International Journal of Research

International Journal of Research (IJR)

e-ISSN: 2348-6848, p- ISSN: 2348-795X Volume 2, Issue 06, June 2015

Available at http://internationaljournalofresearch.org

Numerical modeling and analytical validation of stress and stress intensity factor for SENB bending specimen of P265GH steel material

M.Lahlou[1]; A.Hachim[2]; N.Mouhib[1]; H.Ouaomar[1]& M. El Ghorba[1]

[1] Laboratory of Control and Mechanical Characterization of Materials and Structures, National Higher School of Electricity and Mechanics, BP 8118 Oasis, Hassan II University, Casablanca, Morocco.

Lahloumohammed89@gmail.com

[2] ISEM/Higher Institute of Maritims Studies, Laboratory of Mechanics, Km 7 Road El Jadida Casablanca Morocco

Abstract

Thesudden failureof mechanical partsgenerally occursin areas wherethere is a default because of the non-uniformity of the stress distribution particularly in the vicinityof the defect.Therefore, notchgeneratesan increaseof the stressin its vicinity.

The objectiveof this work is to establisha numericalfinite element modelingofa bendingspecimen atthree points (SENB) usingCASTEM2013computer code. The studiedmaterial isP265GHsteel used inpressure vessels.

The results found show that the stresses exhibit a parabolic trend with a minimum value (xmin = 7 mm). The maximum stresses are at the bottom of notch level and at the application point of force, it is also noticed that the stress intensity factor increases by increasing the length of the crack.

Fatigue tests are long, expensive and difficult to achieve. The numerical study proposed in this paperhas for purpose the simulation of this delicate situation.

Key Words:

SENB; CAST3M2013**;** P265GHsteel; numerical study; pressure vessels

1. Introduction

Computer facilities have become an essential tools now a days and specifically in industrial sectors. For reasons of competitiveness and scientific development, companies are constantly looking for predicting thelifetime andconsidering solutionsin the designand maintenance ofpartsand structures.Thefinite element methodsolves theseproblemsespeciallyregardingmechanical problems. [1]

In metallicstructures, cracksare initiatedmostlyatnotches; this governs the crack initiation or propagation and therefore a loss of energy. [2]

In industry, for economic or security reasons we try to know the degree of defects harmfulness and the residual life of structures. This requires the establishment of models based on fracture mechanics.

These include the works of A. HACHIM [3][4] on the numerical study of a double-notched Steel S355, studied the behavior of the material in the presence of the fault. Y.HIROSHI [5] studied the critical stress intensity factor on simple notched specimens.

This work is devoted to finite element analysis of a bending specimen (SENB).

2. Experimentation

To extract the mechanical characteristics of the P265GH steel used in our program, tensile tests of standard specimens (Figure1) were conducted in different directions of rolling (longitudinal and transversal)[6]. The test curves showing the stress versus elongation are given in Figure 2:

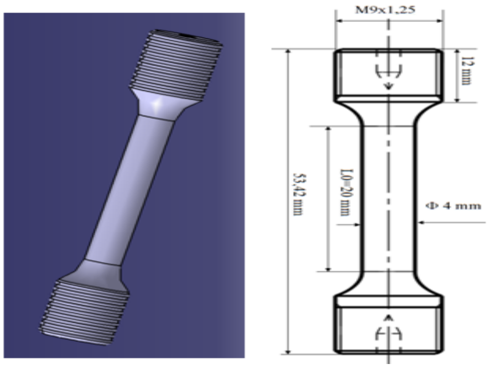


Figure 1: Dimensions of the standard test specimen

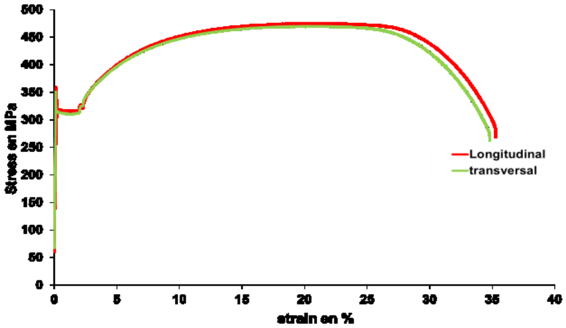


Figure 2: Stress strain curve test

By comparing the mechanical characteristics of the specimen in both rolling directions, it is found that there is a negligible difference between the two curves. The mechanical characteristics of P265GH steel, at the ambient temperature, are reported in the table1:

Table 1: Mechanical properties of the material

Young's modulus *E* *(MPa□ □*	elastic limit: $\square_e$ (□Pa)	Breaking stress: $\square_g$ (□Pa)	Elongation %	Poisson's ratio ν
2.10^5	320	470	35	0,3

We notice that the elongation is about 35%, which is higher than 14% required by the CODAP [7]. Therefore, this P265GH steel used is well adapted for pressurized structures.

3. Numerical modeling

The calculation code Cast3m 2013 isused to developa finite element model for the analysis of the specimen (SENB) subjected to a bending stress in three points.

3.1. Geometry

The geometry and dimensions of thestudied specimen are illustrated in Figure 3. The study is restricted to the mode I,

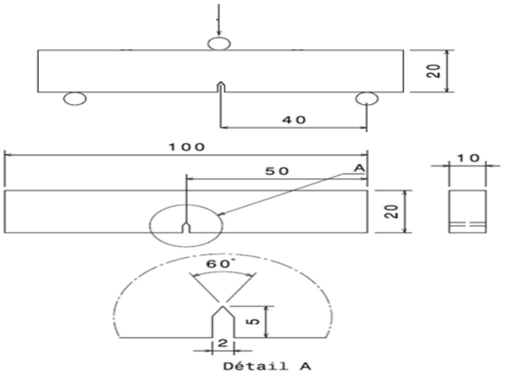


Figure 3: Dimensions of the studied specimen (mm)

3.2. Mesh and boundary conditions

By taking into consideration the symmetry of the problem, only half of the test specimen is discretized. The numerical results are intended for mechanical analysis of fracture, special attention is paid to mesh particularly crack and its vicinity (Mesh Refinement using Barsoum elements)[8]. Details of the mesh are illustrated in Figure 4 and Figure 5.

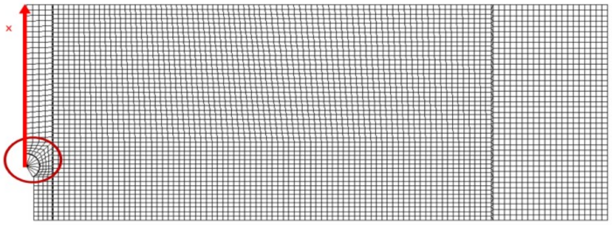


Figure 4: Mesh to half of the specimen SENB

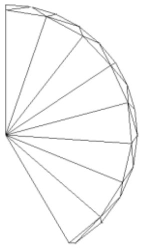


Figure 5: Mesh the vicinity of the notch

3.3. Loading

The simulated load is a bending stress at three points along the vertical axis of the test part (x axis). Successful loads are calibrated so that the applied force to be 8800N and 22000N respectively.

4. Results & discuss

4.1. Evolution of the stress in the ligament

The curves in Figure 6 illustrate the evolution of stress in the ligament of the specimen, ie, along the x axis of figure 4(origin at the crack bottom) for the two levels of applied loads F = 8800N and 22000 N.

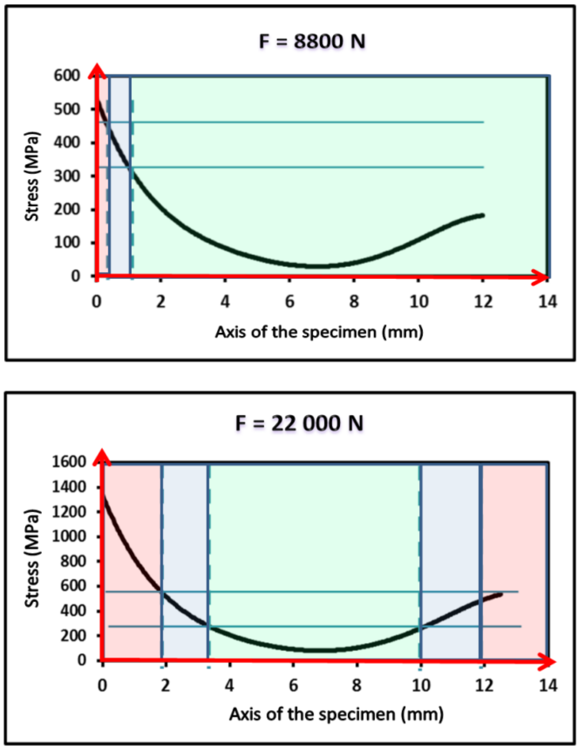


Figure 6: Evolution of the stress in the ligament for the two levels of loads (F = 8 800N and 22000N)

- For F=8800N three zones are identified:

• The first zone [0, 0.2mm], near the bottom of the notch corresponds to stress values within the interval [470, 540 MPa]. The maximum stress σ_{max} = 540 MPaexceeds the strength of the material, Then it decreases until reaching the breaking stress σ_g = 470 MPa, which leads to local failure of the bonds and so a crack propagation.

• The second zone [0.2, 1.2mm], corresponds to stress values within the interval [320, 470MPa]. The stress decreases up to the elastic limitresulting a plastic deformation in this zone.

• The third zone (x> 1.2 mm) corresponds to stress values within the interval [25,320MPa] .We note that the curve follows a parabolic change with a minimum value (x_{min} = 7 mm) corresponding (σ_{min} = 25 MPa) .The stress is lower thanelastic limit, this undergo a purely elastic deformation.

- For F=22000N five zones are distinguished :

• the first zone [0, 1.9mm], near the bottom of the notch corresponds to stress values within the interval [470, 1350 MPa]. The maximum stress is (σ_{max} = 1350 MPa) exceeds the strength of the material, after it decreases to the breaking limitσ_g = 470 MPa, which leads to local failure of the bonds and so a crack propagation.

• The second zone [1.9, 3.5mm], corresponds to strain values within the interval [320, 470MPa] the stress decreases to the elastic limit resulting a plastic deformation in this zone.

• The third zone [3.5, 9.9mm], corresponds to strain values within the interval [60,320MPa] .We note that the curve follows a parabolic change with a minimum (x_{min} = 7 mm) corresponding to(σ_{min} = 60 MPa) .the stress is lower than elastic limit, so this zone undergo a purely elastic deformation.

- The fourth zone [9.9, 11.9mm] which corresponds to [320, 470MPa] the stress increases up to the ultimate tensile strength which corresponds to (x = 9,8mm) this area is permanently deformed

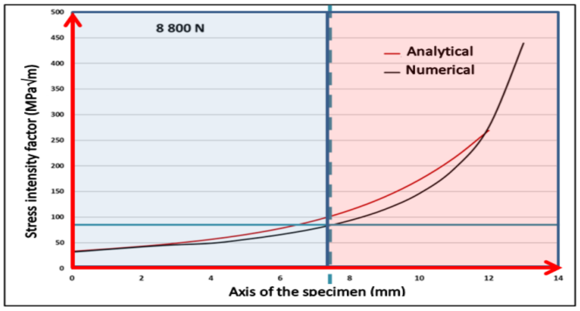
- The fifth zone (x> 11.9mm). The stress increases exponentially (point of application of the constraint). The stress is greater than breaking strength.

4.2. Evolution of the stress intensity factor

The curves of Figure 7 show the numerical evolution and analytical stress intensity factor (following equation (1)) in the ligament of specimen for both loading levels: F = 8800N and 22000N

$$K_I = \frac{F}{t\sqrt{w}}[11,58(\frac{a}{w})^{1/2} - 18,42(\frac{a}{w})^{3/2} + 87,18(\frac{a}{w})^{5/2}$$
$$-150,66(\frac{a}{w})^{7/2} + 153,30(\frac{a}{w})^{9/2}] \qquad (1)$$

With:
KI = stress intensity factor
F = applied force
t = thickness of the specimen 10mm,
w = width of the specimen 20mm.



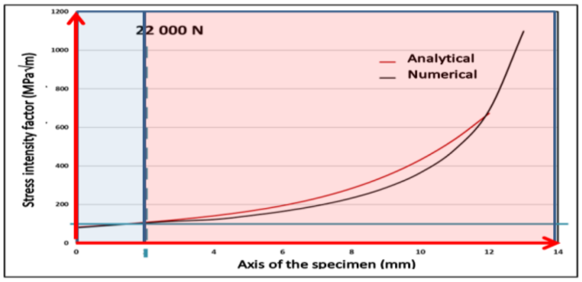


Figure 7: The stress intensity factor for two forces (F = 8800N and 22000N)

The analysis of the curves in Figure 7 shows that there is a significant increase in the stress intensity factor as a function ofcrack length and applied stress.

The results of the stress intensity factor obtained numerically are in perfect agreement with those obtained analytically which validates our study.

4.3. Evolution of critical notch length

Figure 8 represents the evolution of the notch length according to critical loading level

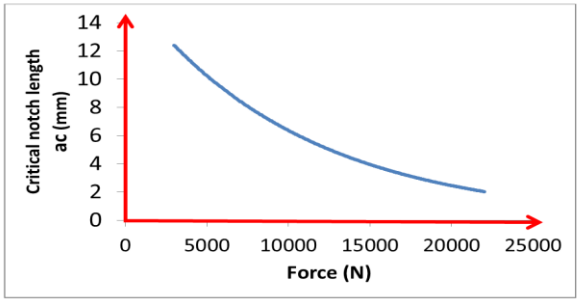


Figure 8: Evolution of thenotch length as function of critical force

The curve analysis shows that to avoid brutal crack propagation it is necessary to not exceed a force of F = 3000 N so that the critical crack length is greater than the rest of the specimen (15 mm)

An increase in strength leads us to careful inspection of crack length

5. Conclusion

In the sectors of unsafe structures such as pressure vessels and in the presence of defect, it is essential to detect precisely the degree of defect harmfulness. Numerical finite element modeling method is an extremely efficient tool to address this issue.

We performed a numerical model using Cast3m 2013 on a bending specimen at three points (SENB) to study the evolution of the stress in the ligament of the specimen for two loading levels (F = 8800N and 22 000N). For all results we find that the two stress curves exhibit a parabolic trend with a minimum value (x_{min} = 7 mm). The maximum stresses are at the bottom of the notch and at the point of force application.

The evolution of stress intensity factor is studied numerically and validated analytically. The fact that we could validate this study leads us to think that it is possible to apply in our way of doing some cases that it would be difficult to achieve experimentally.

Références

[1.] Martin Kempeneers, (2006) *éléments finis statiquement admissibles et estimation d'erreur par analyse duale*, Mémoire de thèse doctorat Université de Liège Faculté des Sciences Appliquées,

[2.] Damien Fournier, (2011) *Analyse et Développement de Méthodes de Rafinementhp en Espace pour l'Equation de Transport des Neutrons*, Mémoire de thèse doctorat Université de Provence Marseille,

[3.] A. HACHIM,(2012) *Numerical Evaluation and Experimental Validation of Stress Concentration and Crack Propagation a Bi-notched Specimen of Steel S355*, International Journal of Research and Reviews in Mechatronic Design and Simulation (IJRRMDS) Vol. 2, No. 3, PP 58-64

[4.] A. HACHIM, (2012)*Numerical evaluation of stress triaxiality at the top of notch for a specimen steel notched bi-S355*, International Journal of Engineering and Science, Vol. 1, Issue 1, PP 088-93

[5.] Hiroshi Yoshihara,(2014)*Mode I Critical Stress Intensity Factor of MediumDensity Fiberboard Obtained by Single-Edge Notched Bending Test* ,Original scientifi c paper • Izvorniznanstveni rad. Accepted – prihvaceno,

[6.] LAHLOU, M., et al. "*Numerical modeling and analytical validation of stress and stress intensity factor for SENT tensile specimen of P265GH steel material*."IPASJ International Journal of Mechanical Engineering (IIJME):Volume 3, Issue 4, April 2015 pp. 042–048

[7.] CODAP: *Code de Construction des Appareils à Pression non soumis à la flame* 2005

[8.] Barsoum, "*Furthur application of quadratic isoparametric elements to linear fracture mechanics of plate bending and general shells*". Int.J.Num.Meth,Engng , 11,167-169. (1976).