Automatic finite element solid modeling, burst and error analyses of corroded pipelines

Rita C. C. Silva, João N. C. Guerreiro and Patricia R. C. Drach

Abstract— A reliable assessment of the remaining load carrying capacity of pipes containing single or multiple corrosion defects is a continuous matter of interest to engineers to reduce economical cost and a possible environmental damage. Solid finite element (FE) models have been widely used to perform failure analyses but the generation of the required models is, in many cases, a hard task to do. In this work, we present the principal tools of the PIPE program developed to provide a friendly graphical user interface for the Ansys software to perform finite element analyses of pipes with multiple rectangular defects in arbitrary position. The code allows a quick solid modeling, a guided non-linear analysis to obtain the failure pressure and also an error estimation to carry out a mesh refinement strategy. We present a validation test for the model generated by the program, demonstrating its capacity in simulating an experimental burst test of pressurized pipes. Error and burst analyses were carried out and experimental, numerical, and assessment methods results are compared.

Keywords—Corroded Pipeline, Finite Element Modeling, Mesh Generator, Numerical Simulation, Rupture Analysis.

I. INTRODUCTION

THE metal loss due to corrosion defects has become one of the leading causes of pipeline failure. The assessment of the structural integrity of corroded pipes has become of main interest to help engineers to take the important decision of replacing or repairing a pipeline.

Numerical approaches as the Finite Element Method (FEM) can be used to evaluate the pipeline integrity and to figure out the interaction effects due to multiple corrosion defects, as performed in [1], [2], and [3].

Finite element non-linear analysis of solid models requires expertise during the pre processing, processing, and post processing steps, which may spend a great deal of time and human effort.

In this work, we present a program, named PIPE, which we developed to, automatically, generate and solve solid models of pipes. The code is totally developed within the general-purpose program Ansys, which is a worldwide accepted commercial FE code and has been used in various fields of engineering, see for example some applications in, [4], [5], and [6].

The PIPE program allows a fast model generation of undamaged or corroded pipes with single or multiple rectangular defects in arbitrary position.

Four layouts can be modeled: the top left quarter, the top half, the left half, or the entire pipe, simulating, respectively, double, circumferential, longitudinal, or no symmetries of the defect patch configuration.

The undamaged pipe models are uniformly meshed while the corroded pipe models are meshed with a minimum of 1 (uniform mesh) and a maximum of 5 mesh refinement levels. Each refinement level includes a mesh transition along the pipe wall thickness and a mesh transition on the pipe surface, doubling the element size. The user has to inform the smaller element side length and the number of refined regions of the model. The length of each refined region can be automatically calculated by the program or can be given by the user.

If desired, the user can perform a linear analysis with *a posterior*i error estimation to carry out an appropriate mesh refinement strategy.

The burst analysis is performed automatically by an incremental procedure, adopting a controlled time step and a failure criterium used in joint projects of LNCC and Petrobras R&D Center.

Numerical, experimental, and assessments methods results are compared.

II. THE PIPE PROGRAM

The PIPE program was totally developed within the general-purpose program Ansys and was conceived to produce a graphical user interface to quickly generate 3D models and to perform elastic-plastic geometric non-linear burst analyses of non-corroded or corroded pipes.

The program is composed by a set of subroutines, containing standard Ansys commands and Ansys parametric design language (APDL) commands [7]. When executing an input file, the Ansys code is normally restricted to a linear flow and each statement is executed in the order that it is encountered. However, APDL provides a set of tools to

control the program flow, such as, commands to call subroutines and to specify branches and loops. The APDL is also used to activate components of the Ansys graphical user interface (GUI), such as, dialog boxes, buttons, and messages.

These tools allow a friendly interface to guide the user during the whole process that involves solid geometry modeling, mesh generation, and FE analysis.

The PIPE program encompasses three principal modules: input data, execute model, and solve as indicated in Fig. 1.
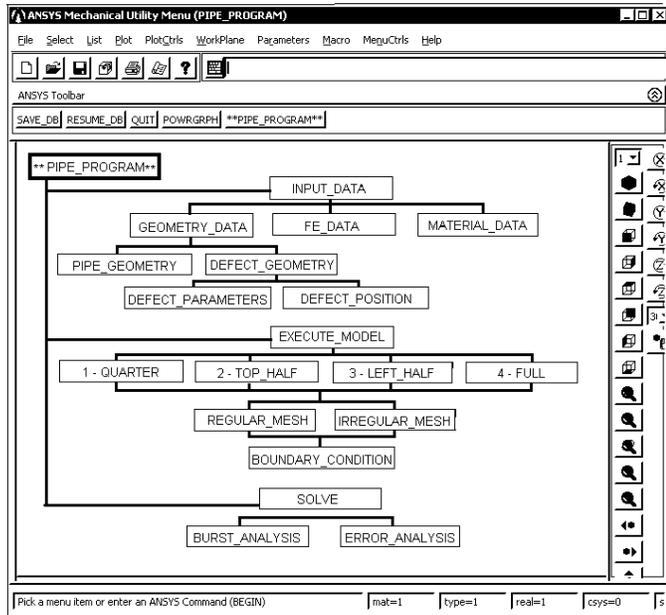
requesting informations about its parameters and position are displayed, allowing the generation of multiple different rectangular defects in arbitrary position.

Fig. 3 shows the screen of the PIPE program to input the geometric parameters of each defect: circumferential and longitudinal lengths, depth, top fillet radius, and frontal fillet radius.

Fig. 4 shows the screen to input the position of each defect: circumferential and longitudinal cylindrical coordinates of the center of the defect.

Fig. 1 the structure of the PIPE program

Fig. 2 the screen of the PIPE program to input the pipe geometric parameters

III. THE "INPUT DATA" MODULE

In this module all the data necessary for the model construction are introduced through a friendly interface, composed by instructive figures and dialog boxes. The geometry, the finite element and the material data are requested sequentially as the correspondent GUI tools become available.

A. Geometry Data

In the geometry data step, the user has to input the geometry data of the pipe and of the defects.

When starting the pipe geometry step, an appropriate dialog box and an instructive figure appear allowing the user to input the pipe geometric parameters: outside diameter, wall thickness and model length, as shown in Fig. 2.

After that, the user is asked about the number of defects to be considered in the model, taking into account that there is no limitation for this number. For zero defect, the program switches to the next module and conducts the modeling of an undamaged pipe. For a number of defects higher than zero, the program generates rectangular corrosion defects located on the external pipe surface. For each defect, dialog boxes
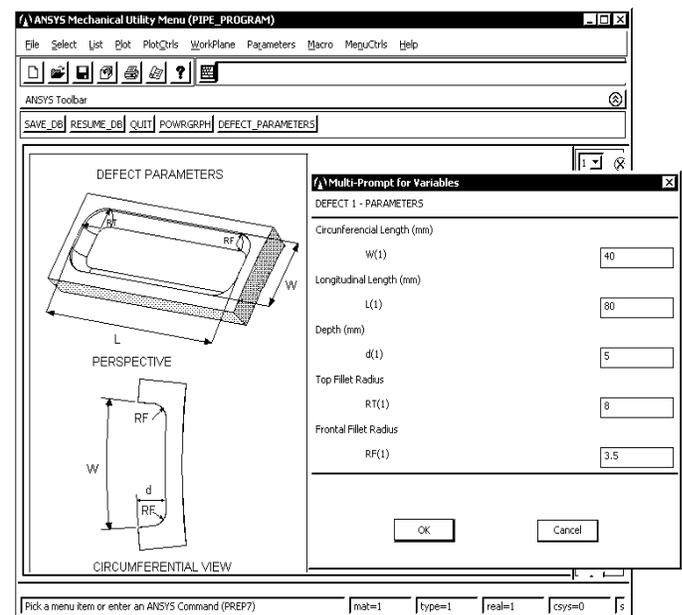

Fig. 3 the screen of the PIPE program to input the geometric parameters of each defect
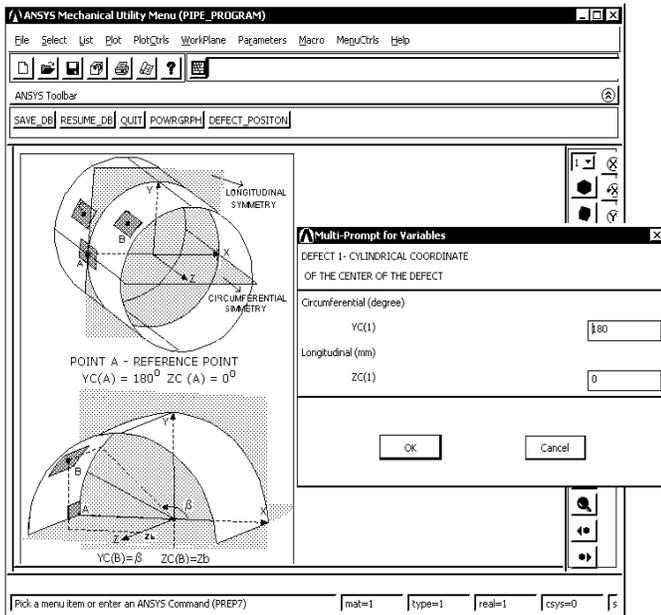
Fig. 4 the screen of the PIPE program to input the position of each defect

B. Finite Element Data

In this step, the mesh density is established. Undamaged pipes are modeled with uniform meshes. Corroded pipes can be designed with a maximum of 5 regions with different mesh refinements levels, according to user's choice. The PIPE program recommends a default value for the element size to be adopted in the first refinement level. To establish this value, the pipe wall thickness is meshed with a particular number of elements, multiple of 4, so that the element side length does not exceed 3 mm. The actual element side length along the pipe wall thickness is then suggested as a default value for the element size on the pipe surface, generating solid elements with an aspect ratio approximately equal to one. The user can modify this value and a new mesh along the pipe wall thickness is defined.

C. Material Data

The PIPE program allows the definition of one type of material only. Clicking the correspondent button the user is oriented to input the material properties using the GUI or by an input file. No default material type is available.

IV. THE "EXECUTE MODEL" MODULE

In this step, solid modeling and mesh generation are performed simultaneously. The model is meshed with the element Solid 45 included in the Ansys elements library [7].

At the proper time, buttons indicating the possibilities to start up, to return or to continue the modeling become available to control the flow of the program.

Initially, an outline of the corroded pipe model is showed to the user, as indicated in Fig. 5. In this step, collapsed defects or defects out of the domain are detected. In this case, the user is oriented to restart the PIPE program clearing all data and

initiating a new data input. For an acceptable defect patch configuration, the user should continue the pipe modeling.

The PIPE program allows to model four layouts: the top left quarter, the top half, the left half , or the entire pipe simulating, respectively, double, circumferential, longitudinal or no symmetries, as shown in Fig. 6.
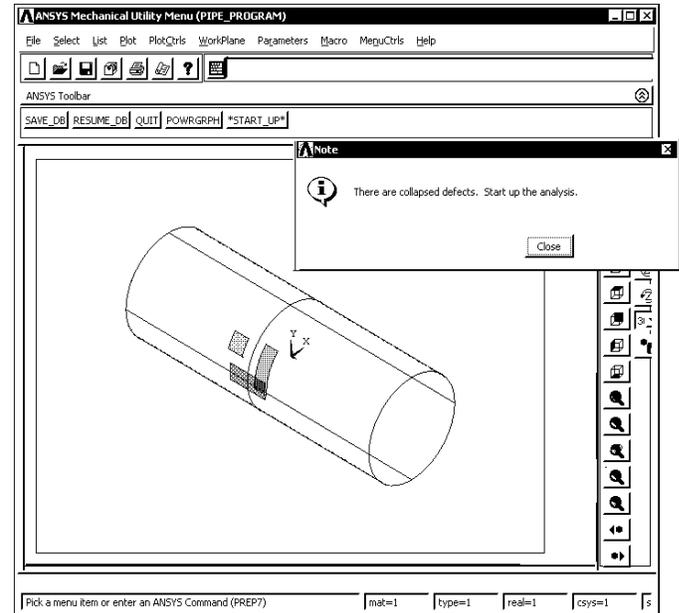

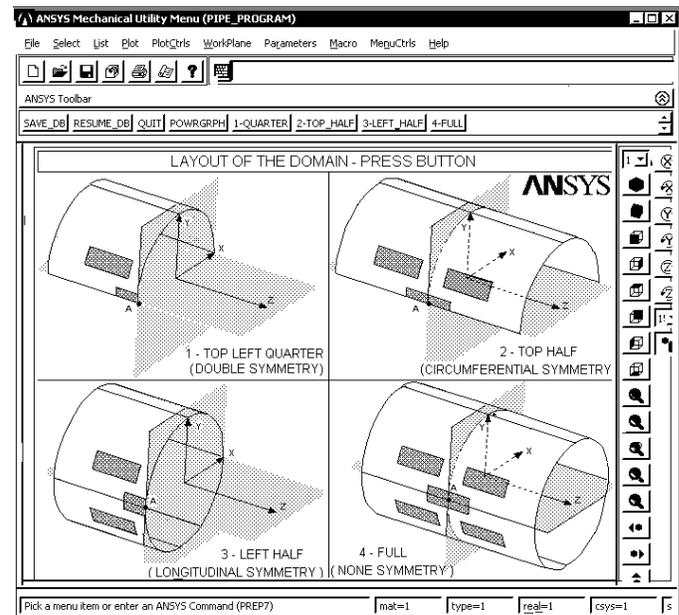Fig. 5 the screen of PIPE program, showing the position of the defects in a shell domain

Fig. 6 the screen of PIPE program, showing the options of the domain representation

Choosing one of the layouts, the user visualizes the correspondent domain including all the defects within it. An

option to return to the main screen of Fig. 6 becomes available, providing to the user the opportunity to visualize the other layouts until his final decision to continue the pipe modeling.

Afterwards, a region involving all the defects enclosed in the selected layout is delineated. This region, called defect patch region, is then meshed adopting the Ansys resources of mapped or free meshes, leading to regular or irregular meshes, respectively. Regular meshes contain only 3-D solid elements with 8 nodes, as shown in Fig. 7. Irregular meshes can contain 3-D solid elements with 6 or 8 nodes, as shown in Fig. 8. The user can visualize both options to make his decision.
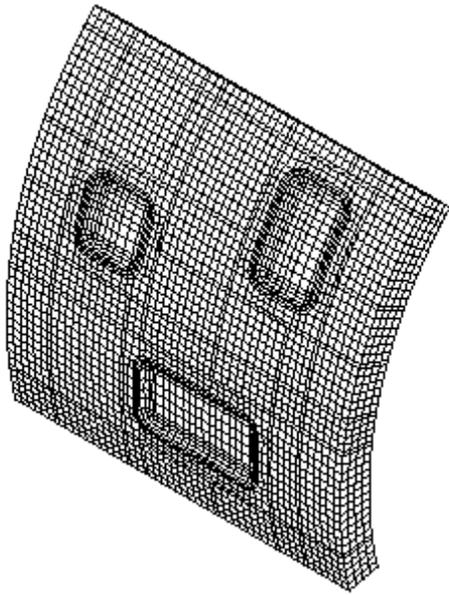

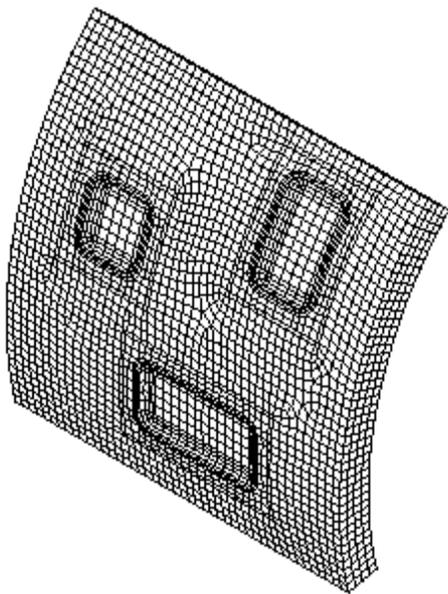Fig. 7 finite element model in the defect patch region with regular mesh

Fig. 8 finite element model in the defect patch region with irregular mesh

Continuing the pipe modeling, the program automatically generates and meshes all the regions with predefined refinement levels. An uniform mesh is used in the remaining domain.

Each refinement level includes a region for the mesh transition along the pipe wall thickness and a region for the mesh transition on the pipe surface, as indicated in Fig. 9.

In the transition regions, the PIPE program attempts to reduce to half the number of elements along the pipe wall thickness. This number of elements must be even and a minimum of 2 elements is required. On the pipe surface the procedure is to double the element side length.

The program informs a minimum and a suggested value for the dimension of each region of different refinement levels, i.e., the circumferential and the longitudinal total lengths of the region. The user can modify or accept one of these values.

The recommended values are calculated considering the distance between two adjacent transitions on the pipe surface as approximately equal to half of the minimum dimension of the greatest defect. Due to constructive convenience, this region has to be composed by at least 5 elements, establishing a minimum dimension for the refined region.

Before performing the transition on the pipe surface, each edge of the refined region must be composed by a number of elements multiple of 4.
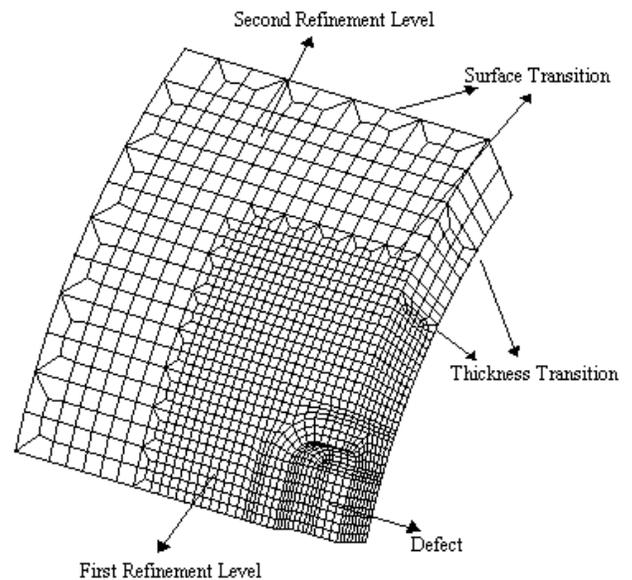

Fig. 9 a detail of the finite element mesh refinement

V. THE "SOLVE" MODULE

In this module the user can initiate a linear analysis to obtain an error estimation or a non-linear analysis to reach the pipe burst pressure.

A. Error Analysis

The purpose of this analysis is to provide the necessary tools to implement an adaptive strategy based on error estimators, allowing a more appropriate choice of the mesh refinement parameters: element size, number of refinement levels and dimension of each refinement level.

When starting the error analysis in the "solve" module of the PIPE program, a linear analysis is performed considering the pipe subjected to an unitary internal pressure and to the correspondent longitudinal force due to the closed ends condition. The program automatically provides the post processed error estimation described below.

The point-wise discretization error is simply the difference between exact and finite element solutions. In the classical displacement formulation of the finite element method for elasticity problems, the point-wise error can be referred to the primal variable (displacements) or to its derivatives (stresses, strains or other quantities of interest). The error referred to the stress tensor (e_σ) can be expressed as

$$e_\sigma = \boldsymbol{\sigma} - \boldsymbol{\sigma}_h, \tag{1}$$

where $\boldsymbol{\sigma}$ and $\boldsymbol{\sigma}_h$ represent, respectively, the exact stress tensor and the approximated numerical stress tensor.

Usually, the exact solution is unknown and the exact error of (1) can not be appraised. Post processed error estimation based upon the numerical solution can be evaluated to figure out the mesh quality. The idea is to replace the unknown exact solution ($\boldsymbol{\sigma}$) by a improved solution ($\overline{\boldsymbol{\sigma}}$) of the numerical solution ($\boldsymbol{\sigma}_h$), obtained by a post processing technique. The estimated error ($\overline{e}_\sigma$) can be defined as

$$\overline{e}_\sigma = \overline{\boldsymbol{\sigma}} - \boldsymbol{\sigma}_h. \tag{2}$$

In the error approximation technique of the software Ansys, available only for a linear analysis, the numerical stress field is improved by the nodal average recovery technique, which can lead to adequate results for elements with linear interpolations.

The point-wise discretization errors are difficult to be interpreted and adopted in adaptive strategies, so the errors are generally expressed in an appropriate norm.

Different adaptive procedures can be obtained, according to the choice of the norm measurement and of the adopted post processing technique, see for example [8], and [9].

The natural norm for the classical displacement formulation of the finite element method for elasticity problems is the energy norm. The estimated error in the energy norm is expressed as

$$\| \overline{e}_\sigma \| = \left(\int_\Omega (\overline{\boldsymbol{\sigma}} - \boldsymbol{\sigma}_h)^T \, \boldsymbol{D}^{-1} \, (\overline{\boldsymbol{\sigma}} - \boldsymbol{\sigma}_h) d\Omega \right)^{1/2}. \tag{3}$$

where $\boldsymbol{D}$ is the elasticity tensor.

In practice, (3) is calculated by summing over all elements in the domain Ω in the form

$$\| \overline{e}_\sigma \|^2 = \sum_{i=1}^{nel} \| \overline{e}_\sigma \|_i^2, \tag{4}$$

where nel is the total number of elements and $\| \overline{e}_\sigma \|_i$ is the estimate error in the energy norm of the element i, defined as

$$\| \overline{e}_\sigma \|_i = \left(\int_{\Omega_i} (\overline{\boldsymbol{\sigma}} - \boldsymbol{\sigma}_h)^T \, \boldsymbol{D}^{-1} \, (\overline{\boldsymbol{\sigma}} - \boldsymbol{\sigma}_h) d\Omega_i \right)^{1/2}, \tag{5}$$

where Ω_i is the element i of the domain.

An average error can be defined as

$$e_m = \sqrt{\left(\sum_{i=1}^{nel} \| \overline{e}_\sigma \|_i^2 \right) \Big/ nel}. \tag{6}$$

For each element, the estimated error in the energy norm (5) is related to the energy error $\in_i$, available in the Ansys program, in the form

$$\| \overline{e}_\sigma \|_i = \sqrt{2 \in_i}. \tag{7}$$

The energy error over the model $\in$ can be normalized against the strain energy U, providing a global percentage error η, also available in the Ansys code, as

$$\eta = 100 \left[\in / (\mathsf{U} + \in) \right]^{1/2}, \tag{8}$$

where

$$\in = \sum_{i=1}^{nel} \in_i, \tag{9}$$

$$\mathsf{U} = \sum_{i=1}^{nel} \mathsf{U}_i = \frac{1}{2} \sum_{i=1}^{nel} \int_{\Omega_i} (\boldsymbol{\sigma}_h)^T \, \boldsymbol{D}^{-1} \, (\boldsymbol{\sigma}_h) d\Omega_i. \tag{10}$$

Local, global and percentage error estimators are very useful to elaborate criteria and strategies in order to obtain an adequate mesh refinement. A global criterium can be used to obtain an appropriate element size and a local criterium can be adopted to establish the refinement levels.

An alternative for a global criterium is to consider a mesh adequate when the percentage error associated with it has an acceptable value.

According to the error balancing concept, a finite element mesh is efficient when it has a uniform error distribution, i.e., the error of each element is approximately equal to the

average error. Then, an alternative for a local criterium is to refine an element according to the relation between its error in the energy norm and the average error.

When finishing the error analysis, the user can visualize the distribution of the estimated error in the energy norm calculated by using (5). The average error (6) and the percentage error (8), available in Ansys code, will be also displayed.

B. Burst Analysis

The purpose of this analysis is to obtain the failure pressure of pipes subjected to an internal pressure.

When starting the burst analysis in the "solve" module of the PIPE program, a non-linear analysis is performed taking into account the material non-linearity and the large strains and displacements effects. The analysis is controlled by a step-by-step procedure using the restart resource of the Ansys software.

The pipe is subjected, simultaneously, to an internal pressure and to a longitudinal force due to the closed ends condition.

The loads are applied incrementally until the failure criterium is reached. The program calculates the initial pressure increment as being approximately equal to a quarter of the pressure value that makes the pipe reach a stress level close to the material yield stress. At each step of the analysis, the load increment is applied in four sub-steps. The load increment is reduced when the maximum plastic strain exceeds the predefined value of 0.005 or when the convergence is not achieved within 50 iterations.

The adopted failure criterium establishes that failure is reached when the von Mises stress along a section in the radial direction exceeds the true ultimate tensile stress, taking into account all the points situated across the thickness. The incremental analysis stops when this failure criterium is attained. The analysis is also interrupted when the pressure increment becomes smaller than 0.01 MPa. In this situation, the failure pressure is considered as the maximum applied pressure load.

The Ansys will automatically quit at the end of the analysis. The PIPE program also provides an output ASCII file that records for each sub-step the pressure increment, the number of iterations, the maximum von Mises stress and the maximum plastic strain.

VI. PRESSURE TEST

The accuracy of the finite element analysis is highly dependent upon the features used in the generation of the model.

In this section, we present a validation test for the model generated by the PIPE program, demonstrating its capacity in simulating an experimental burst test of pressurized pipes. Error and burst analyses were carried out and experimental, numerical, and assessment methods results are compared.

A. Description of the Case Study

Our goal is to reproduce the results obtained in the laboratory test of the specimen called IDTS 11 reported in [10].

The tubular specimen was cut from a longitudinal welded tube made of API 5L X80 steel with an actual outside diameter of 459.4 mm and an actual wall thickness of 8.0 mm.

The true stress-strain curve of the API 5L X80 steel was calculated using the Ramberg-Osgood equation that can be expressed as

$$\varepsilon = \sigma/\mathrm{E} + 0.002\,(\sigma/\sigma_{yield})^n \ , \tag{11}$$

where σ and ε are, respectively, the stress and strain, σ_{yield} is the yield stress, E is the elasticity modulus, and n is a constant of the material.

The true stress-strain curve of the material, shown in Fig. 10, was obtained considering in (11) the following values: σ_{yield} =595 MPa, E=200000 MPa, and n=16.333.

The curve was defined up to 7.11% strain, above this level the stress remains constant and equal to the ultimate strength of σ_{uts} =738 MPa. The values of the yield stress and of the ultimate strength were obtained by tensile and impact tests registered in [10].
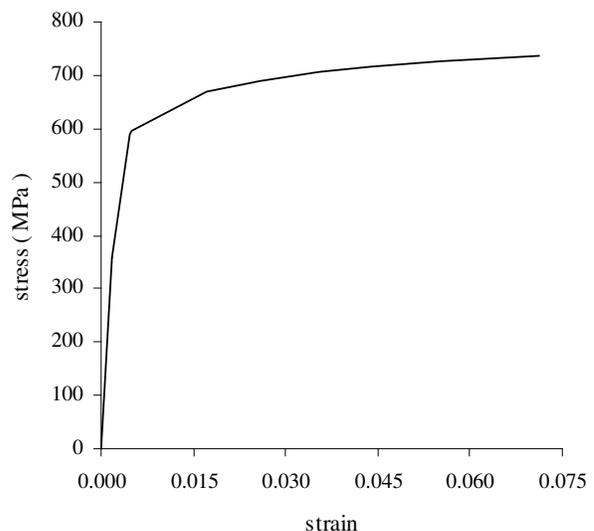

Fig. 10 true stress-strain curve of the material

The specimen IDTS 11 contains 5 defects machined using spark erosion. They are external rectangular defects with smooth edges made with a small radius of 3.2 mm. Each defect has actual width, length and depth of 32.11 mm, 40.06 mm and 3.81 mm, respectively. In the picture of the defects of the specimen IDTS 11, shown in Fig. 11, we

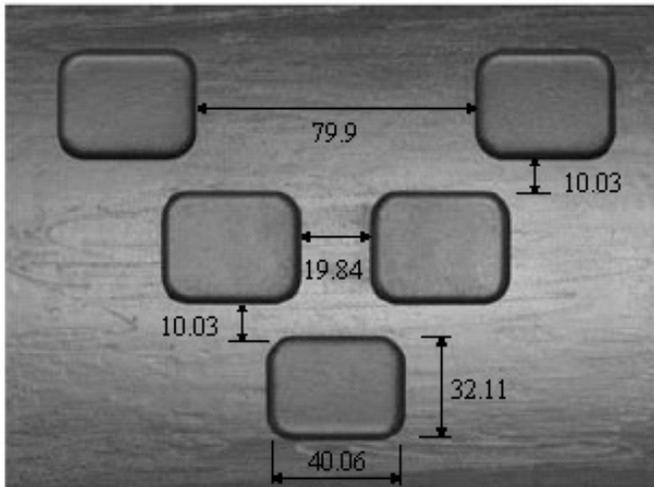indicate the actual length, width and spacing of the defects.

Fig. 11 picture of the specimen IDTS 11 showing the length, width, and spacing of the defects

B. Mesh Validation

In this section, we carried out error analyses to obtain adequate values for the mesh parameters requested by the program during the pipe modeling.

Linear analyses are performed considering the pipe subjected to an unitary internal pressure and to the correspondent longitudinal force. Due to the position of the defects, only the left half of the pipe was modeled and the appropriate default boundary conditions were applied to simulate the longitudinal symmetry.

Initially, we performed an error analysis in a segment of the specimen IDTS 11 with total length of 1300 mm. The model was designed adopting a uniform mesh with 4 elements through the pipe wall thickness and with element on the pipe surface with a side length of 3.6 mm.

Fig. 12 shows the von Mises stresses distribution obtained in this error analysis. As can be observed in this figure, there is no localized effect due to the application of the longitudinal force, indicating that the total length adopted in the model is adequate.

Our goal is to model the pipe with a mesh containing 3 refinement levels. An alternative local criterium, based on the error balancing concept, is to refine the elements according to the relation between estimate and the average errors.

Fig 13 shows the distribution of the estimated errors in the energy norm, intentionally displayed in 3 pre defined intervals. This strategy automatically delineates the desired 3 regions of different error levels.

In the first region, close to the defects, all the elements present errors higher than 2 times the calculated average error of 0.005564. The second region contains elements with estimated errors higher than 1 and smaller than 2 times the average error. All the elements inside the third region have estimate errors smaller than the average error.

The dimension of each refinement level, requested by the PIPE program, can be easy obtained by the measurement of these regions.
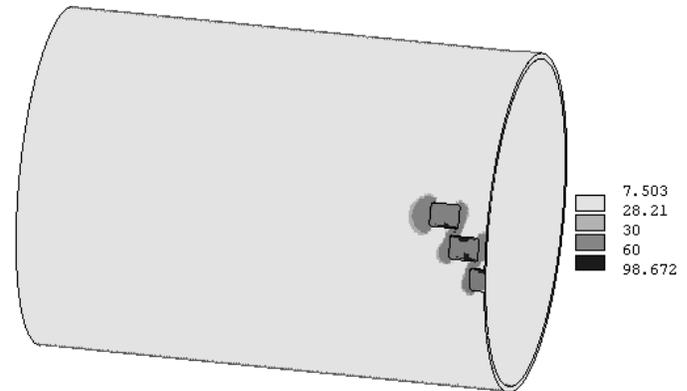

Fig. 12 von Mises stresses distribution obtained in the error analysis with uniform mesh
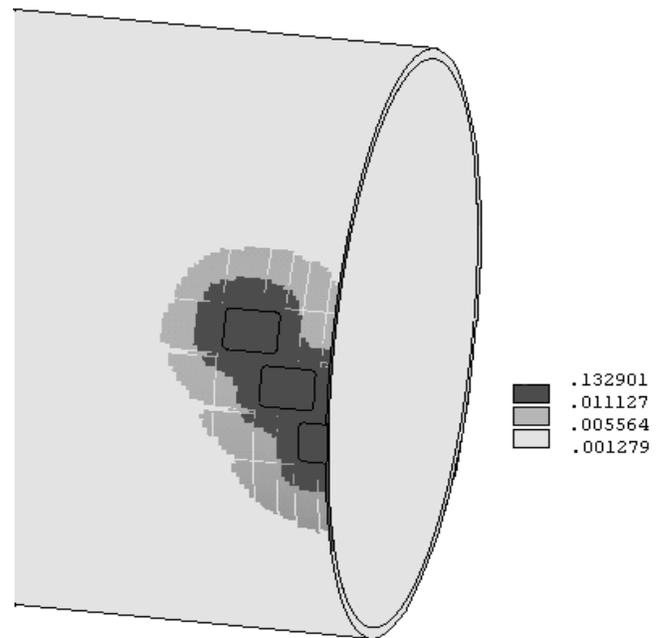

Fig. 13 distribution of the estimate errors in the energy norm obtained in the error analysis with uniform mesh

Based on this strategy, a finite element solid model, designed with 3 mesh refinement levels, was constructed by the PIPE program. A detail of this model can be visualized in Fig. 14.

In the more refined region of the model, we adopted elements on the pipe surface with a side length equal to

1.8 mm, this value was increased to 3.6 mm, and 7.2 mm in the neighboring regions. In the more refined region, 4 elements were used through the pipe wall thickness. In the remaining domain 2 elements were used through the pipe wall thickness, since it is the minimum number required by the program.

A new error analysis was performed to verify the accuracy of this refined mesh. A percentage error of 3.0% was obtained and this error level was considered acceptable for the reliability of the numerical simulation.
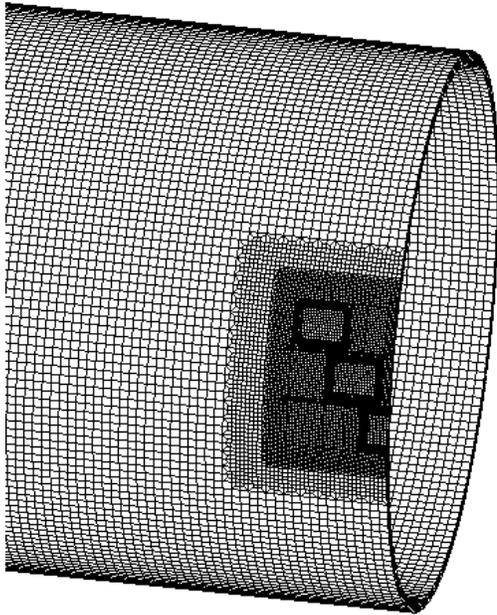

Fig. 14 detail of the finite element solid model designed with 3 refinement levels
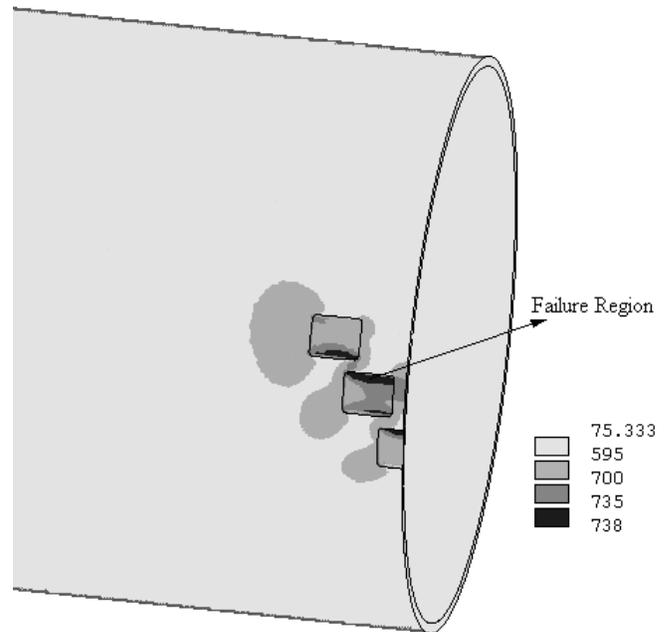

Fig. 15 detail of the von Mises stresses distribution at an internal pressure of 20.91 MPa

Fig. 16 picture of the defects of the specimen IDTS 11 after failure

C. Pipe Assessment

An incremental non-linear analysis was carried out to obtain the failure pressure of the speciment IDTS 11. The pipe was modeled adopting the refined mesh established in the previous error analysis, shown in Fig. 14.

In Fig 15 we present a detail of the von Mises stresses distribution obtained at the burst pressure level of 20.91 MPa. This pressure level was achieved according to the adopted failure criterium, which establishes that failure occurs when the von Mises stresses of all the points situated along a cross section in the radial direction reach the true ultimate tensile stress.

In the numerical analysis of the specimen IDTS 11, the failure process develops from the external to the internal pipe surface. The place where the failure occurs is indicated in Fig 15. It can be observed that, the predicted failure configuration is in agreement with that obtained in the laboratory test shown in Fig. 16.

D. Results Comparison

In this section, the failure pressured of the specimen IDTS 11 measured in the laboratory test [10] is compared with the numerical result and with those predicted by three assessments methods for isolated defects namely: the ASME B31G method [11], the RSTRENG *0.85dl* method [12], and the DNV RP-F01 method [13].

The IDTS 11 presents a group of defects in which the defects are separated circumferentially but its individual profile overlaps when projected onto the longitudinal plane through the wall thickness, as shown in Fig. 17. The failure

pressure of this kind of interacting defects can be calculated using one the of the available assessment methods for isolated defect and considering that the dimension of the combined defect is the overall dimension of the cluster (maximum cluster depth d_{clus} and overall length L_{clus}).
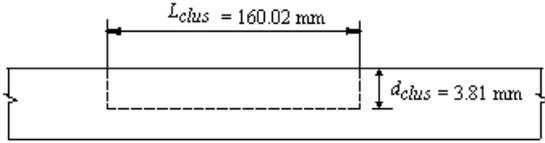

Fig. 17 cluster formed by the group of defects of specimen IDTS 11 projected onto the longitudinal plane

In the ASME B31G method, the failure pressure for short defects is calculated as

$$pf = 1.1\ \sigma_{yield}\ \frac{2t}{D_e}\left[\frac{1-(2/3)(d/t)}{1-(2/3)(d/t)M^{-1}}\right] \tag{12}$$

where

$$M = \left[1+0.8\frac{L^2}{D_e\,t}\right]^{1/2} \tag{13}$$

The RSTRENG *0.85dl* method establishes the failure pressure as

$$pf = (\sigma_{yield}+69MPa)\frac{2t}{D_e}\left[\frac{1-0.85\,(d/t)}{1-0.85\,(d/t)M^{-1}}\right] \tag{14}$$

where

$$M = \left[1+0.6275\frac{L^2}{D_e\,t}-0.003375\left(\frac{L^2}{D_e\,t}\right)^2\right]^{1/2} \tag{15}$$

In the DNV RP-F01, the failure pressure is evaluated in the form

$$pf = \sigma_{uts}\ \frac{2t}{(D_e-t)}\left[\frac{1-(d/t)}{1-(d/t)M^{-1}}\right] \tag{16}$$

where

$$M = \left[1+0.31\frac{L^2}{D_e\,t}\right]^{1/2} \tag{17}$$

To evaluate the predicted failure pressures by these equations, we adopted the actual dimensions of the tubular specimen and of the cluster formed by the group of the defects: outside diameter, D_e=459.4 mm; wall thickness, t=8.0 mm; defect longitudinal length, $L=L_{clus}$=160.02 mm; and defect depth, $d=d_{clus}$=3.81 mm. The yield stress σ_{yield}=595 MPa and the ultimate strength σ_{uts}=738 MPa. In Table 1, the failure pressure measured in the laboratory test is compared with those predicted by the assessment methods and with the FEM result. In the same table, the errors of the failure pressure predictions are presented.

Table 1 Actual and predicted failure pressures and correspondent errors

Method	Failure Pressure (MPa)	Error (%)
Burst test	21.260	0
FEM	20.910	-1.65
ASME B31G	17.756	-16.48
RSTRENG 0.85*dl*	16.732	-21.30
DNV	18.715	-11.97

Error(%)=((predicted-experimental)/experimental)*100

All the predicted failure pressures are conservative, i.e., the predictions are smaller than the actual failure pressure.

The worst performance was obtained by the RSTRENG 0.85*dl*. It is important to mention that the conservatism of the RSTRENG 0.85*dl* and of the ASME B31G results, when applied in the assessment of flat bottomed defect, is artificially reduced due to the non-rectangular geometric shape that these method adopt to represent the longitudinal area of metal loss.

The most accurate result was the one provided by the numerical simulation. This analysis presented an absolute error level of 1.65% smaller than the estimated percentage error of 3% obtained in the error analysis, showing its reliability.

VII. CONCLUSION

In this paper we presented the principal tools of the PIPE program developed to generate solid finite element models and to automatically manage FE analyses of undamaged or corroded pipelines. The program provides a friendly graphical user interface for the Ansys software allowing a guided and quick modeling of pipes containing multiple different defects in arbitrary position. A validation test was presented showing that the program leads to an appropriate model generation and to a reliable numerical simulation.

REFERENCES

[1] R. C. C. Silva, J. N. C. Guerreiro, and A. F. D. Loula, "A study of pipe interacting corrosion defects using the FEM and neural networks," *Advances in Engineering Software*, no.38, 2007, pp. 868-875.

[2] E. Q. Andrade, A. C. Benjamin, P. R. S. Machado Jr, L. C. Pereira, B. P. Jacob, E. G. Carneiro, J. N. C. Guerreiro, R. C. C. Silva, and D. B. Noronha Jr, "Finite element modeling of the failure behavior of pipelines containing interacting corrosion defects", *in Proc. 25th International Conference on Offshore Mechanics and Arctic Engineering*, Hamburg, Germany, 2006, pp. 1-11.

[3] A. C. Benjamin, D. J. S. Cunha, R. C. C. Silva, J. N. C. Guerreiro, G. C. Campello and F. E. Roveri, "Stress concentration factors for a drilling riser containing corrosion pits,*" in Proc. 26th International Conference on Offshore Mechanics and Arctic Engineering*, San Diego, California, 2007.

[4] J. S. Bang, S. H. Han, J. K. Lee, and S. W. Park, " An automation of fatigue durability for welded bogie frame using system integration technique," *in Proc. 6th WSEAS Int. Conf. on Simulation, Modeling and Optimization*, Lisbon, Portugal, 2006, pp. 349-354.

[5] S. K. Bhatti, S. Kumari, M. L. Neelapu, C. Kedarinath, and N. Kumar, "Transient state stress analysis on an axial flow gas turbine blades and disk using finite element procedure," *in Proc. 4th WSEAS Int. Conf. on Heat Transfer, Thermal Engineering and Environment*, Elounda, Greece, 2006, pp. 323-330.

[6] J. K. Sakellaris, "Finite element analysis of micro-electro-mechanical systems: towards the integration of MEMS in design and robust optima control schemes of smart microstructures," *WSEAS Trans. on Applied and Theorical Mechanics*, vol. 3, April 2008, pp. 114-124.

[7] *Ansys Release 9.0 Documentation,* Ansys, 2004.

[8] R. C. C. Silva, A. F. D. Loula, and J. N. C. Guerreiro, " Local gradient and stress recovery for triangular elements," *Computers and Structures*, no. 82, 2004, pp. 2083-2092.

[9] L. Vasiliauskiené, S. Valentinavicius, and R. Bausys, "Postprocessing error estimator taking to the account singularity effect," *in Proc. 9th WSEAS Int. Conf. on Applied Mathematics,* Istanbul, Turkey, 2006, pp. 379-384.

[10] A. C. Benjamin, R. D. Vieira, J. L. F. Freire, and E. Q. Andrade, "Burst testes on pipeline containing closely spaced corrosion defects," *in Proc. 25th International Conference on Offshore Mechanics and Arctic Engineering*, Hamburg, Germany, 2006, pp. 1-14.

[11] Anon, "ASME-B31G – Manual for Determining the Remaining Strength of Corroded Pipelines – A Supplement to ANSI/ASME B31," Code for Pressure Piping, American Society of Mechanical Engineers, New York, 1991.

[12] J. F. Kiefner, and P. H. Vieth, " A modified criterion for evaluating the remaining strength of corroded pipe," Final Report on Project PR 3-805, *Pipeline Research Committee*, America Gas Association, 1989.

[13] Anon, "DNV recommended practice - DNV – RP-F101 – Corroded Pipelines," *Det Norske Veritas*, Norway, 1999.