mechanical behaviour of the pipeline with respect to the dent defect. Dent defects are generated synthetically by non-linear contact analysis using a spherical indenter with size variation. To ensure the quality of the result, self adaptive mesh refinement was used in the computational simulations, with analysis of the convergence of the results and comparison between the use of shell elements and solid elements. The results show the influence of the defect and demonstrate the extent to which the dent defects induce stress concentrations that compromise the integrity of the ducts, particularly in the case of the smaller punches and curved ducts where the defect is generated inside the curvature.*

Keywords: *pipelines, curved pipelines, dent defect, finite element method, failure pressure.*

1. INTRODUCTION

Rigid steel pipelines are considered the most efficient and economical means for transporting oil and gas, being generally built with high strength steel, however depending on service conditions defects may appear, mainly due to exposure to hostile environments (JIN et al., 2020).

The most common cause of mechanical damage and failure in pipelines is external disturbances. According to Wang et al. (2021), dents often appear in long-distance buried pipelines due to impact of landfill stones, impacts of excavation objects and local earth subsidence.

According to Torres (2016), the type and dimensions of pipelines depend on the needs and installation conditions. They can be classified according to their use and the fluid to be conveyed. The structural arrangement depends on the available infrastructure, and sometimes rigid pipelines must be able to overcome topological obstacles, with the need to perform curvature in these pipes, which are also subject to mechanical damage.

According to Cianciarullo (2022), pipelines are assets in constant growth, as technological and productive advances in oil and gas exploration drive joint efforts in research and development to avoid harmful risks to the production process, the environment and people. This requires constant evaluation of the problems that may arise.

Among the methods for evaluating engineering problems, FEM (Finite Element Method) stands out as an effective computational analysis method that is widely used to investigate the structural integrity of pipelines with defects. This

requires skilled personnel to generate models that adequately represent the geometry, loads and boundary conditions within the real configurations, and subsequently requires proper interpretation of the results obtained (SOUZA, 2008).

With this in mind, the aim of this work is to numerically model and analyse, using the finite element method, straight and curved pipelines with dents, in order to evaluate how these dents influence the integrity of the pipelines. For this, the present study counts on a comparative analysis of straight and curved pipelines with and without dents. In the analysis of the straight pipeline with a dent defect, the variation of the depth of the punching is taken into consideration. For the curved pipeline, the analysis is focused on the effect of the defect in different positions in relation to the curvature.

This paper is divided into four main parts. The first, presented so far, consists of the introductory part with a brief explanation of the subject and the objectives to be achieved. The second part deals with the computational procedure, where the use of the software is clarified, detailing how the material was specified, the type of analysis, geometry modelling, type of element used, mesh configurations, boundary conditions, contact configuration and how the results were analysed. The fourth step describes the results obtained, listed according to the analysis of operating conditions, comparison between the types of elements and analysis of critical pressure. Finally, the main conclusions of the analyses carried out are presented.

2. COMPUTATIONAL PROCEDURE

The computational procedure used to develop this study is based on numerical modelling and simulations using the Ansys 2022-R2 student version software, in three of its interfaces: Ansys Workbench, Ansys Space Claim and Ansys Mechanical. The first interface manages the subsequent ones, allowing the type of analysis to be defined and the material to be used to be configured. The second interface allows 3D or shell modelling of the required geometry. The third interface configures the mesh, inserts boundary conditions and configures the contacts for analysing the results.

2.1 Type of analysis and material specifications

The type of analysis defined for the study is static structural, as it allows the determination of the displacements, structural forces, stresses.

The specifications of the material used in all the analyses correspond to the duct model studied in the work of Pimentel (2014). It is a structural steel whose parameters are shown in Table 1, followed by the values of the stress-strain curve shown in Table 2.

Table 1. Material properties.

Elastic Modulus, MPa	205,966
Poisson's Ratio	0.3
Yield Strength, MPa	359
Ultimate Tensile Strength, MPa	472.94

Table 2. Points on the stress-strain curve.

True Strain (m m^{-1})	True Stress (MPa)
0.001743	359
0.009085	390
0.019432	425
0.045966	465
0.099845	472.94
0.199691	472.94

In the Workbench interface, a new material, different from the standard structural steel of the software, was created; the input data specified in the tables were fed through the isotropic elasticity data, yield stress, ultimate stress and multilinear isotropic hardening, in order to describe the plastic behaviour of the material composing the desired model. The definition of the use of the new material was made in the Mechanical interface, where, in addition to defining the material of the structure, the stiffness behaviour, flexible for the pipeline and rigid for the punch, is defined by selecting among the options described in the software.

2.2 Modelling geometry and element type

In this work, the pipeline geometry data also follows the dimensions of the Pimentel (2014) pipeline model, as shown in Table 3.

Table 3. Dimensions of the pipeline geometry.

External Diameter (mm)	Thickness (mm)	Length (mm)
547.6	8.1	1,500

These dimensions have been used for both straight and curved pipelines, with a 90° bend considered for the curved pipeline model.

In order to reduce the computational effort during the simulations, the geometries were modelled taking into account the symmetry of the pipe. For the straight pipeline, conditions of double symmetry were considered, so the geometry of 1/4 of the pipeline was modelled (Figure 2), and for the curved pipeline, only one symmetry was considered, with the modelling of 1/2 of the pipeline (Figure 3).

For the geometry of the indenter, a spherical punch with a diameter equivalent to 40% of the external diameter of the pipeline was chosen, based on what was adopted in the studies of Nogueira (2014) and Araújo (2015). To understand the influence of the punch size, a smaller punch was also modelled, with a diameter equivalent to 1/3 of the diameter of the previous punch (13.3% of the external diameter of the pipeline).

Both two-dimensional shell elements and solid elements were used to model the pipeline. For the analysis of this study, rectangular elements of the Shell 181 type with 4 nodes and 6 degrees of freedom per node were adopted, as it is known that these present greater savings in computational effort due to the discretization process that reduces the number of nodes, and are indicated for the analysis of large non-linear deformations. However, in order to establish a comparison between the types of elements and to verifiy if the elements of Shell 181 could be used reliably for stress analysis of the study, the simulation was carried out with Solid 187 tetrahedral elements, which has 10 nodes and 3 degrees of freedom per node, adopting 4 elements in the thickness of the channel in the defect area, in order to compare the results from the convergence analysis of the mesh with the aim of obtaining more accurate results in terms of maximum equivalent stress.

2.3 Mesh configurations, boundary conditions and contact settings

In terms of mesh configuration, the aim was to characterize a structured mesh, with local refinement control in the regions subject to higher stress values. For this purpose, quadratic elements were used and to ensure an adequate refinement in all cases, the mesh was generated using the auto adaptive configuration mode, with 3 maximum refinement loops and 3 refinement depths, followed by a convergence analysis to obtain accurate results.

The same boundary conditions and internal pressure were used to analyze the different cases, as shown in Figure 1. In this figure it is possible to visualize the symmetry conditions established by the zero displacement for the X axis (A) and for the Z axis (C), the condition to avoid movement of the rigid body established in the Y axis by the constraint (B), the pressure in the inner surface of the pipeline (D), the axial line pressure in the negative Z axis (E) and the condition of remote displacement of the indenter with a displacement of 60 mm in the negative Y axis, other displacements and zero rotations (F).
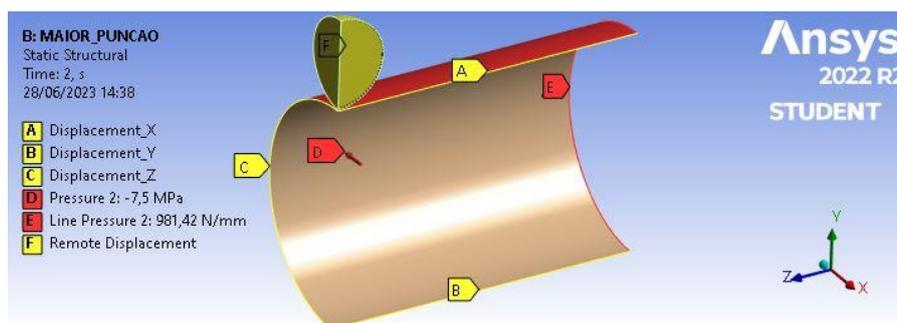

Figure 1. Boundary conditions and internal pressure.

For the cases of curved pipelines, the conditions adopted are the same, but a single symmetry condition is established (in the Y axis), the axial line pressure is applied at both ends and the displacement of the indenter, although of the same magnitude, will be different in relation to the position of the axes in each of the cases.

The axial line pressure is a tensile stress at the ends, which refers to the force that the internal pressure would cause in the p cover, and is calculated by Eq. (1) (ARAÚJO, 2015).

$$P_l = \frac{P_i \, \pi \, r_i^2}{2 \, \pi \, r_m}, \tag{1}$$

where P_i refers to the internal pressure in the pipeline, r_i to the internal radius and r_m is the mean radius.

The simulations steps are prescribed by time intervals in the software. The first analysis step uses two time steps. In the first step the internal pressure is set to 7.5 MPa, and the displacement of the punch is applied. In the second time step, the punch is pulled back, with the internal pressure kept at 7.5 MPa. The dent defects are generated under these conditions for all the cases studied.

In the following time steps, the pressure is increased in order to obtain the critical pressure for each case. The failure criterion used for these analyses is the pressure at a point reaching the ultimate stress for pipelines with defects or reaches the yield stress for intact pipelines, since in the case of an intact pipeline when the yield stress is reached the software ends the analysis without allowing further pressure increases.

In the contact analysis setup, the established contact area was the pipeline surface and the target area was the punch surface, with a prescribed punch displacement of 60 mm for both punch sizes. The contact type was assumed to be standard unilateral frictionless. Asymmetric behavior was defined as the bodies do not have the same stiffness characteristics, it is a contact between a rigid and a flexible material. The formulation of the contact analysis chosen was the augmented Lagrangian, as it is the method that generally presents a better conditioning and is less sensitive to the magnitude of the contact stiffness coefficient (ARAÚJO, 2015).

The analysis of the results takes into account the equivalent stresses (von Mises) and the total deformation for each case. To generate the results, the large deformation tool, which is a specific Ansys module, was activated so that the software performed a non-linear analysis considering large deformations and displacements.

3. RESULTS AND DISCUSSIONS

The simulations used to generate the results were performed on a computer with 8 GB of RAM and an Intel Core i7 processor with base frequency of 2.9 GHz.

3.1 Operating conditions

With the operating conditions established as described in the computational procedure, simulations were done for each of the cases studied. In Figure 2 are shown the result of the straight pipeline intact and in Figure 3 of the curved pipeline intact.
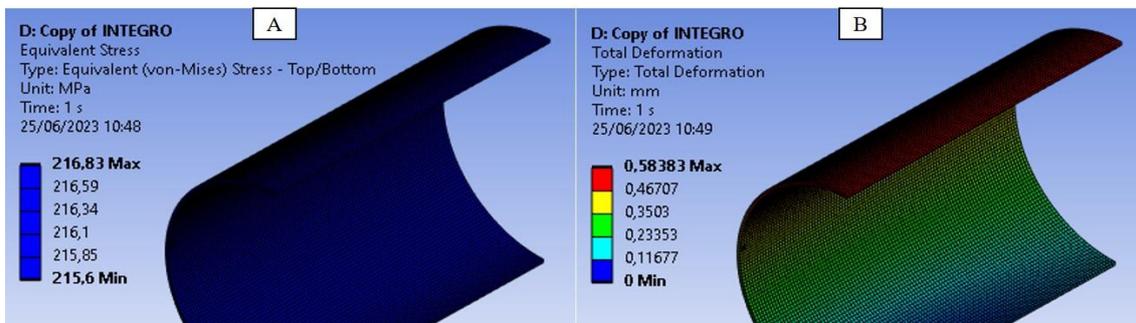
Figure 2. Intact straight pipeline with an operating load of 7.5 MPa. A) Equivalent stress. B) Total deformation.
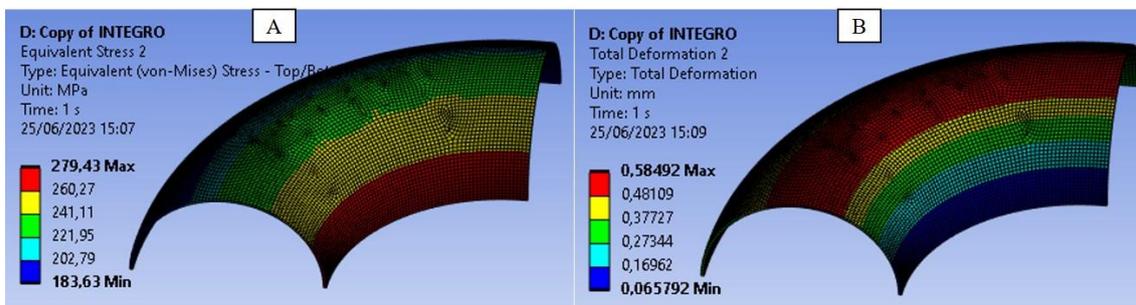
Figure 3. Intact curved pipeline with an operating load of 7.5 MPa. A) Equivalent stress. B) Total deformation.

From these figures, it is possible to see that the stress distribution along the intact straight pipeline is uniform 216.83 MPa, while in the intact curved pipeline the maximum stress of 279.43 MPa occurs in the inner part of the bend, presenting higher stress levels than the straight pipeline. For both cases the material does not reach the yield stress (359 MPa) and maximum displacement is less than 1 mm. The intact straight pipeline has of 6,375 elements in the final mesh and the intact curved duct 9,768 elements.

As for the contact analyses, Figures 4 and 5 show the von Mises stress distribution and total strain of the simulations referring to the defective straight pipeline using different punch sizes after the pull back of the punch.
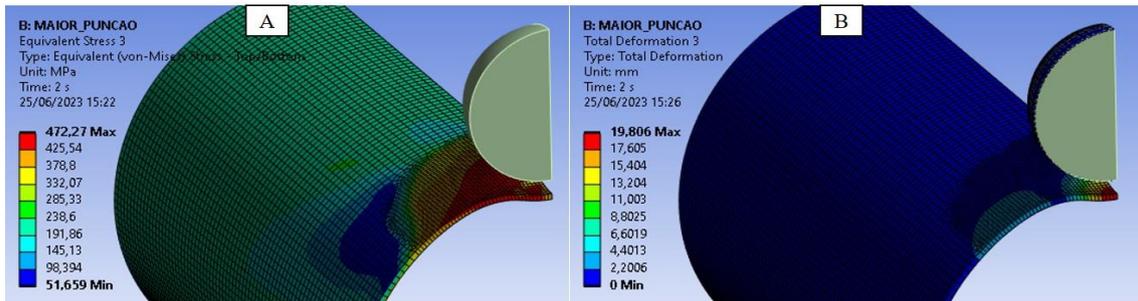

Figure 4. Defective straight pipeline caused with larger indentor, i.e. 40 % of the pipeline outer diameter.
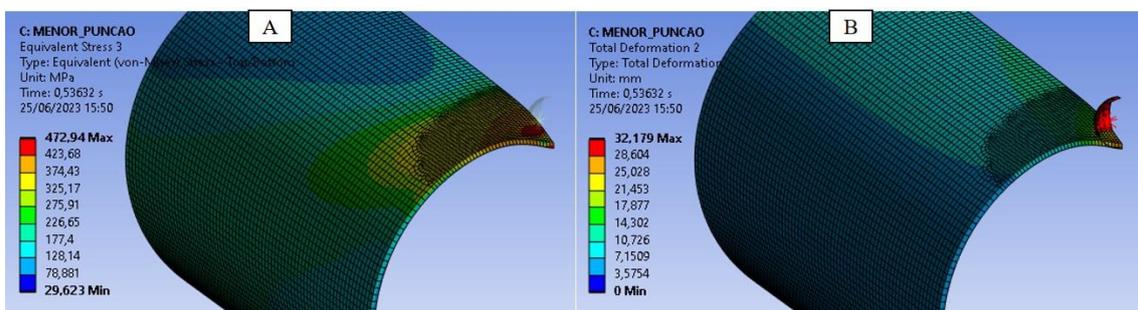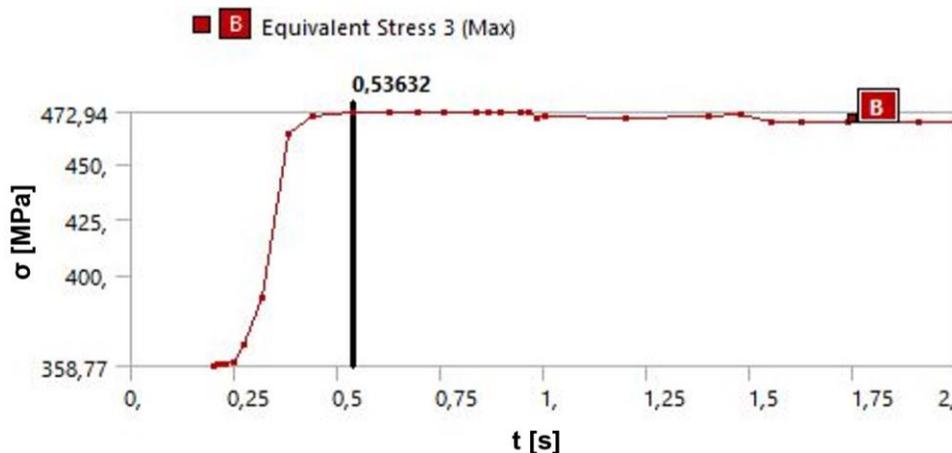A) Equivalent stress. B) Total deformation.

Figure 5. Defective straight pipeline caused with smaller indentor, i.e. 13.33 % of the pipeline outer diameter.
A) Equivalent stress. B) Total deformation.

As expected, the maximum stress and maximum displacement in both cases happen in the region of contact with the punch. In the case of the larger indenter, the pipeline reaches the yield stress and exhibits a permanent deformation. The smaller indentor, even before the total displacement time of the indentor is completed the duct reaches the yield stress of the material, showing the influence of the size of the indentor. The straight duct with defect caused with a larger indentor has 6,785 elements in the mesh and the duct with defect caused with a smaller indentor 6,430 elements.

It is worth noting that for the operating conditions, in the first time step of the simulation the displacement of the indenter is prescribed and in the second time step the pull back. Figure 5 shows the time (0.53632 s) at which the ultimate tensile strength is reached, this is also shown in Graph 1.

Graph 1. Maximum equivalent stress versus time (straight pipeline with defect caused with smaller indentor).

As the straight pipeline reaches ultimate stress with the smallest indentor size, for the analyses of the curved pipeline the defects are generated only with larger indentor, i.e. 40% of the pipeline outer diameter. In Figure 6 is shown the duct with defect in the region superior to the curvature and in Figure 7 the defect in the lateral region external to the curvature.
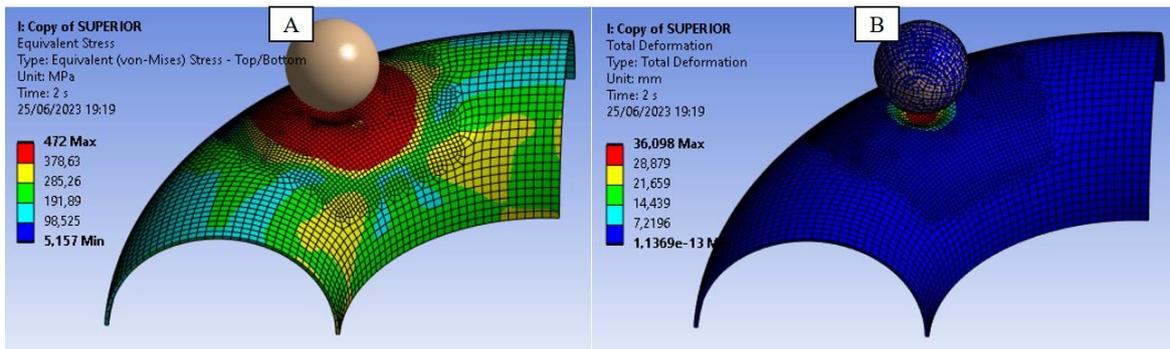
Figure 6. Curved pipeline with defect in the region to the curvature. A) Equivalent stress. B) Total deformation.
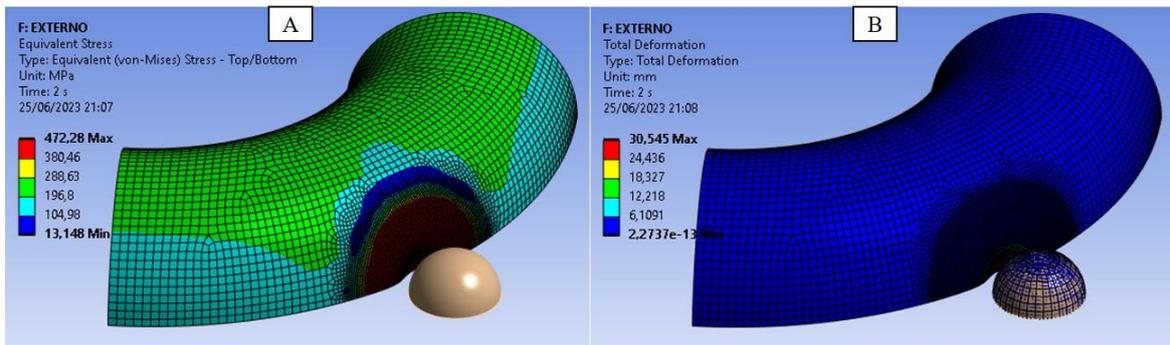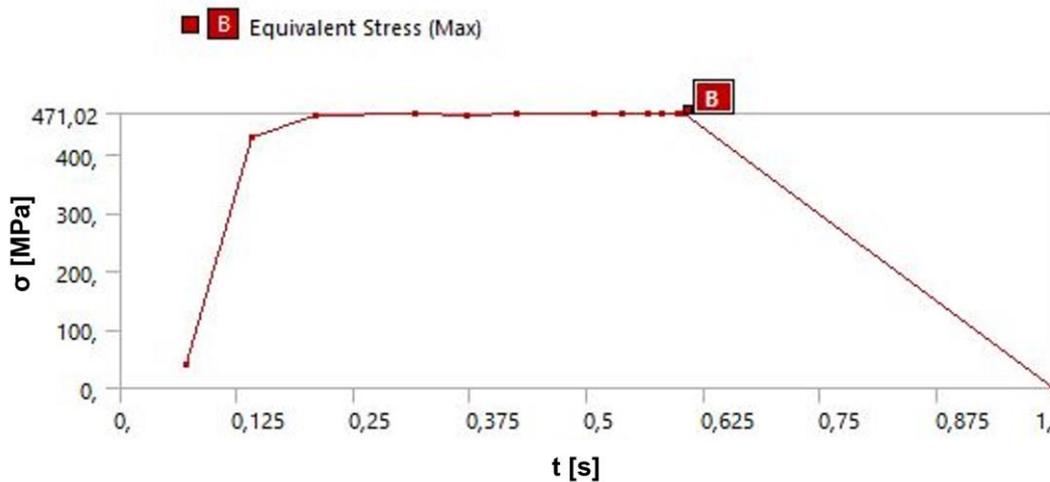

Figure 7. Curved pipeline with defect in external curvature. A) Equivalent stress. B) Total deformation.

The equivalent stress, both in the case of the defect above the bend (472 MPa) and in the external position (472.28 MPa), presents a maximum value close to that obtained in the case of the straight pipeline (472.27 MPa), but present a percentage of approximately 65 % of increase of the permanent deformation after the return of the indenter. The curved duct with superior defect presents 5,951 elements in the mesh and the curved duct with defect in the external curvature 6,283 elements.

In the case of the curved pipeline with defect in the internal curvature, the simulation indicates that during the displacement time of the indenter the duct fails, still under operating conditions. In Graph 2, it can be observed the moment at which the failure occurs, from the instant at which the curve presents a decreasing behaviour. It is worth noting the result obtained in the simulation of the intact curved pipeline (Figure 3), which shows that the maximum stress concentration is located in the inner part of the curvature. For this case, the mesh has 4,064 elements.

Graph 2. Maximum equivalent stress versus time (curved pipeline with defect in the internal curvature).

3.2 Comparison between the types of elements

For preliminary comparison between the use of shell elements and solid elements, a simulation of the equivalent stress for the case of curved pipeline with a defect in the external curvature was generated using tetrahedral solid elements with 4 elements in the thickness of the defect region (Figure 8). Figure 9 shows the graph and convergence table displayed by the software referring to this simulation, and Figure 10 shows the graph and table referring to the equivalent stress simulation using shell elements (Figure 7A).
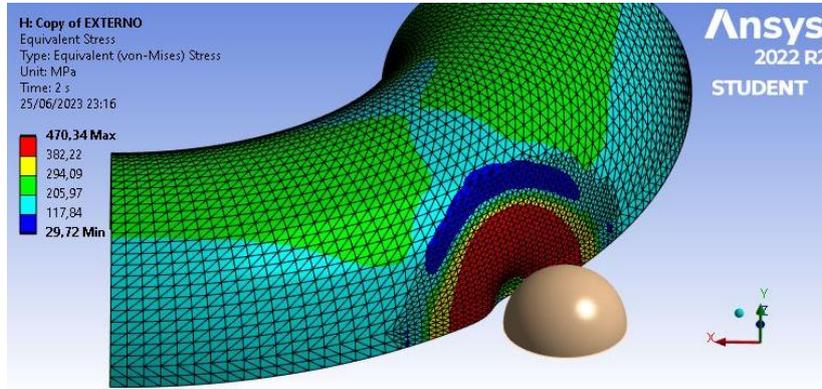

Figure 8. Curved pipeline with defect in external curvature using solid elements.
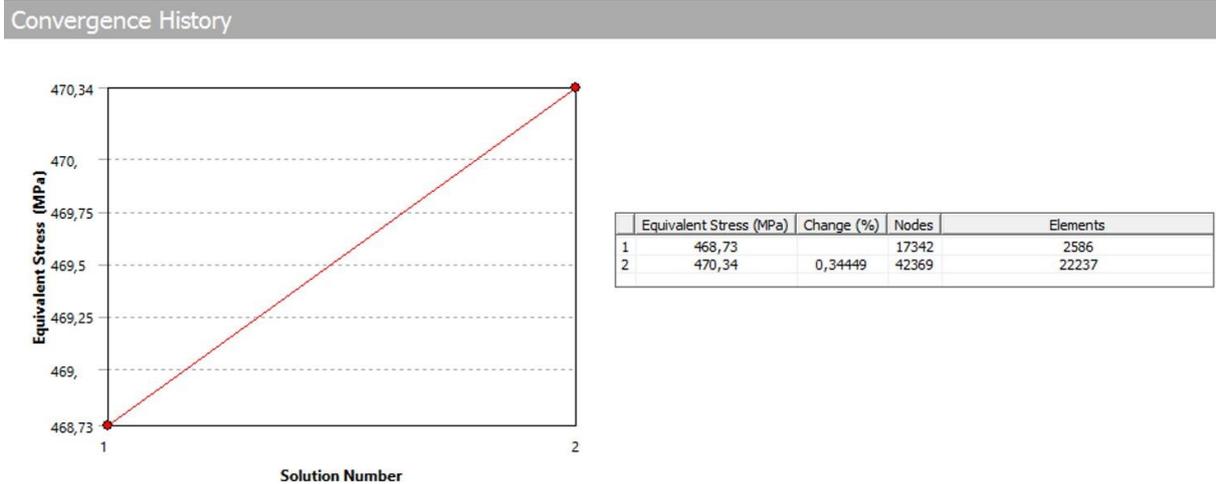

	Equivalent Stress (MPa)	Change (%)	Nodes	Elements
1	468,73		17342	2586
2	470,34	0,34449	42369	22237

Figure 9. History of convergence of maximum stress using solid elements.

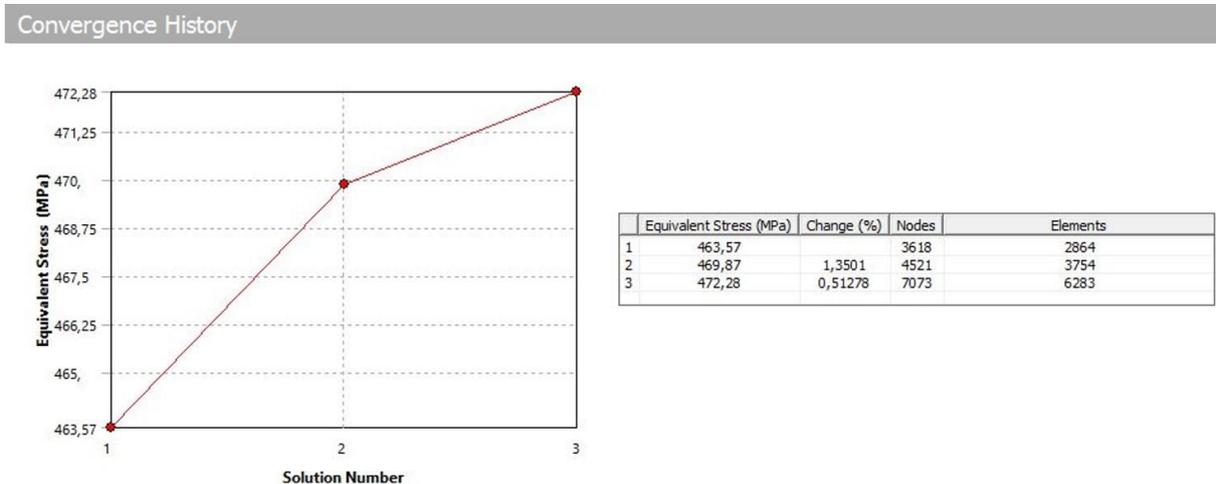|	Equivalent Stress (MPa)	Change (%)	Nodes	Elements
1	463,57		3618	2864
2	469,87	1,3501	4521	3754
3	472,28	0,51278	7073	6283

Figura 10. History of convergence of maximum stress using shell elements.

It was not possible to obtain adequate mesh convergence using solid elements, as this would exceed the limit of the number of elements allowed by the student version of the software. However, it can be seen that the use of shell elements requires a much smaller number of elements and shows convergence. The consistency between the results for the different element types shows that the use of shell elements was feasible for these analyses.

The time required to generate the results of the solid element simulation was 11 hours and 49 minutes, compared to 1 hour and 25 minutes for the shell element simulation. Therefore, it is estimated that the solid element simulations take about 8 times longer than the shell element simulations for the studies carried out.

3.3 Analysis of the critical pressure

The following results refer to the critical pressure analysis at yield stress for the intact pipelines and the ultimate stress for the faulty pipelines. Figure 11 shows the simulation results in terms of stress for the intact straight pipeline and Figure 12 shows the simulation results for the intact curved pipeline.
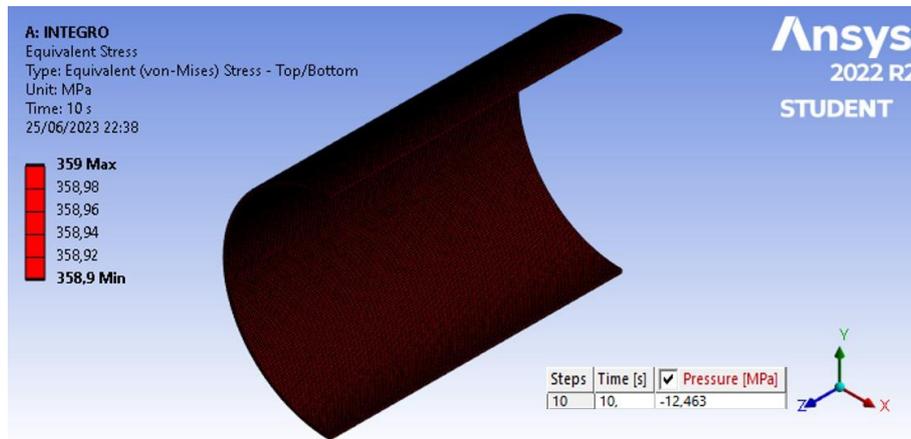

Figure 11. Equivalent stress of the intact straight pipeline with pressure increment.
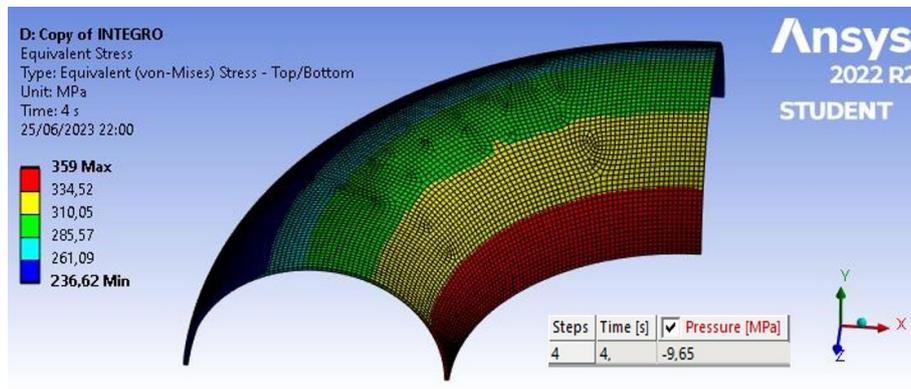

Figure 12. Equivalent stress of the intact curved pipeline with pressure increment.

With the increase of internal pressure the straight pipeline reaches the yield stress with a pressure of 12.463 MPa, consistent with the study of Araújo (2015), which analyzes this same model of pipeline intact and through analytical analysis obtains yield stress with pressure just above 12 MPa. On the other hand, the intact curved pipeline reaches the yield stress with a lower pressure than the straight pipeline, with 9.65 MPa. When the yield stress is reached, for the cases of the intact pipeline, the software does not allow further pressure increments.

For the contact cases, the yield stress was reached during the defect generation, this explains the permanent plastic deformation after the punch return, by incrementing the internal pressure the pipelines behave in order to undo the dents, but as there are residual stresses the yield stress is reached. The results of these simulations can be seen below. In Figure 13 is shown the equivalent stress of the straight pipeline with defect caused with larger indentor.
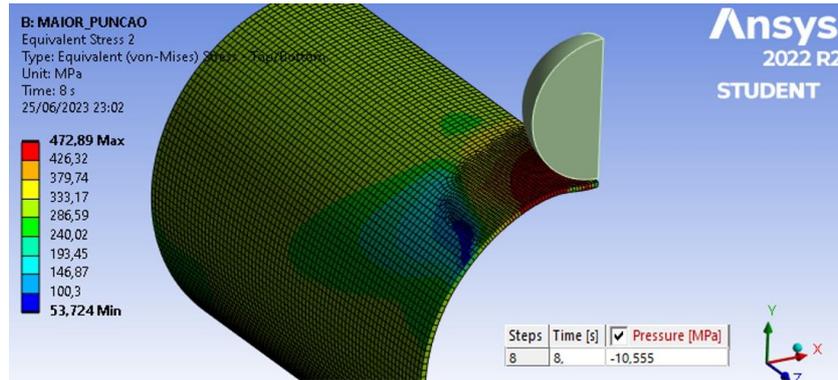
Figure 13. Equivalent stress of defective straight pipeline caused with larger indentor under pressure increment.

The critical pressure for the case of a straight pipeline with a defect caused by a larger indenter is 10.555 MPa, lower than the pressure to reach the yield stress for the same pipe without the defect, which was 12.463 MPa.
In Figures 14 and 15 show the results of the critical pressure analyses for the defective curved pipe cases.
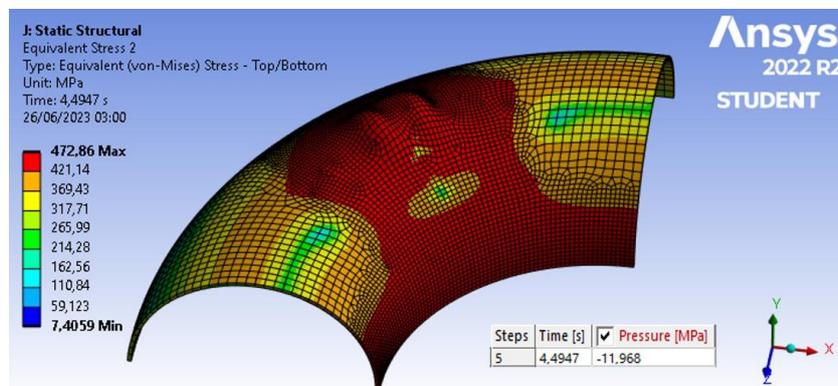

Figure 14. Equivalent stress of curved pipeline with defect higher than the curvature under pressure increment.
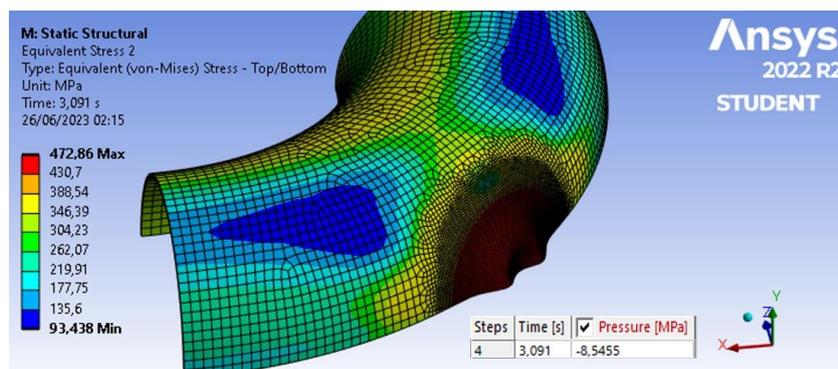

Figure 15. Equivalent stress of curved pipeline with defect in external curvature under pressure increment.

The critical pressure for the curved pipeline with defect above the curvature is 11.968 MPa, higher than the pressure value at which this same pipeline reaches the flow, which was 9.65 MPa. From Figure 14 it can be seen that the critical region extends in the region of the defect and in the region of the internal curvature, a behavior expected for failure of the intact pipeline. The pipeline with defect in the external bend has a critical pressure of 8.545 MPa, lower than the pressure to reach the yield stress for the case of this same pipeline intact.

4. CONCLUSIONS

In general, the influence of the dent defect on pipeline integrity has been observed, since this type of defect induces stress concentration, especially in curved pipelines, where the equivalent stress indices are higher compared to straight pipelines, even before the presence of the defect.

In the analysis of the operating conditions, the cases of the straight pipeline with the defect caused by the indenter of smaller size and the curved one with the defect caused inside the curvature reach the failure, reaching the fracture stress during the displacement of the punch. From this it can be concluded that the size of the indenter has an influence, the smaller contact area generating higher stresses. In addition, it is observed that the curved duct that presents a greater susceptibility to failure is the one with defect in the internal curvature, because it is the region that has a higher concentration of stress.

Still under operating conditions, it could be observed that the curved pipelines with defects that did not reach failure have maximum stresses very close to those obtained in the straight pipeline with defect caused by an indenter of the same size, but both defects in curved pipelines have greater permanent deformations.

When considering the critical pressure analysis, it is evident that the curved pipeline supports lower pressures even in the case of intact pipelines. In relation to the failure pressure for faulty pipelines, in terms of percentage, the curved pipeline that has a defect in the external curvature resists the internal pressure of 19.5 % less than the straight pipeline with an equivalent defect.

All the pipelines with defects studied, when subjected to the pressure increase, reach the ultimate stress with a pressure lower than the pressure required to reach the flow stress of the intact pipelines, with the exception of the curved pipeline with a defect higher than the curvature, which fails at a pressure close to that expected for the pipeline without defect.

Furthermore, the analysis of element types shows that models using shell elements require a mesh with a smaller number of elements to ensure convergence, resulting in an economy of computational effort, but it is important to emphasise the relevance of the analysis as in some cases shell elements cannot be used with confidence.

5. ACKNOWLEDGEMENTS

6. REFERENCES

ARAÚJO, Felipe Cardim de. Análise de tensões e integridade de dutos amassados. 2015. 60 f. TCC (Graduação) - Curso de Engenharia Mecânica, Universidade Federal de Pernambuco, Recife, 2015.

CIANCIARULLO, Mauro Iwanow. Construção de gasodutos e oleodutos: orçamento e planejamento. Rio de Janeiro: Interciência, 2022. Recurso digital.

EGIG. Gas Pipeline Incident: 10th Report of the European. Gas Pipeline Incident Data Group. 10th EGIG Report 1970-2016. March, 2018.

HUANG, Yunfei; ZHANG, Peng. Strain response analysis of API 5L X80 pipelines with a constrained dent subjected to internal pressure. Pressure Vessels and Piping, China, p. 1-13, 15 jun. 2021.

JIN, Zhi-Jiang; QIU, Chang; CHEN, Zhanfeng; YAN, Sun-Ting; SHEN, Xiao-Li. Integrity assessment of the pipelines containing an isolated corrosion pit. Engineering Failure Analysis. China, p. 1-8. 16 abr. 2020.

NOGUEIRA, Lívia Mendonça. Análise limite de dutos amassados ou corroídos. 2014. 51 f. TCC (Graduação) - Curso de Engenharia Mecânica, Universidade Federal do Rio de Janeiro, Rio de Janeiro, 2014.

PIMENTEL, Júlio Tenório. Ferramentas automáticas para análise de dutos com perfis de corrosão complexos. 2014. Dissertação (Mestrado) - Curso de Engenharia Civil, Universidade Federal de Pernambuco, Recife, 2014.

SHUAI, Yi; SHUAI, Jian; ZHANG, Xiao. Experimental and numerical investigation of the strain response of a dented API 5L X52 pipeline subjected to continuously increasing internal pressure. Natural Gas Science and Engineering, China, p. 81-92, 31 maio 2018.

SOUZA, Antônio Henrique Tenório de. Ferramentas computacionais para análise de dutos com defeitos de corrosão. 2008. 83 f. Dissertação (Mestrado) - Curso de Engenharia Mecânica, Universidade Federal de Pernambuco, Recife, 2008.

TORRES, Ayron Silva Guimarães. Soldagem de reparo em dutos de transporte pressurizados e em operação. 2016. 47 f. Monografia (Especialização) - Curso de Especialização em Soldagem, Universidade Federal de Minas Gerais, Belo Horizonte, 2016.

WANG, Junqiang; SHUAI, Yi; FENG, Can; ZHANG, Ping; WANG, Tiantian; LIN, Nan; LIN, Nan. Multi-dimensional mechanical response of multiple longitudinally aligned dents on pipelines and its. Thin-Walled Structures. China, p. 1-16. 07 jun. 2021.

7. RESPONSIBILITY NOTICE

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