Fatigue and Fracture Analysis of Submarine Pressure Hulls

TEILOR MACIEL KOPAVNICK - teilor@cpn.mar.mil.br
CPN (Brazilian Naval Ship Design Office) – Ilha das Cobras, s/Nº, Rio de Janeiro
JULIO CÉSAR RAMALHO CYRINO - julio@peno.coppe.ufrj.br
COPPE – UFRJ (Federal University of Rio de Janeiro) – Ilha do Fundão, Centro Tecnológico, bl.C, Rio de Janeiro

Abstract. *The development of approaches concerning about fatigue is one of the most important control strategies in this process that consumes millions of dollars in lives and materials all over the world. In this work, a method is presented to calculate a lifetime until the fatigue of submarine pressure hull, using Linear Elastic Fracture Mechanics based on Paris equation (ROBLES, BUELTA e GONÇALVES), besides your validation by the finite elements method. It is known that this structure is submitted to high strains caused by variable hydrostatic pressure and residual strains derived from welding process. The association of those strains may cause, after a certain period of time, the fracture of the material by fatigue. In the use of Fracture Mechanics for determination of a fatigue lifetime, a quantity of fundamental importance is the stress intensity factor. The purpose of this article is to compare the stress intensity factor and the maximum crack size calculations by three different formulations, when these formulations are used in the specific case of a pre-established region of submarine pressure hull. In this study will be considered a methodology developed by ROBLES, BUELTA and GONÇALVES, which is based in the application of analytical formulations for the stress intensity factor calculus with simple models for the crack elements. Moreover it will be considered two finite elements modeling; the first, using a program in FORTRAN language with a crack element proposed by ATLURI and the other considering the stress intensity factor calculus by the commercial program ANSYS. The results obtained by these methods showed values so close to those obtained by the experimental tests with real state models by DUNHAM. These real state models helped the results validation, because they possessed dimensions very similar to those adopted in this study. These results could help in establish the periods for repair the possible cracks in submersible structures, because, a lot of time, the aleatory and early repairs could originate cracks with sizes more critical than those before.*

Keywords: *fracture mechanics, finite elements, stress intensity factor, submarine pressure hull*

1. INTRODUÇÃO

In the use of Fracture Mechanics for determination of fatigue operational life, one of the most important value to be calculated is the stress intensity factor. The aim of this article is to compare the stress intensity factor and critical crack depth calculation using three different formulations, when these are applied to the specific case of a pre-established region of submarine pressure hull. In this study will be considered a methodology developed by Robles et al.2000, which is based on the application of analytical formulations for the stress intensity factor calculation using a simple crack element models. It will also be considered two finite elements modeling, the first employing a crack element proposed by Atluri (1986) and the other considering the stress intensity factor calculus by the finite elements program ANSYS (2004). It will be verified that these three methods showed results so close to those finded by Dunham (1965) with full-scale models of submarine pressure hulls submitted to external pressure. These models were useful to validate the results finded in this article, because they had very similar dimensions to that adopted in the present study.

2. EXPERIMENTAL RESULTS TO FATIGUE OF SUBMARINE PRESSURE HULLS

Submarine pressure hull fatigue has been analysed many times since the beginning of the 1960s, after the development of submarines using nuclear power, due to the fact that they could operate for longer periods of time, and at greater depths underwater than diesel-electric submarines, usually used by navies at that time (Heller 1962).

Several researchers have studied fatigue-induced crack propagation in submarine pressure hulls. Dunham (1965) and Kilpatrick (1986) conducted experiments with models of pressure hulls, submitted to the action of oscillating external pressure, monitoring crack propagation in the structure.

According to the Dunham (1965) experiments (Fig. 1), it is possible to verify that from the beginning of the cracks detection, a growth in the total circumferential length of the cracks happens, so that in a specific time the ratio of crack growth becomes very high. The cracks grow in the pressure hull inner part, in the toe of the welds, and propagate first around the circumference of the pressure hull, becoming a long circumferential crack, and then propagating through the thickness of the pressure hull causing his perforation. According to Dunham (1965), the approximate value of 25% of the circumferential length, represented by 25% of the hull thickness either, was considered as a critical value for his model, where the cracks beginning to coalesce one to each other inside the circumferential perimeter, promoting, thus, a extremely high crack propagation ratio, represented by the “peak” in the graph of Figure 1.

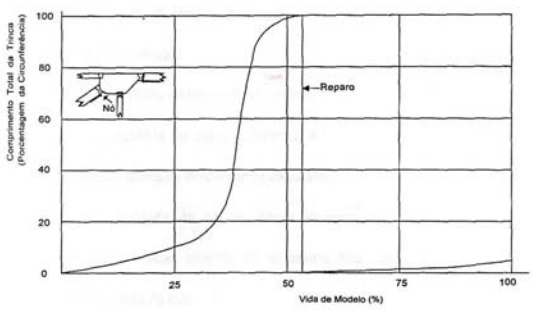

Figure 1. Evolution of crack circumferential length as a function of the total time of experiment. Connection of conical hull to the reinforced intersection.

It was verified that the most critical areas for fatigue evaluation are the intersection between two axisymetrical solids, such as cone-cylinder, cone-cap, or between two cones with different conicity (Fig. 2), due to the high stress acting on these areas induced by hydrostatic loading. Thus, the fatigue operational life of submarine structure will be given by the fatigue life of this critical region.

Furthermore, in these areas, due to the hull construction planning, there are several welding lines that can induce the presence of residual stress and cracks. Therefore, regarding crack growth under cyclic loading, these areas can be considered to be the most critical for a fatigue analysis in this study. The most critical area adopted to the fatigue analysis in this article was the cone-cone region (Robles et al. 2000), near the stern of the submarine.

The conclusion of this theorical study is similar to the experimental results presented by Dunham (1965) and Kilpatrick (1986), where fatigue failure, defined by crack growth under cyclic loading, until hull perforation, always happened in the areas where there were intersections of two axisymetrical solids, and close to the welding lines.

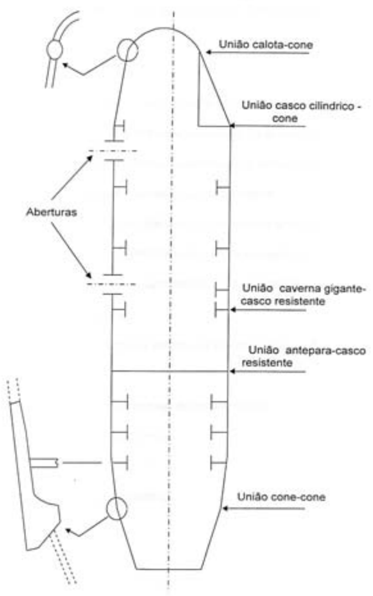

Figure 2. Longitudinal position of pressure hull regions most likely to have fatigue failures. (Robles et al. 2000)

3. ANALYTICAL METHOD OF CALCULATION

In this section, a finite element modeling of the area to be analysed (intersection of cone–cone at astern), using the ANSYS (2004) program, is showed. The output results of this analysis will be serve as input values in the analytical modeling developed by Robles et al. 2000, and will be represented here as a illustration of the process.

3.1. *Stresses due to Hydrostatic Pressure*

A finite elements model accomplishes the calculus of stress due to the hydrostatic pressure. The stresses are calculated using a hydrostatic pressure of 2,5MPa (250m of submarine maxim depth) applied at the external face of submarine pressure hull in the cone-cone intersection, as early described. Thus, using the ANSYS (2004) program, the modeling of the considered region is made. This modeling is showed in the Figure 3, where it is possible to observe the reinforced intersection and the frames.

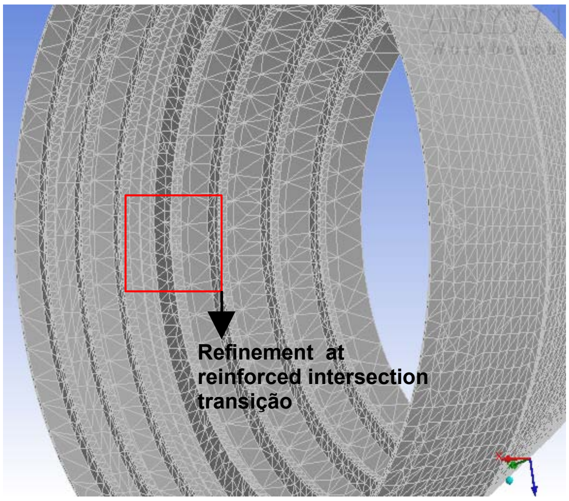

Figure 3. Mesh representation with refinement at reinforced intersection.

The output results for the stress and "total deformations" are represented in the Figures 4-5.

The model is processed by the ANSYS (2004) program and the output results are:

$\sigma_{Lint(hmax)} = -\ 224,46$ MPa - Longitudinal normal stress in the pressure hull inner part, considering a submarine maximum operational depth of 250m.

$\sigma_{Lext(hmax)} = -\ 51,48$ MPa - Longitudinal normal stress in the pressure hull outer part, considering a submarine maximum operational depth of 250m, at the correspondent point of $\sigma_{Lint(hmax)}$.

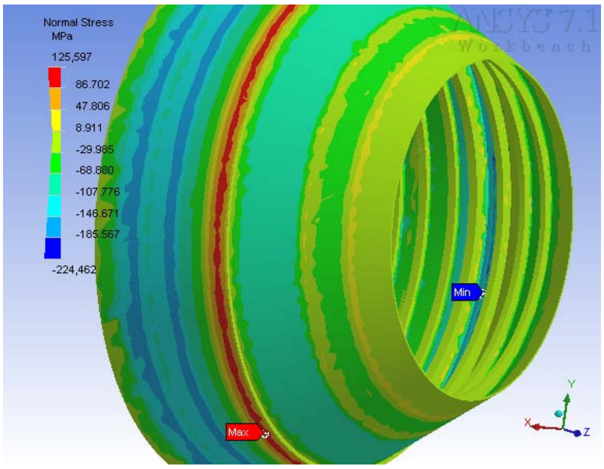

Figure 4. Output results to the normal stressing the hull.

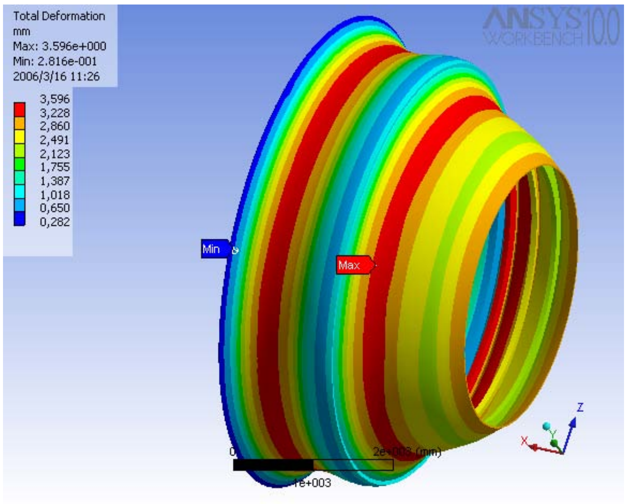

Figure 5. “Total deformation” of pressure hull.

The K_I (stress intensity factor) calculation is expressed in the function of the longitudinal stresses. The total longitudinal stress due to external pressure is divided into a flexural component and a membrane component. This division is important to conform the stress intensity factor calculation to the analytical model developed by Robles et al. 2000. Adopting these results and knowing that the membrane stress distribution and the bending stress distribution are based on the depth (*h*), the following equations are used to express this variation:

$$\sigma_{lext} = \sigma_{lext(h\max)} \frac{h}{h_{\max}} \tag{1}$$

$$\sigma_{l\,\mathrm{int}} = \sigma_{l\,\mathrm{int}(h\max)} \frac{h}{h_{\max}} \tag{2}$$

$$\sigma_{F(h)} = \frac{\sigma_{l\,\mathrm{int}} - \sigma_{lext}}{2} \tag{3}$$

$$\sigma_{n(h)} = \sigma_{l\,\mathrm{int}} - \sigma_{F(h)} \tag{4}$$

Where:

$\sigma_{lext(h\max)}$ - Longitudinal normal stress in the pressure hull outer part at maxim depth;

$\sigma_{l\,\mathrm{int}(h\max)}$ - Longitudinal normal stress in the pressure hull inner part at maxim depth;

$\sigma_{l\,\mathrm{int}}$ - Longitudinal normal stress in the pressure hull inner part at operational depth *h*;

σ_{lext} - Longitudinal normal stress in the pressure hull outer part at operational depth *h*;

$\sigma_{F(h)}$ - Bending stress at operational depth *h*;

$\sigma_{n(h)}$ - Membrane stress at operational depth *h.*

According to Robles et al. 2000, through the stress calculation model showed above, it is possible, using a crack element analytical modeling, to obtain the stress intensity factors as functions of crack length, submarine operational depths and the stresses components (normal, bending and residual). The stress intensity factors, corresponding to each component mentioned above can be summed up, according to the superposition principle (Barson 1987, Broek 1986), and therefore can be obtained the total stress intensity factor value. Besides, it is possible to attain the variations of this stress intensity factor as a function of crack length and submarine operational depth, which will be very important to the fatigue calculus of number of cycles, when the Paris (1963) law is applied. The output results can be showed in the graphical form like those of Figures 6-7, and they are according to the results attained by Robles et al. 2000:

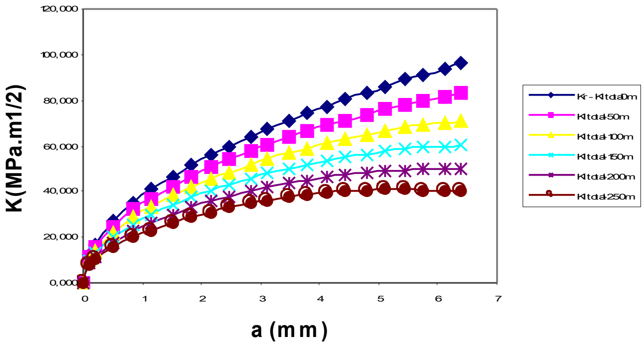

Figure 6. Stress intensity factor V_S crack depth with a correction due to plasticity.

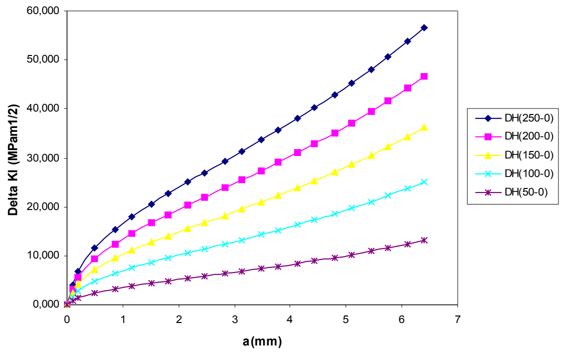

Figure 7. Variation of stress intensity factors (ΔK) V_S crack depth.

4. FINITE ELEMENTS METHOD FOR THE STRESS INTENSITY FACTOR CALCULATION AND FOR THE VALIDATION OF ANALYTICAL METHOD RESULTS

The aim in Linear Elastic Fracture Mechanics (LEFM) is to evaluate the relevant stress intensity factor (K), which is then compared with the critical stress intensity factor (K_c), as a criterion for crack propagation. If the cracks are encountered in complex stress fields or in intricate geometries, a numerical solution technique becomes imperative.

This study adopted the Linear Elastic Fracture Mechanics (LEFM) but using an "exact field modeling" for the stress and displacement fields (Fawkes et al. 1979), added by the Irwin (1957) correction so that the plastification effect in the crack tip could be considered. The modeling will also be added by a residual stress distribution, which will cause the appearing of tension stresses in the crack tip provoking her growth and the fatigue of material. This "exact field modeling" will present some precise results, by the way, for an economical solution, that is, for a mesh not so refined, as will be showed in the next items.

4.1. *Analytical Solution for the Stress and Displacement Fields*

Muskhelishvili (1953) showed that every biharmonic function $A\ (x, y)$, of the variables x, y may be represented in a very simple manner by the use of two functions of the complex variable $z = x+iy$. Muskhelishvili (1953) showed that for two analytic functions $\phi(z)$ and $\chi(z)$ of the complex variable z, the stress function, stresses and displacements could be respectively written as:

$$\sigma_x + \sigma_y = 4\,\mathrm{Re}\,\phi'(z) = 2\left[\phi'(z) + \overline{\phi'(z)}\right]$$

$$\sigma_y - \sigma_x + 2i\sigma_{xy} = 2[\bar{z}\phi''(z) + \chi''(z)] \tag{5}$$

$$2\mu\,(u + iv) = \kappa\,\phi(z) - z\overline{\phi'(z)} - \overline{\chi'(z)}$$

Where:

$$\phi'(z) = \frac{\partial \phi(z)}{\partial z} \text{ e } \chi'(z) = \frac{\partial \chi(x)}{\partial z}$$

Re - denotes the real part of an expression;
$\bar{z}$ - denotes the complex conjugate of z, i.e,
$\bar{z}$ = x – iy;
υ - Poisson’s ratio;

E - Young modulus;
μ = E / 2(1+υ) – shear modulus

Now consider the following complex eingenvalue functions:

$$\phi(z) = \sum_{n=0}^{\infty} A_n z^{\lambda_n} \text{ e } \chi(z) = \sum_{n=0}^{\infty} B_n z^{\lambda_n + 1}$$

The algebraic manipulation of these functions in equations (5) give, as results, the stresses and displacements formulations:

$$\sigma_y = \sum_{n=1}^{\infty} \frac{n}{2} r^{\left(\frac{n}{2}-1\right)} \times \left\{ \begin{array}{l} a_n^1 \left[\left(2 - \frac{n}{2} - (-1)^n\right) Cos\left(\frac{n}{2}-1\right)\theta + \left(\frac{n}{2}-1\right) Cos\left(\frac{n}{2}-3\right)\theta \right] \\ - a_n^2 \left[\left(2 - \frac{n}{2} + (-1)^n\right) Sen\left(\frac{n}{2}-1\right)\theta + \left(\frac{n}{2}-1\right) Sen\left(\frac{n}{2}-3\right)\theta \right] \end{array} \right\}$$

$$\sigma_{xy} = \sum_{n=1}^{\infty} \frac{n}{2} r^{\left(\frac{n}{2}-1\right)} \times \left\{ \begin{array}{l} a_n^1 \left[\left(\frac{n}{2}-1\right) Sen\left(\frac{n}{2}-1\right)\theta - \left(\frac{n}{2} + (-1)^n\right) Sen\left(\frac{n}{2}-1\right)\theta \right] \\ - a_n^2 \left[\left(\frac{n}{2}-1\right) Cos\left(\frac{n}{2}-3\right)\theta - \left(\frac{n}{2} - (-1)^n\right) Cos\left(\frac{n}{2}-1\right)\theta \right] \end{array} \right\}$$

$$\sigma_x = \sum_{n=1}^{\infty} \frac{n}{2} r^{\left(\frac{n}{2}-1\right)} \times \left\{ \begin{array}{l} a_n^1 \left[\left(2 + \frac{n}{2} + (-1)^n\right) Cos\left(\frac{n}{2}-1\right)\theta - \left(\frac{n}{2}-1\right) Cos\left(\frac{n}{2}-3\right)\theta \right] \\ - a_n^2 \left[\left(2 + \frac{n}{2} - (-1)^n\right) Sen\left(\frac{n}{2}-1\right)\theta - \left(\frac{n}{2}-1\right) Sen\left(\frac{n}{2}-3\right)\theta \right] \end{array} \right\}$$

$$u = \sum_{n=1}^{\infty} \frac{r^{\frac{n}{2}}}{2\mu} \times \left\{ \begin{array}{l} a_n^1 \left[\left(\kappa + \frac{n}{2} + (-1)^n\right) Cos\left(\frac{n}{2}\right)\theta - \left(\frac{n}{2}\right) Cos\left(\frac{n}{2}-2\right)\theta \right] \\ - a_n^2 \left[\left(\kappa + \frac{n}{2} - (-1)^n\right) Sen\left(\frac{n}{2}\right)\theta - \left(\frac{n}{2}\right) Sen\left(\frac{n}{2}-2\right)\theta \right] \end{array} \right\}$$

$$v = \sum_{n=1}^{\infty} \frac{r^{\frac{n}{2}}}{2\mu} \times \left\{ \begin{array}{l} a_n^1 \left[\left(\kappa - \frac{n}{2} - (-1)^n\right) Sen\left(\frac{n}{2}\right)\theta + \left(\frac{n}{2}\right) Sen\left(\frac{n}{2}-2\right)\theta \right] \\ + a_n^2 \left[\left(\kappa - \frac{n}{2} + (-1)^n\right) Cos\left(\frac{n}{2}\right)\theta + \left(\frac{n}{2}\right) Cos\left(\frac{n}{2}-2\right)\theta \right] \end{array} \right\}$$

In this way, the analytical solutions for the stresses and displacements fields were showed and will be used, as follows, in the mixed formulation for the crack element so that the stress intensity factor could be calculated. These solutions were implemented in the “Calcelementosfinitos” program to perform the computational calculus of *K.*

4.2. *Finite Element Formulation for the Stress Intensity Factor Calculation*

Following the formulation showed by Atluri (1986), firstly, the energy functional in matrix form must be applied at element considered:

$$\pi = U - W = \frac{1}{2}\int_V \sigma^T D \sigma \, dV - \int_S u^T \bar{T} dS$$

Where:
U = internal strain energy;
W= work done by surface tractions;
σ = column vector of stresses at a point;
u = column vector of displacements at a point;
D = elastic modulus matrix;
$\bar{T}$ = column vector of prescribed surface tractions;
V = element area;
S = section of boundary over which $\bar{T}$ is applied.

For the problem of an edge crack in an infinite plate, the analytic solution for the stress and displacement fields can be written in a matricial form:

$$[\sigma] = [P][a] \tag{6}$$

$$[u] = [A][a] \tag{7}$$

In the equations (6) and (7) the components of "a" are related to the stress intensity factors K_I and K_{II} according to:

$$a_1 = \frac{K_I}{\sqrt{2\pi}} \qquad \text{and} \qquad a_{n-1} = \frac{K_{II}}{\sqrt{2\pi}}$$

The unknown coefficients "a" can be related to the nodal displacements δ by evaluating equation (7) at every nodal position, thus giving:

$$[\delta] = [\overline{A}][a] \tag{8}$$

Substituting from (6) and (7) in the functional, taking the variation of this functional with respect to the components of the unknown vector [a] and finally applying (8), the structure force / displacement is obtained as follows:

$$[K][\delta] = [F] \tag{9}$$

Where the stiffness matrix is given by:

$$K = \left[\overline{A}^{-1}\right]^T \int_V P^T D^T P dV \left[\overline{A}\right]^{-1} \tag{10}$$

And the equivalent nodal force vector due to applied tractions is:

$$F = \left[\overline{A}^{-1}\right]^T \int_S \overline{A}^T \overline{T} dS \tag{11}$$

Since the crack element (Fig.9) to be used is based on the 8-node parabolic isoparametric element (Fig.8), this restricts the number of nodal displacements [δ] to 16 and hence limits the number of unknown coefficients [a], likewise to 16.

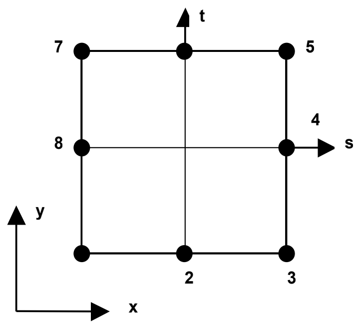

Figure 8. Plane isoparametric element of 8 nodes.

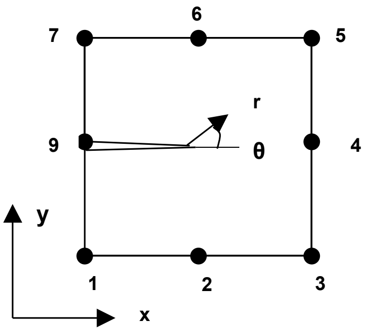

Figure 9. Plane crack element.

Shape functions:
Edge nodes: *i = 1, 3, 5, 7*

$$N_i = \frac{1}{4}(1+ss_i)(1+tt_i)(ss_i+tt_i-1)$$

Midside nodes: *i = 2, 4, 6, 8*

$$N_i = \frac{s_i^2}{2}(1+ss_i)(1-t^2)+\frac{t_i^2}{2}(1+tt_i)(1-s^2)$$

It is necessary to include the three modes of rigid body motions, in the *x, y* and rotational direction. This is done by allocating the last three terms a_i to be the rigid body displacements *θ, x* and *y* and modifying the [$\overline{A}$] matrix accordingly:

$$u = \begin{bmatrix} \overline{A} & y & 1 & 0 \\ & x & 0 & 1 \end{bmatrix} \begin{Bmatrix} a_1 \\ . \\ . \\ a_{13} \\ \theta \\ x \\ y \end{Bmatrix} \tag{12}$$

Since rigid body terms do not induce any stress in the element the corresponding terms in the [*P*] matrix are:

$$\sigma = \begin{bmatrix} P & 0 & 0 & 0 \\ & 0 & 0 & 0 \\ & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} a_1 \\ . \\ . \\ a_{13} \\ \theta \\ x \\ y \end{Bmatrix} \tag{13}$$

As implied in (8), [$\overline{A}$] is a constant matrix being a function of the nodal coordinates. This then leaves the energy integral in terms of the stress matrix [*P*] and the elastic modulus matrix [D] as follows:

$$H = \int_V P^T D^{-1} P dV \tag{14}$$

And [*K*], thus, is the stiffness matrix:

$$K = \left[\overline{A}^{-1}\right]^T H \overline{A}^{-1} \tag{15}$$

Since the last three terms in the [*P*] and [$\overline{A}$] matrices have to be rigid body terms, then the unknown coefficients [a] are defined as: $a_1 - a_7$ the mode I cracking coefficients, $a_8 - a_{13}$ the mode II cracking coefficients and the last three the rigid body components *θ, x* and *y*. It can be seen that this results in the first seven terms in the infinite series solution being used in the representation of the mode I field, but only the first six terms for the mode II field.

It should be noted that the K (stress intensity factors) values are automatically given from the program by merely storing the [$\overline{A}^{-1}$] value and, upon solution of the nodal displacements [δ], applying equation (8) in the form:

$$a = \overline{A}^{-1}\delta \tag{16}$$

Using equation (16) it is easy to obtain:

$$K_I = a_1 \times \sqrt{2\pi} \quad \text{e} \quad K_{II} = a_8 \times \sqrt{2\pi}$$

4.3. ***Finite Elements Modeling Used in the "CALCELEMENTOSFINITOS" Program***

According to the mathematical model showed above, 4 (four) finite elements meshes were studied (Fig. 10-13) for the case of a superficial crack through the thickness of pressure hull, positioned at the inner part of this hull, at 5mm to the welding beam. This position was based on the analytical method presented early in this article (Robles et al. 2000), besides the experiments with real state models made by Dunham (1965). This modeling followed some suggestions presented by Spyrakos (1996) and Steele (1989) according to the mesh construction and her refinement. It is important to note that were implemented transitional elements, matching elements and CST (Constant Strain Triangles) elements in the mesh construction.

Therefore, it was structured a finite element program using FORTRAN language, called "Calcelementosfinitos", to make the stress intensity factors calculation according to the numerical modeling presented in the item 4.2. The crack element formulations to the stress, deformations, as well as the stress intensity factors (*K)* calculation, were implemented having been based on the finite elements programming studies made by Akin (1982), Smith (1982) and Owen (1977). The program showed convergence towards the analytical values (Robles et al. 2000) when the mesh was been refined, as presented in the Figures 14-16.

The difference between the finite element calculation of the stress intensity factor and that of analytical method stayed nearly 2,3%, therefore all of the modelings was accomplished to the maxim depth of 250m (Fig. 14-16). Besides, it is possible to notice that, for the case where the residual stresses as well as the plastification effect in the crack tip were summed up, the analytical method (Robles et al. 2000) proved to be more rigid and so, in the side of safety. Other point that confirm this conclusion is that the critical crack depth calculated by the finite element method, using the "Calcelementosfinitos" program, stayed nearly 5,77mm, a little bit more than the value of 5,45mm finded by the analytical method (Robles et al. 2000) and both in the same order to the experimental results obtained by Dunham (1965) that, for a full-scale model, with dimensions near to those adopted in this article, stipulate as a initial approximation for the critical crack depth (the beginning of the cracks coalescences); 25% of the pressure hull thickness, what symbolizing for the model used in this study, the value of 5,50mm.

It was accomplished three studies separately. The first study represents the application of "mixed" formulation presented in the item 4.2, without the consideration of residual stress and the plastification effect at crack tip. The second study doesn't consider the residual stress either, but implement the effect of plastification at crack tip and the third study take into consideration both the residual stress and the plastification effect. The residual stress was modeled as a thermal stress applied to the nodes, from one surface to another, at equidistant points through the pressure hull thickness (Gonçalves 1987). In this modeling wasn't used a distributed load on the crack surface, this could be accomplished in a posterior study.

It is important to note that the Linear Elastic Fracture Mechanics was adopted in this modeling, considering the region where the plastification occurs at crack tip with dimensions so small in comparison to the dimensions of the crack studied. This modeling can be used in this case, applying the criteria presented by Barson (1987), and developed by Hahn & Rosenfield (1968), where, if the condition $\frac{\left(K_{IC}/\sigma_{ys}\right)^2}{B} \geq 1$ be satisfied, then the plain strain state is achieved and the Linear Elastic Fracture Mechanics can be used in the modeling, adopting, in this way, a region of plastification at crack tip with dimensions so small, which effect can be simulated by the Irwin (1957) correction. The criterion used above is less rigid than that adopted by the ASTM (1985), but is more realistic and, therefore was utilized in this study. The correction cited above must summed up to the original crack depth, simulating the effect of plastification at crack tip:

$$\Delta a = \frac{1}{6\pi}\left(\frac{K_t}{\sigma_{esc}}\right)^2 x1000(mm) \tag{17}$$

Where:
K_t: Total stress intensity factor without plastification correction (MPa);
σ_{esc} : Yielding stress of HY-80 steel MPa;

Δa : correction to the crack depth due to the effect of plastification at crack tip (mm).
B : Material thickness.

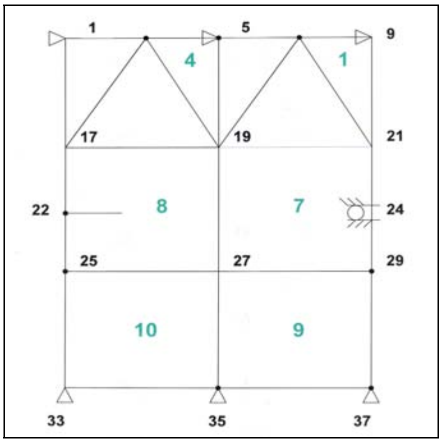

Figure 10. Modeling using mesh 1.

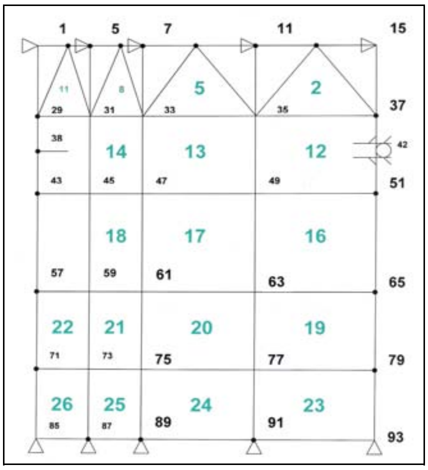

Figure 11. Modeling using mesh 2.

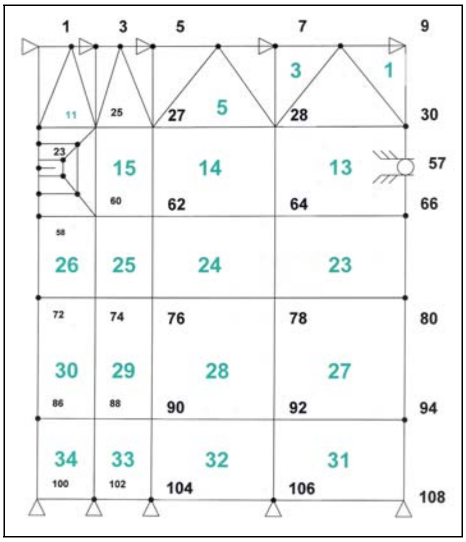

Figure 12. Modeling using mesh 3

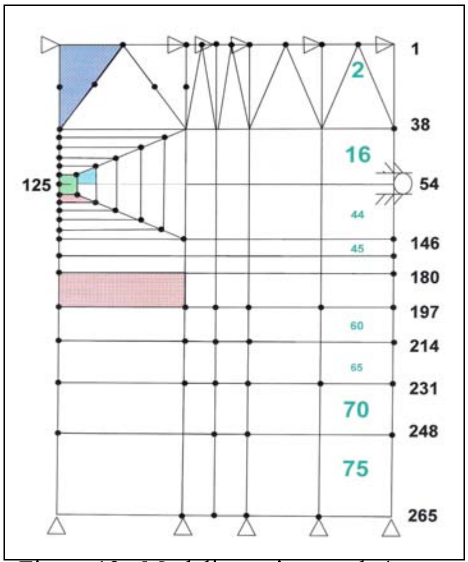

Figure 13. Modeling using mesh 4.

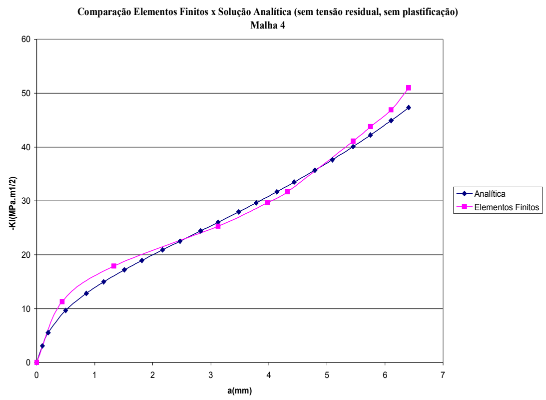

Figure 14. Comparison between the finite element method and the analytical method results to the stress intensity factors (without residual stress, without plastification).

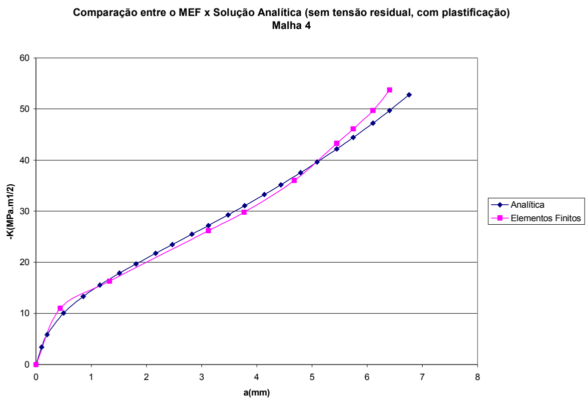

Figure 15. Comparison between the finite element method and the analytical method results to the stress intensity factors (without residual stress, with plastification).

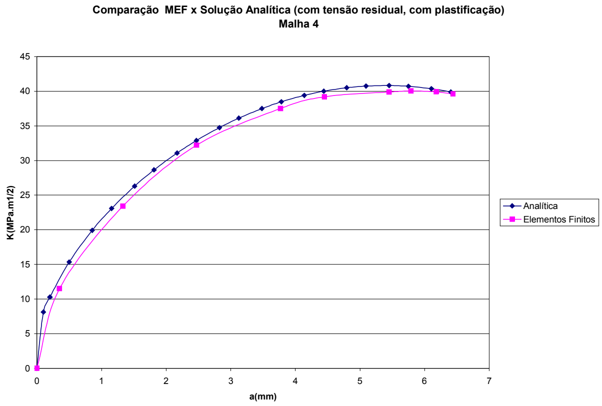

Figure 16. Comparison between the finite element method and the analytical method results to the stress intensity factors (with residual stress, with plastification).

4.4. ***Modeling Using the ANSYS Program for the Stress Intensity Factor Calculation***

The validation of the analytical method results and those obtained by the "Calcelementosfinitos" program, was accomplished by a bidimensional modeling using the finite elements program ANSYS (2004).

The modeling of the singularity ($1/\sqrt{r}$) in this study used the PLANE 82 element, which is a quadrilateral element with 8 nodes, but with the midside nodes positioned at 1/4 of element side length, as well as using the merging of three nodes at his vertice, thus, forming a triangular element (Fig.17-18). This element is known as QPE (Quarter-Point Element).

The results finded, for a modeling doesn't considering the residual stress and the other considering, as well as, computing the effect of plastification at crack tip in both situations, were presented in the Figures 19-20. It can be observed that the results are close to each other, showing a variation range of 2,4 to 4,5% between these values.

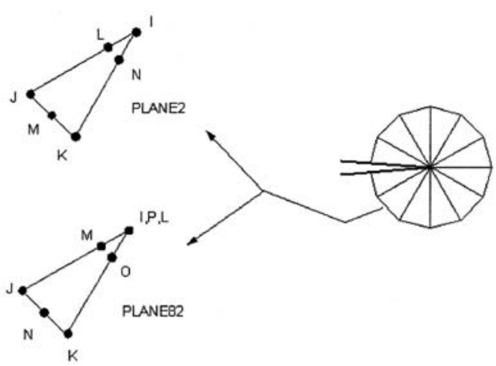

Figure 17. Quarter-Point Element – (QPE)

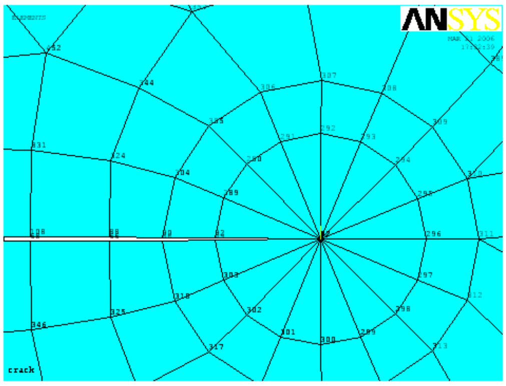

Figure 18. Modeling of the crack tip.

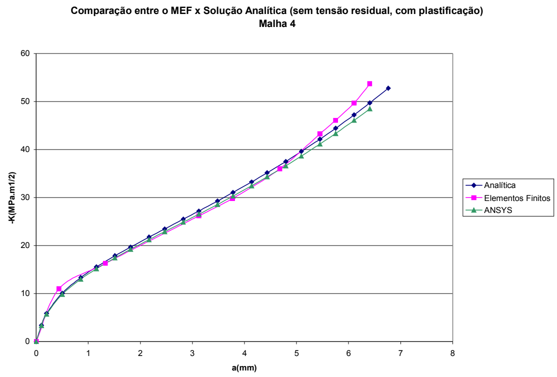

Figure 19. Comparison of results, without residual stress and with plastification.

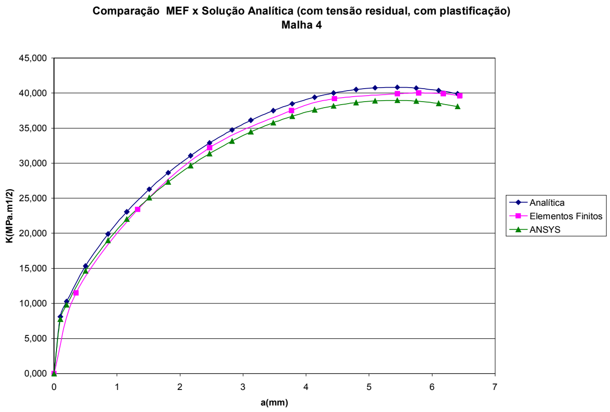

Figure 20. Comparison of results, with residual stress and with plastification.

5. CONCLUSION

A model to evaluate the fatigue of a submarine pressure hull was proposed, based on the Linear Elastic Fracture Mechanics theory. This model was applied to a typical region (cone-cone) of the submarine, in the area of intersection of two weld beams, one longitudinal, the other circumferential, which according to the Kilpatrick (1986) experiments, is the critical region to be analysed.

An analysis of the model parameters shows that the residual stress distribution has a great influence in the fatigue process of submarine pressure hull.

The process of calculus of fatigue operational life of submarine pressure hull has the beginning with the calculation of the stress intensity factors. From these, his variations are calculated as functions of the submarine operational depths, and then is applied the Paris (1963) law to obtain the number of cycles to fatigue of pressure hull. Thus, in this study, was presented a numerical modeling by finite elements, through "Calcelementosfinitos" program, as well as a modeling using the ANSYS (2004) program, so that the results showed in the analytical method and the experiments of Dunham (1965) could be validated to the stress intensity factors values.

These results were so close to each other, with a maxim variation about 5%, what is very reasonable in terms of engineering precision. The values of crack critical depth calculated by the analytical method (Robles et al. 2000) and by the finite elements method presented in this article, were close to each other too, being these values 5,45 and 5,77mm, respectively. These values could be validated by the experiments of Dunham (1965) with full-scale models with dimensions very similar to that adopted in this study.

6. REFERENCES

L.B.R. ROBLES & M.A.BUELTA & E.GONÇALVES & G.F.M.SOUZA, "A Method for the Evaluation of the Fatigue Operational Life of Submarine Pressure Hulls", Int. Journal of Fatigue, v. 22, issue 1, pp. 41-52, jan. 2000.

HELLER, S. R. "A Personal Philosophy of Structural Design of Submarine Pressure Hulls", Naval Engineers Journal, v. 74, n.2, pp. 223-233, May 1962.

DUNHAM, F. W. "Fatigue Testing of Large-Scale Models of Submarine Structural Details", Marine Technology, v.2, n.3, pp. 299-307, July 1965.

KILPATRICK, I. M., "The Fatigue Characteristics of Submarine Structures Subjected to External Pressure Cycling". International Conference Held at the Admiralty Research Establishment, Procedings Dunfermline, s.ed, Dunfermline, 1986.

ANSYS – "Finite Element Analysis System", Manuais de Utilização do Programa, versão 7.1 e 10.0, 2002 e 2004.

BARSON, J. M. & ROLFE, S. T., Fracture and Fatigue Control in Structures. 2 ed. Englewood, N. Jersey, Prentice-Hall, 1987.

BROEK, D., Elementary Engineering Fracture Mechanics. 4. ed. Netherlands, Martinus Nijhoff, 1986.

PARIS, M., P. C.& ERDOGAN, F. Basic Eng. Trans. ASME 85. New York, ASME, p.528, 1963.

FAWKES, A. J.& OWEN, D. R. J.& LUXMOORE, A. R. "An Assesment of Crack Tip Singularity Models for Use with Isoparametric Elements", Engineering Fracture Mechanics, v. 11, pp.143-159, 1979.

MUSKHELISVILLI, N. I. "Some Basic Problems of the Mathematical Theory of Elasticity", Groningen, Holanda, Noordhoff ,1953.

ATLURI, S. N.,Computational Methods in the Mechanics of Fracture. 3 ed. New York, North-Holland, 1986.

SPYRAKOS, C., Finite Element Modelling in Engineering Practice. 3 ed. New Jersey, Algor Publishing Division, 1996.

STEELE, J. M., Applied Finite Element Modelling. 3 ed. New York, Marcel Dekker, 1989.

AKIN, J. E., Application and Implementation of Finite Element Methods. 4 ed. New York, Academic Press, 1982.

SMITH, I. M., Programming the Finite Element Method. 2 ed. New York, John Wiley & Sons, 1982.

HINTON, E. & OWEN, D. R., Finite Element Programming. 3 ed. Londres, Academic Press, 1977.

GONÇALVES, E., Análise de Fraturas em Estruturas Cilíndricas Soldadas. Tese de Livre-Docência, Departamento de Engenharia Naval/USP, São Paulo, SP,Brasil,1987.

G.T. HAHN & A.R. ROSENFIELD, "Sources of Fracture Toughness: The Relation Between KIC and the Ordinary Tensile Properties of Metals," ASTM STP-432, American Society of Testing Materials, pp. 5-32, 1968.

IRWIN, G. R. "Analysis of Stresses and Strains Near the End of a Crack Traversing a Plate", Trans. ASME, J. Applied Mechanics, 1957.

AMERICAN SOCIETY FOR TESTING AND MATERIAL. "Standard Test Method for Plane Strain Fracture Toughness of Metallic Materials", ASTM – E399-83, Annual Book of ASTM Standards, EUA, v. 3.01, 1985.

7. RESPONSIBILITY NOTICE

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