ISSN 2229-5518

Predicting the Structural Response of a Corroded Pipeline Using Finite Element (FE) Analysis

DICK. I.F1; INEGIYEMIEMA.M2
(1&2) RIVERS STATE UNIVERSITY OF SCIENCE AND TECHNOLOGY, PORTHARCOURT; NIGERIA.
DEPARTMENT OF MARINE ENGINEERING
CORRESPONDING AUTHOR TEL: +234(0)7035054881
E-MAIL:dick.ibitoru@yahoo.com

Abstract: This paper presents how the response of a cylindrical pipe with an external rectangular corrosion defect under internal pressure can be predicted accurately using the finite element method. Finite element analysis is an approximate solution method to any complex engineering problem. The method involves three stages which include the pre-processing stage where the material properties of the pipe are inputted into the Finite element software to facilitate modelling. The model is then meshed after which load and boundary conditions are inputted for the solution stage. The second stage is the solution stage where the software solves the model so created. The third stage is the post-processing stage that involves the visualisation and the analysis of the results obtained. A hand calculation of the stresses is finally done using approved codes to compare with the Finite element results obtained from which judgement is made. From the result of the FE analysis, it was revealed that, though the defect area bulged with more pronounced bulging at specific nodes at the defect area, there was no leakage or rupture given the limits of the analysis. The same result was observed based on the Von Misses stress and safe operating pressure failure criteria chosen to validate the FE analysis as both criteria showed that the pipeline is safe. It is therefore safe to conclude that the pipeline can be operated safely under the applied internal pressure however, a rupture analysis is recommended to reveal the effect of bulging, particularly where stress is highest at the defect area.

Keywords: fe analysis, corrosion defect, pipeline, failure, prediction, validation, semi-empirical

1. INTRODUCTION

A network of pipelines is a common feature of the downstream sector of any oil and gas industry world over. These pipelines are designed to convey both crude and refined petroleum products from the reservoir or refineries to consumer locations safely and timely (1); (2). In-fact, they are the best means economically possible among other alternatives in this service (3). However, to safely perform this duty year-round, the integrity of the pipelines has to be assured, particularly during the operational phase. Different categories of pipeline exist such as the onshore, offshore, above-ground/surface and underground pipelines. All these categories of pipeline operate in environment that exposes the pipeline material to different spheres of surface defects, one of which is corrosion; a part-wall defect. Corrosion is the tendency of the pipeline material (steel) to return to its natural impure state and has been identified as one of the major causes of pipeline failures (4); (5). Steel pipelines, surface and underground, onshore and offshore are inevitably vulnerable to corrosion in spite of various kinds of protection because of the severity of the environmental conditions both at the surface and at depths several meters away. Corrosion may appear in different forms, such as general corrosion with the uniform loss of the wall thickness or pitting corrosion, which corresponds to the local wall thickness reduction. The effect leads to deterioration of line-pipes and endangers production, facilities and even human life when rupture develops eventually (5); (6); (7). To avoid failures therefore, corrosion has to be detected, measured and the remaining strength of the corroded region determined in order to operate the pipeline within safe margins if outright repair is not the scenario (8). Different corrosion assessment standards such as the BG/DNV, Ritchie and Last or Shell 92, DNV-RP-F101 (LPC), PCORRC (Stephens and Leis), API 1160 and ASME B13.G (9); (5) ; (10); (11) ; (8); (7) have been developed in consequence requiring pipeline operators to develop pipeline integrity management plan particularly for hazardous liquid pipelines, measuring up to 500 miles or more in high consequence areas. The ASME B13G standard specifies regulations to assess, evaluate, repair and validate, through comprehensive analysis, the integrity of hazardous liquid pipeline segments that in the event of a leak or failure, could affect populated areas, areas unusually sensitive to environmental damage and commercially navigable water ways. Most of these standards are primarily concerned with the longitudinal extent of the corroded area and internal pressure loading and employ empirical or semi-empirical approaches. While the older methods are based on the original Battelle part-wall failure criterion (the NG-18 equations), the more recent methods, DNV-RP-F101 (LPC) and PCORRC (Stephens and Leis), have partly developed from extensive numerical studies validated against test data for which reason they are adjudged the more accurate (8); (7). Numerical studies such as Finite Element (FE) analysis is an analytical method based on approximate solutions to solve any complex engineering problem by subdividing the problems into smaller, more manageable elements (12); (13). The analysis provides additional visual benefit as an accurately solved model built into the FE software can be animated to provide a visual picture of the reaction of the model under

ISSN 2229-5518

load as it would act in real life situation. Also, the maximum and minimum loads and displacements and their points of action on the model can accurately be read off from the result file, a feat that is hard to come-by using empirical or semi-empirical approaches.

In this research therefore, an FE analysis shall be performed on an externally corroded crude oil pipeline in the Niger Delta region of Nigeria where oil exploration began in the 1950s. The aim is to predict the structural strength/response of the pipeline under the influence of an internal operating pressure of 10MPa. Results from the FEA shall be validated against result from a chosen empirical method from which judgement on whether to operate or not to operate the pipeline under the given pressure shall be made based on a chosen pipeline failure criterion. In the analysis that shall follow, the entire corrosion defect area will be treated as having an approximate rectangular geometry in order to facilitate easy modelling using ANSYS; an FEA software.

2. MATERIALS AND METHOD

2.1 DEFECT PRINCIPAL DIMENSIONS:

Table 1. The details of the defective pipeline.

Yield Stress (δ_y)	464.5MPa
Ultimate Strength (δ_u)	563.8MPa
Length of the Corrosion (L_c)	90mm
Width of the Corrosion (B_c)	60mm
Depth of the Corrosion (d_c)	9mm
Internal Pressure in the Pipeline (P)	10MPa
Steel Grade	API 5L X65
External Diameter(D)	762mm
Wall Thickness of The Pipeline (t)	17.5mm
Young's Modulus (E)	210000MPa
Poisson's Ratio (ν)	0.3

.

2.2 CONSIDERATIONS IN AN FE ANALYSIS

The first consideration in any FE analysis lies not on the capabilities of the FE software program but, instead on the education, experience and professional judgement of the analyst. Only the analyst can determine what the objectives of his analysis must be. The objectives so established at the start will influence the remainder of the choices as the model is generated. A wrong choice of analysis will hamper succeeding steps and eventually the final results. Before beginning the model generation therefore, conscious efforts have to be made in order to determine how accurately the physical system can be mathematically simulated. Considerations such as the type of analysis, how much detail to include in the model; whether a full model or just a portion of the physical system is to be modelled by taking advantage of the benefits of symmetry have to be ascertained. Others include the kinds of elements to use and the density of the FE mesh. In general, the idea is to attempt to balance computational expense (CPU time, data handling capacity, etc) against accuracy.

2.3 CHOICE OF ANALYSIS TYPE

It is true that every real structure exhibits one form of nonlinearity or the other under varying conditions however; the choice of analysis shall be linear and this is informed by the properties of the defective pipeline. Firstly, Young's modulus is constant and there is no information that the pipeline is made up of components that can contact each other. Due to the ductility of pipeline material, it is also possible that it could flow and exhibit either geometric or boundary nonlinearity under the application of high range of temperature however, because there was no information about temperature application, it suffices to justifiably conduct a linear analysis.

2.4 CHOOSING A MODEL TYPE

ANSYS offers a wide range of models for different analyses. FE model therefore can be categorized as being 2-dimensional or 3-dimensional and composed of point elements, linear elements, area elements or solid elements. There could possibility be an inter-mix of different kind of elements as required to model a complex structure such as a stiffened shell structure. Since we have a corroded pipeline acted upon by tri-axial stress components, a 3-dimensional element shall be the model for this analysis. To actually model the corrosion depth and give it the required thickness and still have some ligaments left for good analysis, a solid model is equally a sure bet although, a shell element could also be used except for its thin structure which can only accommodate certain level of thickness.

2.5 CHOOSING BETWEEN LINEAR AND HIGHER ORDER ELEMENTS

The ANSYS program's element library includes two basic types of area and volume elements: linear (with or without extra shapes) and quadratic. For linear structural analysis with degenerate element shapes, that is, tri-angular 2-D elements and wedge or tetrahedral 3-D elements, the quadratic elements will usually yield better results at less expense than will the linear elements. For this analysis, a higher order version of the 3-D, 8-Node Solid 45, i.e., an isotropic 3-D, Brick-20 Node Solid is chosen. It is preferred since it can tolerate irregular shapes without much loss of accuracy. Further, it has the advantage of exhibiting compatible displacements and is well suited to model curved boundaries such as pipeline.

2.6 ASSUMPTIONS:

1. The element must not have a zero volume.
2. The element may not be twisted such that the element has two separate volumes. This occurs when the element is not numbered properly.
3. The element sizes, when degenerated, should be small in order to minimize the stress gradients.

ISSN 2229-5518

4. An edge with a removed mid-side node implies that the displacement varies linearly rather than parabolically along that edge.
5. In the creation of volumes, volumes (1) and (2) are arbitrarily assigned a z-co-ordinate of -90 to allow for easy modelling of the defect region.
6. An arbitrary angle of 80° was chosen in between, -90°, 85.4° and 90° to enable modelling of intermediary volumes.
7. In order to keep the calculation time as low as possible and still get accurate results the resolution was to take advantage of the benefit of symmetry and model only a quarter of the pipe. However, the only geometry limit, which had to be set, was the length of the examined pipe. It was decided to set it equal to 2 outside diameters of the pipe (D) for a ¼ model part; this corresponds to 4D after symmetry expansion. Connecting this parameter with the outside diameter of the pipe made it possible to exclude it from input data and to keep the model size proportional. A main for this length is for the model to be long enough to allow the stress distribution, and to prevent the model's boundary influence.

2.7 BASIC STEPS INVOLVED IN MODEL GENERATION

I. **Start ANSYS Program from the Start Menu.**
II. **Open a Folder for the model to save every action.**
III. **Set Preferences:**

ANSYS requires one to set the preferences for one's analysis. Since a structural analysis is to be run, preferences shall be set for a structural analysis and ANSYS will only make available the menu options valid for structural problems.

IV. **Definition of element type:**
V. **Specification of material properties:**
VI. **Specification of Geometry.**
VII. **Creation of Volumes:**
Making use of the benefit of symmetry, a quarter of the full pipe with the corrosion defect was modelled in six volumes, where five effective volumes represented the envisaged model and one of the volumes, precisely volume No.(3) which was subtracted in a **BOOLEAN** operation, was only created to enable the creation of the defect volume No. (4). See Volume inputs generated for the model in table 2 below:

Table 2. VOLUME IMPUT PARAMETERS GENERATED FOR THE MODEL

Volumes	WPX (mm)	WPY (mm)	Rad-1 (mm)	Theta-1 (°)	Rad-2 (mm)	Theta-2 (°)	Depth (mm)
1	0	0	381	-90	363.5	80	-90
2	0	0	381	80	363.5	90	90
3	0	0	381	85.4	363.5	90	-45
4	0	0	372	85.4	363.5	90	-45
5	0	0	381	80	363.5	90	-1434
6	0	0	381	-90	363.5	80	-1434

Steps for creating volumes:

- *Click main menu>pre-processor>modelling>create>volumes>partial cylinders>enter volume inputs from table 2 above successively in the dialogue box that pops up>ok. See volumes created below in Figs. 1 & 2.*

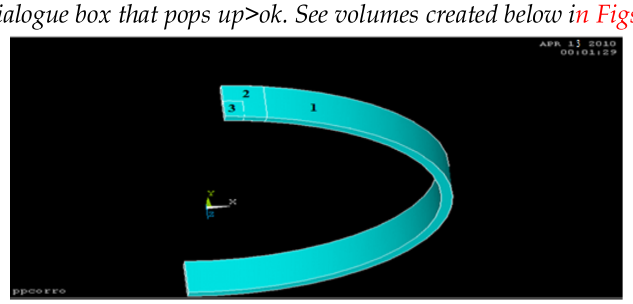


Fig.1: Volumes 1, 2 & 3

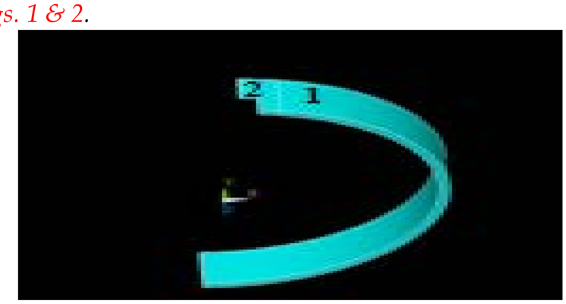


Fig. 2: Result of Boolean Operation

ISSN 2229-5518

VIII. **Booleans Subtraction operation for Volume (3). (**This operation deletes Volume (3) to make a way for volume (4)). See Fig. 2 above.

IX. **Move/Modify Operation:** This operation moves Volumes (5) and (6) by 90m towards the negative Z-axis making the full quarter length to be 1524m. Notice that because volumes (5) and (6) were modelled from point (X,Y,Z), (0,0,0) and these volumes were supposed to have taken off from point 90m, which is the Z-offset or depth of volumes (1) and (2), the pipe length, 1524mm, was deliberately reduced by 90m, i.e. made to 1434mm for these volumes so as to compensate for envisaged **overlap** which is corrected using the" **Move/Modify Operation**" to bring the pipe length to 1524mm, i.e. the model length assumed to be 2xNominal Diameter of pipe. See Figs. 3 and 4 below.

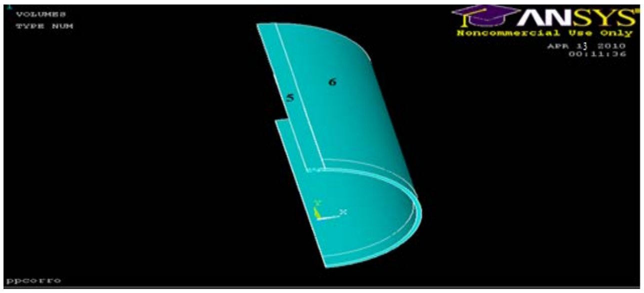


Fig.3: Overlapped geometry

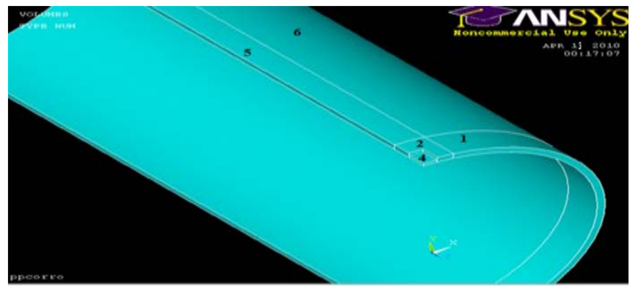


Fig.4: Moved/Modified geometry

X. **Gluing Operation:** The five discrete volumes so formed to make up the quarter pipe geometry are disjointed therefore, to get rid of incidences of double lines or areas or volumes at a particular boundary a "Gluing" operation must be carried out to bind all adjoining boundaries to one common boundary.

XI. **Meshing:** Three steps were used to accurately mesh the model namely, (1) setting the mesh attribute; (2) setting the mesh controls, which has many options to choose from and (3) generating the mesh.

- **Assigning of mesh attribute:** Assigning the element attributes to the solid model entities allows one to pre-assign attributes for each region of the model. By using this method, one can avoid having to reset attributes in the middle of meshing operations.
- **Setting of mesh controls:** Mesh controls allows one to establish such factors as the element shape, mid-side node placement, and element size to be used in meshing the solid model. This step is one of the most important in any analysis, for the decisions made at this stage in the model development will profoundly affect the accuracy and economy of the analysis.
- **Element Shape:** Allowable element shapes were set in line with the set attributes bearing in mind the desired element shape and the dimension of the model to be meshed. Volume elements can often be either hexahedral (brick) or tetrahedral shaped. In addition to specifying element shape, the type of meshing for the model was specified. Here, a mapped mesh was specified.
- **Line divisions:** The lines were divided as shown on the model that follows. Because our interest is to investigate the effect of the internal pressure on the corroded region, more line divisions are given to this region to have a dense but not too dense mesh. Note that line division is done in the order (X,Y,Z) simultaneously and done to maintain an aspect ratio of not more than 2 for better and faster solution).
- **Meshing the Solid Model:** Once the element attributes and meshing controls have been set then, the finite element mesh is ready to be generated. First, however, it is usually good practice to save one's model before initiating the mesh generating command to have a possible return point if, error arises.
- **Generating the Mesh:** To mesh the model, the meshing operation that is appropriate for the entity type that is required for element shape was used.
- **Concatenation Operation:** Aware of the requirements for mapped meshing and volume sweep, Volume (2) was concatenated by Areas, which had more than 6 sides to conform to the geometry requirement for Mapped meshing.
- **Volume Sweep Meshing Operation for Volume (2):** Because volume sweep operation is applicable to volumes that either does not contain a hole in a side area or internal void, the volume sweep meshing operation was chosen for volume (2), itself serving as the target volume whereas Volume (4) functioned as the source volume. See completely meshed model in Fig.5 below.

ISSN 2229-5518

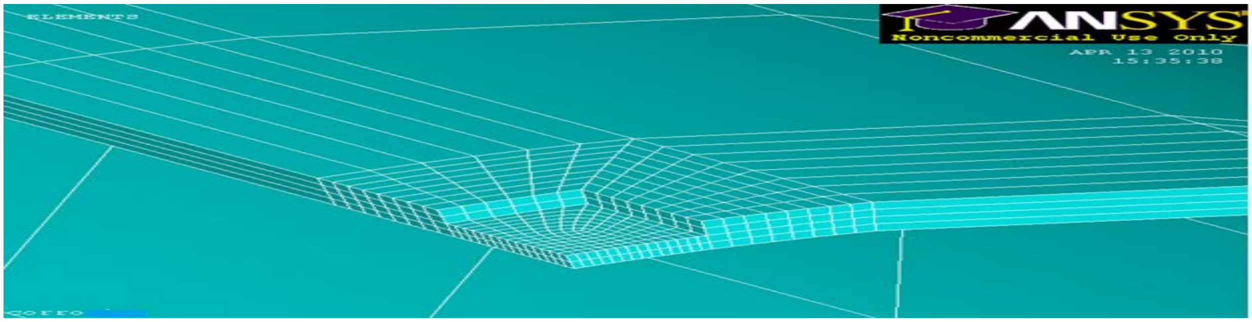


Fig. 5: COMPLETELY MESHED MODEL

XII. **Application of Boundary conditions: Constraints and symmetry:** Consideration made for the boundary condition is that while the bottom is stably fixed to the ground, the ground acting as a resistance to any deformation, the same is not true at the top where the back-fill is assumed not sufficient to resist the resultant vertical pressure from the pipe. Hence, the top end of the symmetry XZ planes was not constrained from displacement along any of the axes while the bottom was constrained from displacement on its lower line along the vertical y-axis (UY=0) since only the outer surface of the pipe is in contact with the ground..

Two sets of constraints were applied; (1) Restraining displacement in the global axial Z-direction (UZ=0) and allowing displacements in both the global radial, X and global circumferential, Y- directions and (2) Restraining displacement in the global circumferential Y-direction (UY=0) and allowing displacements in both the global radial, X and global axial, Z-directions.

XIII. **Application of loads:** Generally speaking, FEM is based on approximations. As model geometry approximates the real shape and constraints approximates how the structure is supported similarly, loads approximate what happens in the real world. Considering the parameters given for this analysis, only one type of load was applied namely; **Internal pressure of 10MPa**.

Loads are applied in the numerical model over a surface as surface loads and it is possible to apply them simultaneously. However, resort was made to an operation that saw the application of the pressure once on all surfaces by specifying an area for the application of the pressure.

- **Steps for the application of load:** Specification of target area on the model – this is necessary in order to apply load at target destination i.e., at internal surface of the quarter pipe made up of Five volumes, each volume having the same effect of pressure. See figure 6 below showing application of boundary conditions and pressure load (pressure load is shown as red arrows).

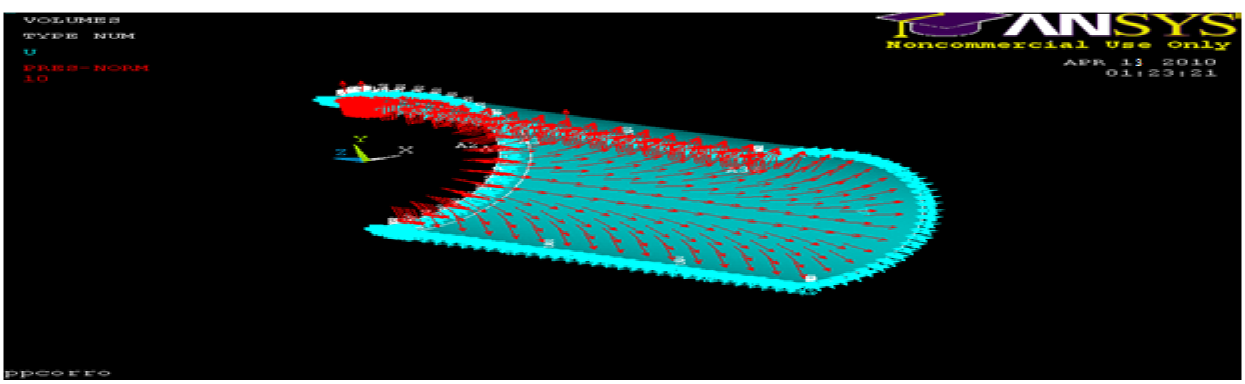


Fig. 6: Boundary conditions and pressure load on the model.

XIV. Solving the model: At this stage ANSYS solves the model in line with the applied boundary conditions on the model. See Fig.7 below showing the solved model ready for further analysis (Post-processing).

ISSN 2229-5518

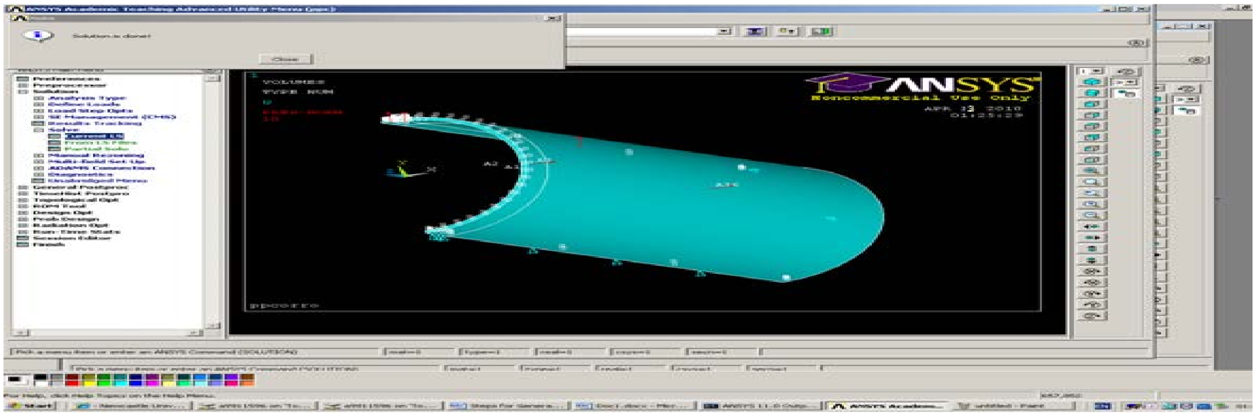


Fig. 7: Solved model.

3. RESULT PRESENTATION

3.1 POST-PROCESSING OF THE RESULTS. This stage involves the plot of all required outputs for the analysis. It consists of a whole lot of activities where contour plots of element and nodal displacements, stresses, etc, depending on the objective of the analysis, can be carried out for visual insight. Displacement and stress plots on graphs are equally done.

3.2 MESH AND ITS ACCURACY:

The starting point of the finite element method is subdivision. The body has to be subdivided into a finite number of smaller pieces which are called elements. These elements are defined by points at their edges called nodes. Nodes and elements together form FEM mesh, which approximates the shape of the real body. The coarser it is, the more simplified the body is and the results less accurate. A fine mesh gives results that are closer to the exact solution, but the analysis is more time consuming.

There are two different ways in which a model in ANSYS can be created: top-down solid modelling and bottom-up generation. In the first, the geometric model shape is produced with points, lines, areas and volumes. After that, the mesh is automatically generated according to the set up mesh controls. This way is very convenient, but, at the time of developing this procedure, impossible to use. It was determined that the calculating capacities were too low even to generate some more complicated shapes of corrosion defects inscribed into an oval pipe. Therefore, it was necessary to follow a so-called bottom-up generation way, in which the location of every node is defined, as well as the shape and size of the elements.

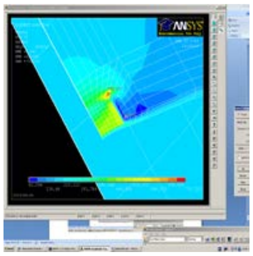
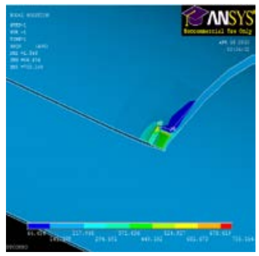
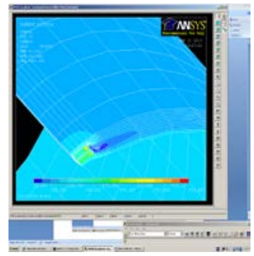
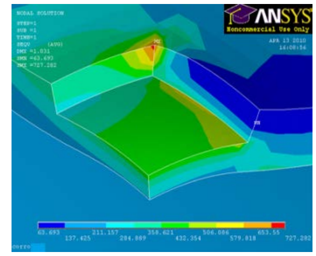
3.3 MESH SENSITIVITY (SOLUTION CONVERGENCE).

Four differently meshed models were used to conduct the sensitivity analysis as shown in table 3.

Table 3: Mesh Sensitivity Analysis using four differently meshed models.

	FIRST PAIR		SECOND PAIR	
DEFECT AREA	MODEL 1 (NO. OF DIVISIONS)	MODEL 2 (NO. OF DIVISIONS)	MODEL 3 (NO. OF DIVISIONS)	MODEL 4 (NO. OF DIVISIONS)
LENGTH	6	6	8	8
WIDTH	6	6	6	6
DEPTH	2	2	2	2
TOTAL ELEMENTS IN DEFECT AREA	72	216 (after resetting mesh attribute)	96	294 (after resetting mesh attribute)
VON MISES EQUIVALENT STRESS(MPa)	751.2	755.21	913.7	727.3

ISSN 2229-5518

MODEL SNAPSHOTS IN CONTOUR PLOTS				

Four models were used for the Mesh sensitivity analysis because of the peculiar similarities they share in terms of element division on the defect area and the corresponding Von Misses stresses obtained. For effective comparison, these four models were grouped in two pairs, where models in each pair have equal element divisions on the defect zone, see Table 3 above. However, the number of elements/mesh density at the defect area in each pair was varied by resetting the mesh attribute in order to study their effect on the analysis. As shown on Table 3, resetting the mesh attribute changed the total number of elements from 72 to 216 and the equivalent von misses stress from 751.2MPa to 755.21MPa after analysis for the first pair of models. Similarly, resetting the mesh attribute changed the total number of elements from 96 to 294 and the equivalent von misses stress from 913.7MPa to 727.3MPa after analysis for the second pair of models.

A quick consideration of these results show that the value of the Von Misses stress for the models of the first pair compare favourably well with each other with model-2 having an incremental value of about 0.53% of model-1. One would have just concluded then, that the mesh quality for model-2 of the first pair is good enough since it has almost the same Von Misses stress value as model-1 with better mesh refinement of about 200% over that for model-1. However, result from model-2 of the second pair show that the Von Misses stress could further be stepped down with further mesh refinement, i.e. from 216 to 294 elements at the defect area leading to Von Misses stress reduction from 755.2 to 727.3 MPA. Of course, the target in this analysis is to achieve an equivalent Von Misses stress that must not be more than the material strength properties. This is to agree with results from semi-empirical calculation which revealed that the defective pipe should be safe, operating at a pressure of 10MPa (see Table 4 below). Hence, with an element increase of 36.1% above model-2 of the first pair, representing about 308.3% above model-1 of the first pair, there was a stress decrease of about -3.7% below that for model-2 and about -3.2% below that for model-1 of the first pair. Obviously, model-1 of the second pair is out of consideration since it has a stress increase of about 21% above that for model-2 with a decrease in element of about -55.6% below that for model-2, all of the first pair. Further sensitivity studies with higher elements gave increasingly higher values of Von Misses stress therefore, model-2 of the second pair was chosen as the candidate mesh for the model. The mesh density generally, is more at the defect region than other regions of the model because it is the point of examination, although too much mesh.

3.4 FAILURE PREDICTION:

Two sets of failure criteria were used for the analysis namely:

(a) THE VON MISSES CRITERIA: - Pipeline steel material is ductile and operates in environment where ductile failure occurs for which reason several failure theories and failure criteria have been developed to describe the failure mode. For corroded pipes however, two of these are commonly used: maximum shear stress theory (Tresca) in which failure occurs when the maximum shear stress equals to the critical shear stress, and maximum distortion energy theory (Von Misses) in which a three-dimensional stress is compared with an effective stress. Although, the difference between both criteria only becomes more significant after leaving the elastic range and taking into consideration the hardening behaviour of the material, the choice of which one to select was simple, as ANSYS uses only Von Misses criterion.

For pipe calculation it is more convenient to use this theory where the three principal stress components acting along the axes of the pipe are combined into one effective/equivalent stress according to the following equation:

$$\sigma EQ = \left(\frac{1}{\sqrt{2}}\right)\sqrt{[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2]} \qquad 1$$

Where;

ISSN 2229-5518

$\sigma_1 = axial\ stress$, acting along the longitudinal direction

$\sigma_2 = hoop\ stress,$ acting along the circumferential/tangential direction

$\sigma_3 = radial\ stress$, acting along the radial direction and always taken as the negative of the internal pressure without any loss of accuracy.

$\sigma EQ = \boldsymbol{Equivalent\ Von\ Misses\ Stress}$;

According to this criterion, failure is said to have occurred if the equivalent Von Misses stress is more than the material strength property of the pipeline.

(b) **COMPARING THE PIPELINE INTERNAL PRESSURE AND THE SAFE OPERATING PRESSURE (S.O.P)**.

According to this criterion, the pipeline is said to have failed if the safe operating pressure calculated using the failure pressure as a function of the pipeline design factor is less than the pipeline internal pressure. Recalling that a comparative studies conducted by (11) for all the semi-empirical methods adjudged PCORRC and DnV methods as the most accurate in predicting failure pressure; this analysis shall adopt the DnV method as the standard for comparison since the investigation satisfies its requirement of diameter to thickness ratio and steel grade. In Table 4 below, a summary of results from chosen semi-empirical methods is shown and a sample calculation is shown at the Appendix.

4. ANSYS RESULT FILE INTERPRETATION

From the result file of the model with the chosen mesh density, it was found that the model is composed of a total of 7838 nodes on 1478 elements; **tables (7) and (8)** on Appendix, refers. While the last node-7838 lies on coordinates (XYZ) (5.3503, 367.84,-233.40) with zero average thermal strains, the first node-1 is lying on coordinates (XYZ) (29.834, 370.80, -45) with equally zero average thermal strains since there was no thermal input. Element 1478 is bounded by four face nodes where the surface pressure of 10MPa was applied namely, 6952, 7108, 2380, 2063. From **Table 6** on the Appendix, it could be observed that a maximum surface Von Misses equivalent stress of 408.07 MPa was exerted on element number 390 implying that failure will start from this element in any eventuality. The least affected element in this regard according to this table is element number 518 with a stress value of 79.051MPa. The three surface principal stresses acting along the pipe are designated by S1= 477.57MPa, S2= 144.74MPa and S3 = 61.943MPa where, S1 is the Hoop or circumferential stress, S2 is the axial of longitudinal stress and S3 is the radial stress which in most cases is negligible or taken as the negative of the pipe internal pressure in pipeline failure criteria assessment using Von Misses equivalent stress. The element with the highest circumferential stress is element number 43 with a value of 477.57MPa, element number 518 being the least with a stress value of 77.227MPa.

From **Table 5**, it's found that the maximum displacement was in the circumferential Y-direction with a value of 1.0303mm. This amount of displacement was suffered by element number 190. The minimum displacement was along the axial Z-direction with a value of -0.26662E-01 on element number 310.

4.1 DISCUSSION OF THE RESULT:

Expectedly, the maximum displacement of 1.0303m was observed at the defect zone on the displacement contour plot where there is a reduced pipe ligament due to the corrosion. The position of maximum displacement on element 190 is observed to be at the tip of the corroded region where there is least support from neighbouring ligaments. Conspicuously, this point is seen as the highest vertically displaced point (Fig. 8) below. It however, diverges towards the centre of the defect area as could be seen as a radiating **red curve** from the tip of the defect area on the contour plot (Fig. 9) below:

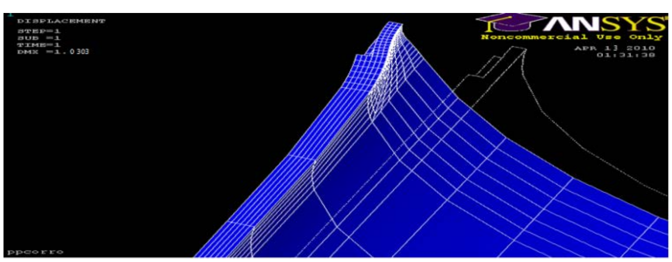


Fig. 8: **Displacement plot of defect zone**

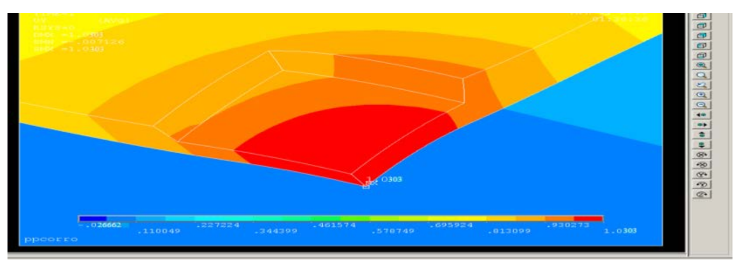


Fig. 9: Contour plot of Displacement at defect zone

Because this point suffers the highest displacement, the bulging effect as a result of the internal pressure is highest at this point. The bulging effect gradually increases from the region of least vertical displacement and becomes highest at the point of highest vertical displacement (the tip of the defect) (see Fig. 10 below). Expectedly therefore, the farthest point from the tip of the corrosion was observed to suffer the highest stress. This part has the highest stress intensity and Von Misses stress since it acts as a support/hinge to resist the upward effect of the internal pressure around the defect zone (the most affected zone because of reduced pipe ligament); the highest being at the tip.

Because of the geometry of the defect, rectangular in shape, the maximum values and positions of stress intensity and

Von Misses stress were observed at the vertex that connects two sides of the defect rectangle that forms boundaries with volume (2). Singular points are known to be areas of high stress concentration as could be revealed on the contour plot on Fig.11 below. The plot shows increasing change in the magnitude of the stress from regions farther away from this point of acuity up to the highest value at the sharpest point itself. It shows therefore that the profile of the defect has some relationships with the stress concentration.

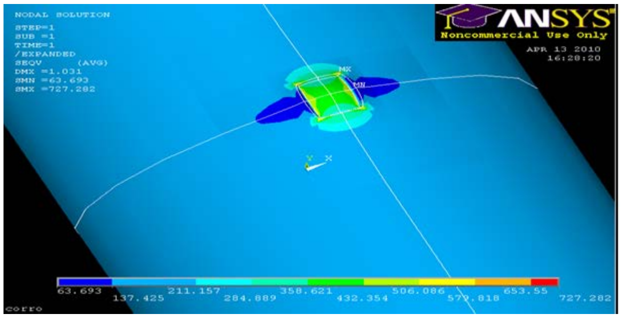


Fig. 10: Full Pipe Section of Defect Size

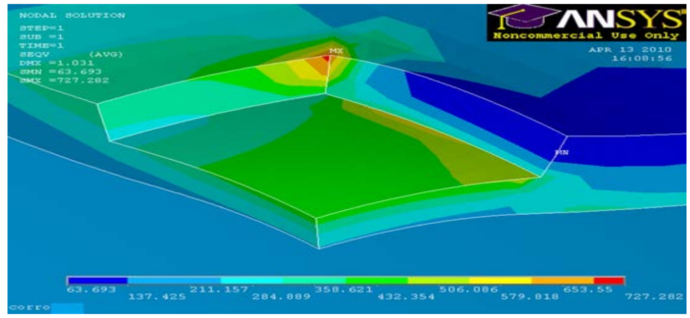


Fig. 11: Von Misses Equivalent Stress on a Contour Plot

The maximum Nodal Von Misses stress was 727.282MPa for the model and it was observed at this point. However, the maximum average Von Misses stress on an element plot was observed to be 398.119MPa and found at some distances away from this point but still along this line. The same features could be observed for the Stress intensity values on a contour plot.

Comparing results from the FE analysis (Table 6 on the Appendix) with the Material strength properties of the pipeline show that at some regions of the defective pipeline, the Von Misses stress is higher, particularly at localised nodes or elements, than the material strength which means that the defect could fail the pipeline. However, because stress is redistributed, the effect is reduced and so evens out eventually. To support this, an observation of both the Average and Non-average element Von Misses stresses show that these stresses are well below the material strength properties indicating that there is neither local nor global element failure due to the effect of the internal pressure on an average/non-average criterion. Again, a comparison of the equivalent Von Misses stress (381.0361MPa), calculated with equation 1 at the Appendix using the three surface principal stresses from the result file of the FE analysis (Table 6 on Appendix), with the strength properties of the pipeline equally show that the defective pipeline is safe. This point is buttressed further by the safe operating pressures and stresses computed using the semi-empirical approaches where the pipeline is adjudged safe as shown on Table 4 below. According to the safe operating pressures calculated, the pipeline operating at 10MPa has some margin of safety within the range of 41 to 58%.

Notwithstanding, because the defect on the pipeline would not allow the stress to be redistributed evenly, since they are points of stress concentration, the stress distribution will be skewed towards this region. As a result, the defect region may be considered to suffer local plasticity because of the ductility of the pipe material at nodes or points where stress is perceived to be highest as can be seen as peak points on the graph plot of nodal distribution around maximum Von Misses stress in Fig. 12 below. Again, a view on Figure 10 above would reveal this feature as the pipe is seen bulged at the defect region, with the tip region most bulged. However, since the extent of plasticity cannot be revealed given the limits of the analysis conducted, it would only be pre-mature to conclude if the local plasticity failed the pipeline or not as only a failure/burst analysis could reveal this feature. It is therefore recommended that a burst analysis be conducted to see the effect of the plasticity

.

Table 4: SUMMARY OF RESULTS FROM CHOSEN SEMI-EMPIRICAL METHODS (see sample calculation on Appendix).

SEMI-EMPIRICAL METHODS	FAILURE STRESS , (δf) , (MPa)	FAILURE PRESSURE, (Pf) , (MPa)	S.O.P(*safe operating pressure*) @ DESIGN FACTOR OF 0.72 (MPa)	FAILURE DECISION: DEFECT WILL FAIL PIPELINE IF *M.O.P>S.O.P*	STANDARD FAILURE EQUATIONS
ASME B31.G CRITERION	450.7283	20.70274	14.90598	**SAFE**	$Failure\ stress = 1.1(SMYS)[\frac{1-(\frac{2d}{3t})}{1-(\frac{2d}{3t})M^{-1}}]$;

ISSN 2229-5518

						$Folias\ factor, \quad M = \sqrt{[1+\frac{0.8L^2}{Dt}]}$ $Failure\ pressure = \frac{failure\ stress*2*pipe\ thickness}{pipe\ external\ diameter};$ $Safe\ Operating\ Pressure = failure\ pressure * desig\ factor$
	RSTRENG OR MODIFIED ASME B31.G CRITERION	463.6117	21.2946	15.33204	**SAFE**	$Failure\ stress = flow\ stress \left[\frac{1-\left(0.85\frac{d}{t}\right)}{1-\left(0.85\frac{d}{t}\right)M^{-1}}\right];$ $Folias\ factor,$ $M = \sqrt{\left[1+0.6275\left(\frac{L^2}{Dt}\right)-0.003375\left(\frac{L^2}{Dt}\right)^2\right]}$ $for;\ \frac{L^2}{Dt} \leq 50\ ,$ $Failure\ pressure = \frac{failure\ stress*2*pipe\ thickness}{pipe\ external\ diameter}$ $Safe\ Operating\ Pressure = failure\ pressure * desig\ factor$
	RITCHIE and LAST or SHELL 92 CRITERION	426.3256	19.58244	14.09896	**SAFE**	$Failure\ stress = 0.9\ (UTS)[\frac{1-\frac{d}{t}}{1-\left(\frac{d}{t}\right)M^{-1}}]\ ;$ $Folias\ factor,\ M = \sqrt{[1+\frac{0.8L^2}{Dt}]}\ ;$ $Failure\ pressure = \frac{failure\ stress*2*pipe\ thickness}{pipe\ external\ diameter}$ $Safe\ Operating\ Pressure = failure\ pressure * desig\ factor$
	BG/DNV LEVEL 1 CRITERION	518.433	24.37227	15.7933	**SAFE**	$Failure\ stress = UTS[\frac{1-\frac{d}{t}}{1-\left(\frac{d}{t}\right)Q^{-1}}]$; $Folias\ factor, Q = \sqrt{[1+0.31(\frac{L^2}{Dt})]}$ $Failure\ pressure = \frac{failure\ stress*2*pipe\ thickness}{(external\ diameter-thickness)}$

						$Safe\ Operating\ Pressure$ $= failure\ pressure$ $* total\ Usage\ factor$

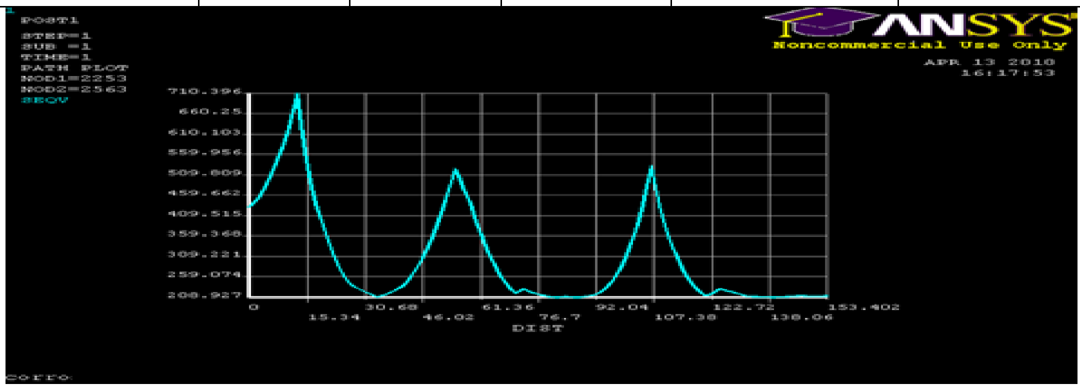


Fig. 12: Graph plot of Von Misses Stress distribution on selected nodes around the defect zone

CONCLUSION

Although, most of the semi-empirical methods are conservative, the relationship provided by BG/DNV is comparatively adjudged most accurate and produces results that compares safe with numerical analysis hence, one can safely conclude that based on the results achieved on the extent of analysis versus calculated safe operating pressures and equivalent Von Misses stress, the pipeline will remain safe at the operating pressure of 10MPa except other operational conditions changed. As noted earlier, the defect region may suffer local plasticity at nodes or points where stress is perceived to be highest however; judgement to the extent of concluding whether the plasticity led to leakage and eventually failed the pipeline through rupture is out of the scope of this analysis and could be achieved by running a rupture/burst analysis. Due to incidences of overpressures that is synonymous with pipeline operations, coupled with failure of pressure relief valves along the pipeline which is inevitable, it is recommended that the defective pipeline be repaired and not operated for too long in order not to fail but assure the integrity of the pipeline.

ACKNOWLEDGEMENT

The authors wish to acknowledge all that have contributed to the success of this report. Particular in this regard is the Niger Delta petroleum company that provided resources for the analysis. We are indeed also grateful to the typist that typed the original manuscript. Finally, we thank the Almighty God for the wisdom and enablement He granted us throughout this research period and of course, not forgetting the understanding of our individual family members.

REFERENCES

1. **Mohinder, L and Nayyer, P.E.** *Piping Handbook.* USA : Mc Graw-Hill, Inc, 1992.

2. **Micheal, F.** *Facility piping systems handbook, 2nd edition.* USA : Mc Graw-Hill, 2002.

3. **Hopkins, Phil.** Fundamentals of Pipeline Engineering. *Pipeline Engineering.* Newcastle University upon Tyne : Penspen Integrity, 2009/2010, pp. 9-10. M.Sc. Lecture Note.

4. *Best Practice in Pipeline Defect Assessment.* **Cosham, Andrew and Kirwood, Mike.** Calgary, Alberta, Canada : Andrew Palmer and Associates (Penspen Group), 2000. International Pipeline Conference. pp. 1-12.

5. *Best practice for the assessment of defects in pipeline-corrosion.* **Cosham, Andrew, Hopkins, Phil and Macdonald, K.A.** s.l. : Science Direct, November, 2006, Vol. 14, pp. 1-21. 10.1016.

6. **Cosham, Andrew and Boreas, Atkins.** Fundamentals of Defect Behaviour. *Pipeline Engineering.* Newcastle upon Tyne : Penspen, 2009/2010, pp. 30-46. M.Sc. Lecture Note.

7. **Szary, Tomasz, Geowissenschaften, Fakultät für and Bergbau, Geotechnik und.** *The Finite Element Method Analysis for Assessing the remaining strength of Corroded Oil Field Casing and Tubing.* Gorlice : Freiberg University, 2006. pp. 1-146.

8. **Hopkins, Phil and Cosham, Andrew.** The assessment of corrosion in pipelines-Guidance in the pipeline defect assessment manual (PDAM). *International Colloquium 'Reliability of High Pressure Steel Pipelines.* 27-28 March 2003, pp. 2-30.

9. **Julia, Race.** In-Service Integrity Assessment. *Pipeline Structural Analysis.* Newcastle upon Tyne, UK : School of Marine Science & Technology; Newcastle University, 2009/2010, pp. 2-10. M.Sc. Lecture Note.

10. **Ellenberger, J.P.** *Piping systems and Pipeline, ASME CODE SIMPLIFIED.* USA : Mc Graw-Hill Companies, Inc., 2005.

11. **Julia, Race.** Code Guidance on defect analysis. *Pipeline structural analysis.* Newcastle upon Tyne, UK : School of Marine Science & Technology, 2009/2010, pp. 5-12. M.Sc. Lecture Note.

12. **Pu, Young Chan.** Finite element method. *Pipeline structural analysis.* Newcastle upon Tyne, UK : School of Marine Science & Technology; University of Newcastle, 2009/2010, pp. 1-6. M.Sc. Lecture Note.

13. **Liu, Yijun.** Introduction to Finite Element Method. *Finite Element Method.* s.l. : University of Cincinnati, 2003, pp. 1, 101-106. Lecture Notes.

APPENDIX

TYPICAL RESULT FILE OF THE ANALYSIS WITH TWO LAYERS HAVING 294 ELEMENTS.

TABLE 5. EXTRACT FROM ELEMENT TABLE.

PRINT ELEMENT TABLE ITEMS PER ELEMENT

***** POST1 ELEMENT TABLE LISTING *****

STAT CURRENT CURRENT CURRENT
CURRENT CURRENT CURRENT CURRENT
CURRENT CURRENT
ELEM UX UY UZ SX SY SZ
SXY SYZ SXZ
1 0.29421E-01 1.0085 -0.15659E-04 439.14 -2.1748
133.16 -18.981 0.15361 1.1043
2 0.29290E-01 1.0074 -0.41239E-04 437.28 -1.9025
132.36 -

; ; ; ; ; ; ; ; ;
; ; ; ; ;
; ; ; ;

1478 0.46234E-02 0.77295 0.61300E-03 201.24 -8.1452
71.283 -3.0953 -2.5623 -1.1697

MINIMUM VALUES
ELEM 162 1109 310 554 357 190
1196 434 403
VALUE 0.13261E-02 0.86955E-03-0.26662E-01 -5.1226 -
17.161 -19.709 -104.36 -13.182 -29.119

MAXIMUM VALUES
ELEM 874 190 288 43 1074 47
871 421 406
VALUE 0.32592 1.0303 0.13918E-01 477.40 208.41
146.71 106.92 37.854 135.72

TABLE 6. EXTRACT FROM ELEMENT TABLE.

PRINT ELEMENT TABLE ITEMS PER ELEMENT

***** POST1 ELEMENT TABLE LISTING *****

STAT CURRENT CURRENT CURRENT
CURRENT CURRENT CURRENT CURRENT
CURRENT CURRENT
ELEM S1 S2 S3 SINT SEQV FX
FY FZ VOLU
1 439.96 133.16 -2.9900 442.95 392.98
0.18190E-11 0.17053E-12-0.56843E-13 29.278
2 438.13 132.32 -2.7120 440.84 391.21
0.0000
; ; ; ; ;
; ; ; ;

; ; ; ; ;
; ; ; ;

1478 201.29 71.356 -8.2746 209.57 183.22
220.52 15161. 0.21714E-09 6673.7

MINIMUM VALUES
ELEM 518 57 422 518 518 390
949 84 154
VALUE 77.227 -8.4136 -27.077 83.888 79.051 -
0.65302E-09-0.15240E+06-0.12025E-02 17.247

MAXIMUM VALUES

ISSN 2229-5518

ELEM 43 47 43 22 390 954 958 254 921
VALUE 477.57 144.74 61.943 447.07 408.07 0.15381E+06 0.14613E+06 0.18405E-02 70513.

TABLE 7. EXTRACT FROM NODAL LOAD PRESSURE FOR ALL SELECTED NODES.

LIST NODAL SURFACE LOAD PRES FOR ALL SELECTED NODES

ELEMENT	LKEY	FACE NODES	REAL	IMAGINARY
50	6	163	10.0000000	0.00000000
		712	10.0000000	0.00000000
		836	10.0000000	0.00000000
		161	10.0000000	0.00000000
51	6	712	10.0000000	0.00000000
;	;	;	;	;
;	;	;	;	;
1477	4	7108	10.0000000	0.00000000
		7080	10.0000000	0.00000000
		2378	10.0000000	0.00000000
		2380	10.0000000	0.00000000
1478	4	6952	10.0000000	0.00000000
		7108	10.0000000	0.00000000
		2380	10.0000000	0.00000000
		2063	10.0000000	0.00000000

TABLE 8. EXTRACT FROM LIST OF NODES.

LIST ALL SELECTED NODES. DSYS= 0
SORT TABLE ON NODE NODE NODE

NODE	X	Y	Z	THXY	THYZ	THZX
1	29.834	370.80	-45.000	0.00	0.00	0.00
2	29.152	362.33	-45.000	0.00	0.00	0.00
3	29.664	368.68	-45.000	0.00	0.00	0.00
4	29.493	366.57	-45.000	0.00	0.00	0.00
5	29.323	364.45	-45.000	0.00	0.00	0.00
6	0.20178E-13	363.50	-45.000	0.00	0.00	0.00
;	;	;	;	;	;	;
;	;	;	;	;	;	;
7835	5.3503	367.84	-663.60	0.00	0.00	0.00
7836	5.3503	367.84	-520.20	0.00	0.00	0.00
7837	5.3503	367.84	-376.80	0.00	0.00	0.00
7838	**5.3503**	**367.84**	**-233.40**	**0.00**	**0.00**	**0.00**

SAMPLE SEMI-EMPIRICAL CALCULATION USING BG/DNV LEVEL 1 CRITERION.

$$Failure\ stress = (UTS)\left[\frac{1-\frac{d}{t}}{1-\left(\frac{d}{t}\right)Q^{-1}}\right]$$

Where; $Q = \sqrt{\left[1 + 0.31\left(\frac{L^2}{Dt}\right)\right]}$

$d = defect\ depth = 90\ mm$
$L = defect\ length(mm)$
$t = pipe\ thickness = 17.5mm$
$D = pipe\ external\ diameter = 762mm$
$UTS = Ultimate\ Tensile\ strength = 563.8\ MPa$

$$Q = \sqrt{\left[1 + \frac{0.31(90)^2}{(762)(17.5)}\right]} = 1.09009241$$

$$Failure\ stress = (563.8)\left[\frac{1-\frac{9}{17.5}}{1-\left(\frac{9}{17.5}\right)1.09009241^{-1}}\right] = 518.4329\text{MPa}$$

Transposing the Barlow's formula,

$$Hoop\ stress = \frac{pipe\ internal\ pressure * (pipe\ external\ diameter - pipe\ thickness)}{2 * pipe\ thickness}$$

$$Failure\ pressure = \frac{failure\ stress * 2 * pipe\ thickness}{(pipe\ external\ diameter - pipe\ thickness)}$$

$$Failure\ pressure = \frac{518.4329 * 2 * 17.5}{(762 - 17.5)} = 24.3723MPa$$

$$Safe\ Operating\ Pressure = failure\ pressure * total\ Usage\ factor$$

$$Total\ Usage\ factor = modelling\ factor * operational\ Usage\ factor\ (Desig\ factor)$$

For liquid pipeline it is safe to assume a design factor of 0.72, hence:

$$Safe\ Operating\ Pressure = 24.3723 * 0.72 * 0.9 = 15.7933MPa$$

where, the modelling factor is taken as 0.9.

EQUIVALENT VON MISSES STRESS CALCULATION USING THE THREE SURFACE PRINCIPAL STRESSES FROM THE RESULT OF THE FE ANALYSIS.

$$\sigma EQ = \left(\frac{1}{\sqrt{2}}\right)\sqrt{[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]}$$

$\sigma_1 = 144.74MPa\ (axial\ stress)$
$\sigma_2 = 477.57MPa(hoop\ stress)$

$\sigma_3 = 61.943MPa(radial\ stress)$

ISSN 2229-5518

$$\sigma EQ = \left(\frac{1}{\sqrt{2}}\right)\sqrt{[(144.74 - 477.57)^2 + (477.57 - 61.943)^2 + (61.943 - 144.74)^2]}$$

$$\sigma EQ = 381.0361 MPa$$