Publishing House
AKAPIT

COMPUTER METHODS IN MATERIALS SCIENCE
Informatyka w Technologii Materiałów

Vol. 11, 2011, No. 2

COMPUTATIONAL APPROACH FOR PREDICTING THE CRACK PROPAGATION OF STAINLESS STEEL PLATE USING DAMAGE MECHANICS

BYUNG-MOON YOO, CHI-SEUNG LEE, MIN-SUNG CHUN, MYUNG-HYUN KIM, JAE-MYUNG LEE*

Department of Naval Architecture and Ocean Engineering, Pusan National University, Busan 609-735, Republic of Korea
**Corresponding author: jaemlee@pusan.ac.kr*

Abstract

The aim of this study is development of numerical method for the prediction for crack propagation of stainless steel plate under cryogenic temperature. Stainless steel is one of the most functional materials at relatively wide temperature ranges. It has strong non-linearities on the mechanical properties under cryogenic temperature such as discontinuous hardening phenomenon induced by the phase transformation, among others. The nonlinear hardening affects significant change of material characteristics, i.e., strength, deformation and fracture phenomenon. This induces some difficulties on the precise evaluation of structural capacity of stainless steel based structures. In the present paper, the crack propagation characteristics of the stainless steel plate under cryogenic temperature have been simulated using a new numerical technique based on a newly proposed damage-coupled constitutive model and orient-purpose user subroutine applicable to commercial FEA code ABAQUS. The numerical results are compared with the experimental results of tensile test and crack propagation tests under cryogenic temperature.

Key words: Stainless steel plate, Crack propagation, Damage coupled constitutive model, Continuum damage mechanics, Finite element analysis, Cryogenic temperature

1. INTRODUCTION

The stainless steel 304(AISI 304 or SUS 304) is one of the most widely used materials for cryogenic temperature application in various industrial fields because it holds superior material performances under cryogenic temperature; high strength, more ductility and toughness. While lots of metallic materials show low temperature embrittlement, austenitic stainless steels do not show noticeable material changes in the cryogenic temperature range (Yoshimura and Shimizu, 1982). Due to these advantages, it can be found that various application examples such as cryogenic control valve, cryogenic pressure valve and LNG cargo system, are available in the industries. The most important material characteristics of ASS is TRIP (Transformation induced plasticity), which is non consistent plastic behavior caused by strain induced phase transformation from austenite to martensite (Ohio, 1976). During the TRIP process, the 2^{nd} hardening phenomenon occurs and it leads to strength increasing (Park et al., 2010). However, since this material nonlinearity, so- called 2^{nd} hardening is strong environment dependent phenomenon, any consistent methodology applicable for numerical evaluation for the purpose of design support is not available. Considering above mentioned matters, in order to design and construct the robust stainless steel structures, which is used under cryo-

 ISSN 1641-8581

genic temperature, it is recognized that these nonlinear material behaviors must be identified. The failure related features such as crack propagation are one of the crucial factors to be investigated for the safe design as well.

In the present study, nonlinear material behaviors as well as failure features under cryogenic temperature are precisely identified based on visco-plastic and damage mechanics approach. In the previous study, Yoo (2010) established a constitutive model for strain-induced martensite transformation, which enables to describe the temperature and strain rate dependence of the transformation phenomena. A unified constitutive equation for strain-induced phase transformation based on Bonder-Partom model by function of the fraction of martensite was proposed. The proposed constitutive equation was verified using cryogenic tensile tests under various temperature and strain rates. In this study, finite element analysis is carried out in conjunction with the developed constitutive equation. For the application into structural analysis, ABAQUS user defined subroutine UMAT is developed. Constitutive equation is implemented into an implicit formulation using definition of the algorithmic tangential stiffness (ATS) tensor. Computational simulation for crack propagation is carried out to verify the usefulness, and obtained results are compared with the experimental results.

2. CONSTITUTIVE EQUATIONS

The strain decomposition principle is postulated in the formulation, namely;

$$\dot{\varepsilon}_{ij} = \dot{\varepsilon}^{e}_{ij} + \dot{\varepsilon}^{(ie)}_{ij} \tag{1}$$

where $\dot{\varepsilon}_{ij}$ is the total strain rate, $\dot{\varepsilon}^{e}_{ij}$ is the elastic part of the strain rate and $\dot{\varepsilon}^{(ie)}_{ij}$ is the inelastic part of the strain rate. The elastic parts are provided by time derivative of the Generalized Hook's law.

$$\dot{\sigma}^{e}_{ij} = D_{ijkl}\dot{\varepsilon}^{e}_{kl} \tag{2}$$

$$D_{ijkl} = 2G\left[\frac{1}{2}(\delta_{ik}\delta_{jl} + \delta_{il}\delta_{jk}) + \frac{v}{1-2v}\delta_{ij}\delta_{kl}\right] \tag{3}$$

where G is the shear modulus and δ_{ij} is the Kronecker delta function. The inelastic strain rate components are represented by the isotropic form of the Prantl-Reuss flow.

$$\dot{\varepsilon}^{(ie)}_{ij} = \lambda s_{ij} \tag{4}$$

$$s_{ij} = \sigma_{ij} - \frac{1}{3}\sigma_{kk}\delta_{ij} \tag{5}$$

The plastic multiplier λ is defined as;

$$\lambda = \sqrt{\frac{D_0^2}{J_2}\exp\left[-\left(\frac{Z^2}{3J_2}\right)^N\right]} \tag{6}$$

where D_0 and N are material parameters and J_2 is the second deviatoric invariant and Z is hardening state variable. Considering the isotropic hardening, evolution equation of hardening state variable Z can be written as;

$$\dot{Z} = m_1\lambda\left[Z_a(1-f^a) + Z_m f^a - Z\right]2J_2 \tag{7}$$

where m_1 is hardening rate coefficient for isotropic hardening, f_a is the volume fraction of martensite for stainless steel 304 (Tomita and Iwamoto, 2001), Z_a is limit value for isotropic hardening of pure austenite state and Z_m is limit value for isotropic hardening of full martensite state. In order to express material degradation, visco-plastic damage model by Bodner and Chan (1986) was adopted. The hardening variable Z in Eq.6 can be replaced by $Z(1-\omega)$, where damage variable ω from 0 to 1. The evolution of material damage can be identified as evolution form;

$$\dot{\omega} = \frac{P}{H}\left[\ln\left(\frac{1}{\omega}\right)^{\frac{P+1}{P}}\right]\omega\dot{Q} \tag{8}$$

where, P and H are the material parameters. Using the multi-axial stress function suggested by Hayhurst (1972), triaxiality of stress function can be expressed as;.

$$\dot{Q} = [\alpha\sigma^{+}_{\max} + \beta\sqrt{3J_2} + \gamma I_1^{+}]^r \tag{9}$$

where $\sigma^{+}_{\max}$ is the maximum tensile principal stress, I_1^{+} is the first stress invariant, r is material constant, α, β and γ are the stress function control parameters satisfying $\alpha + \beta + \gamma = 1$. Hayhurst (1977) suggested as $\beta - 0.75$ and $\lambda = 0.25$ by experimental results.

3. METHOD OF ANALYSIS

It is well known that the commercial FE code ABAQUS allows to define material state at every integration points within FE element. In this regards, ABAQUS user defined subroutine UMAT is commonly used to implement a specific constitutive model into finite element analysis. This can give significant advantage in the FE analysis especially in the case of lack of adequate material model within ABAQUS libraries.In the present study, the developed constitutive model for stainless steel was implemented into ABAQUS/standard platform.

3.1. Abaqus user subroutine UMAT

The subroutine UMAT provides the information of the state variables, such as strain, stress, as the form of incremental values per each increment step. All of these incremental values must be updated in the calculation procedure by the definition of the solution-dependent state variables (SDV). SDV is initialized at the subroutine SDVINI-ABAQUS. The optimal size of time increment Δt and corresponding strain increment, which are computed at the end of the previous time increment, are determined using the Jacobian matrix (ABAQUS User's Manual, 2009). Numerical integration scheme used in this study is described as follows.

The incremental strain is given by;

$$\Delta \boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}_{n+1} - \boldsymbol{\varepsilon}_n \tag{10}$$

The corresponding elastic trial stress tensor is computed from;

$$\boldsymbol{\sigma}_{n+1}^{trial} = \boldsymbol{\sigma}_n + \mathbf{D} : \Delta \boldsymbol{\varepsilon} \tag{11}$$

The updates the stresses can be written as;

$$\boldsymbol{\sigma}_{n+1} = \boldsymbol{\sigma}_{n+1}^{trial} - \Delta\lambda\, \mathbf{D} : \mathbf{s}_{n+1} \tag{12}$$

The scalar $\Delta\lambda$ according to the definition $\Delta\lambda = \lambda \Delta t$ can be written with damage;

$$\Delta\lambda = \Delta t \sqrt{\frac{D_0^2}{J_2} \exp\left[-\left(\frac{Z^2 (1-\omega)^2}{3J_2} \right)^N \right]} \tag{13}$$

At the end of the increment, the time history values for every solution dependent state variables are stored using the STATEV array. The follow equations are representative state variables for the constitutive equation.

$$Z_{n+1} = Z_n + m_1 \Delta\lambda \left[Z_a (1 - f^a) + Z_m f^a - Z \ \right] 2J_2 \tag{14}$$

$$\omega_{n+1} = \omega_n + \frac{P}{H} \left[\ln\left(\frac{1}{\omega_n}\right) \right]^{\frac{P+1}{P}} \omega_n \dot{Q} \Delta t \tag{15}$$

$$f_{n+1}^{\alpha} = f_n^{\alpha} + (1 - f_n^{\alpha})(A\dot{\varepsilon}^p + B\dot{g})\Delta t \tag{16}$$

The ABAQUS uses the Newton-Raphson algorithm to solve the stress and hardening equations. Therefore, the UMAT must provide the material Jacobian matrix, $\partial \Delta\sigma_{ij} / \partial \Delta\varepsilon_{kl}$, for adopted constitutive model. Equation 17 is material Jacobian matrix of present model.

$$\frac{\partial \Delta\sigma_{ij}}{\partial \Delta\varepsilon_{kl}} = D_{ijkl}^a - \frac{1}{\Omega} D_{ijmn}^a s_{mn} \psi_{op} D_{opkl}^a \tag{17}$$

The follow equations are details of equation 17.

$$D_{ijkl}^a = \left[D_{ijkl}^{-1} + \Delta\lambda \frac{\partial s_{ij}}{\partial \sigma_{kl}} \right]^{-1} \tag{18}$$

$$\Omega = \frac{1}{\Delta t} + w \tag{19}$$

$$w = \psi_{ij} D_{ijkl}^a s_{kl} + \frac{\partial \phi}{\partial Z} D^{al} T \tag{20}$$

$$T = -(Z_a(1 - f^a) + Z_m f^a - Z) s_{ij} s_{ij} \tag{21}$$

$$\phi = \sqrt{\frac{D_0^2}{J_2} \exp\left[-\left(\frac{Z^2 (1-\omega)^2}{3J_2} \right)^N \right]} \tag{22}$$

$$\psi_{ij} = \frac{\partial \phi}{\partial Z} \left(\frac{\partial Z}{\partial \sigma_{ij}} - \Delta\lambda D^{al} \frac{\partial T}{\partial \sigma_{ij}} \right) + \frac{\partial \phi}{\partial J_2} \frac{\partial J_2}{\partial \sigma_{ij}} \tag{23}$$

$$D^{al} = \left[m_1^{-1} + \Delta\lambda \frac{\partial T}{\partial Z} \right] \tag{24}$$

3.2. Modeling material failure

In this study, the element weakening method which reduces the element stiffness and stress close to zero was adopted for simulation of crack propagation (Mishnaevsky, 2005). For each calling of UMAT, per each integration point, the damage parameter ω is evaluated using the SDV. When the failure criterion is satisfied, another subroutine FAILURE calculates the material degradation using damage variable.

4. NUMERICAL EXAMPLES

4.1. Verification constitutive model

In order to verify the developed UMAT, the computational results are compared to the experimental results for the case of tensile test.

The configuration of numerical model for tensile test is shown in figure 1. The comparison of the uniaxial tensile behavior under 77 K, 213 K and 273 K temperature is shown in figure 1 as well. It can be found that the numerical results obtained by the developed UMAT coincide very well with the experiments results (Tomita and Iwamoto, 2001).

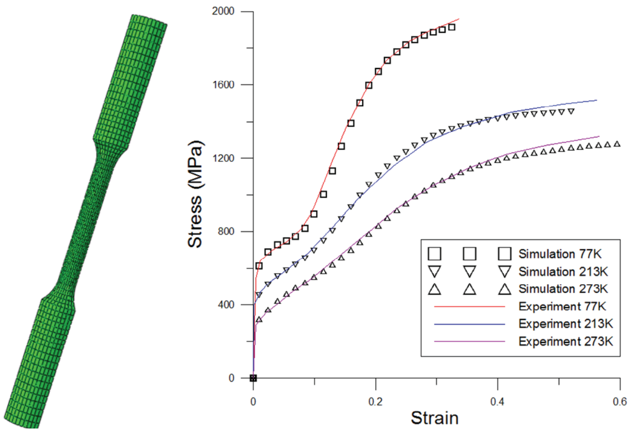

Fig. 1. Simulation of uniaxial tensile test by ABAQUS UMAT.

A special purpose grid lines were fabricated on the specimen to measure the crack initiation and propagation measurement (see figure 2). All of the results were captured as digital data using high speed camera system.

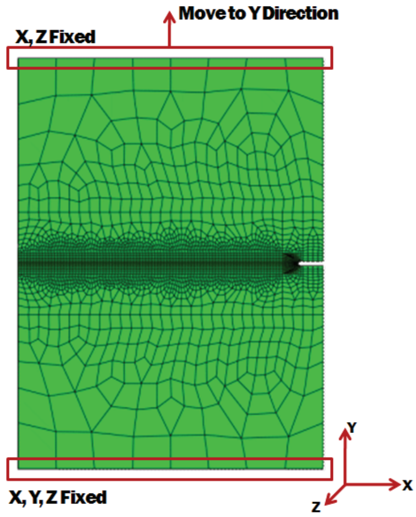

Fig. 3. Finite element model of stainless steel plate specimen for ABAQUS.

Figure 3 shows the FE model for stainless steel plate including the loading and boundary condition. The element type is three-dimensional hexahedron element C3D8R. The number of element is 16485

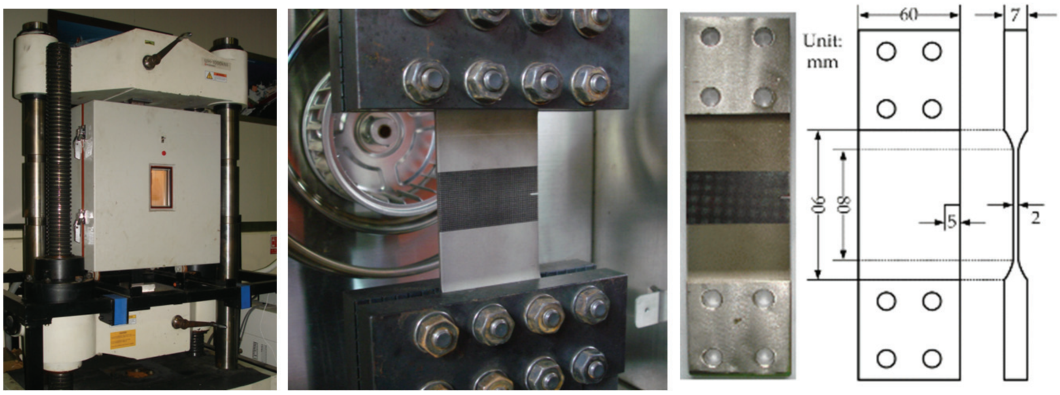

Fig. 2. Photograph of UTM and stainless steel plate specimen.

and the minimum element length is approximately 0.1 mm nearby crack tip line. Figure 4 shows the sample of the obtained result of simulation under 113 K. As shown in figure 4, it can be found that the martensite transformation is concentrated around the crack tip where stress concentration occurs. Figures 5 and 6 show the comparison of the relationship between applied load versus displacement. Compared with figure 5, noticeable 2nd hardening can be found on the tensile behavior for low temperature range (113 K) (see figure 6). It is noted that the

4.2. Cracked plate problem

In order to apply the developed UMAT to crack propagation problem of stainless steel plate, the crack propagation test has been carried out. Figure 2 shows the test specimen and dimension. Universal Test Machine (UH-1000kN, SHMADZU) equipped with a cryogenic chamber. The cryogenic chamber which is controlled by digital system can be maintained constant cryogenic temperature up to -200°C.

completely different tensile behaviors can be simulated by a unified constitutive model. It would be concluded that the proposed constitutive model is very useful for the description of material behavior as well as structural behavior.

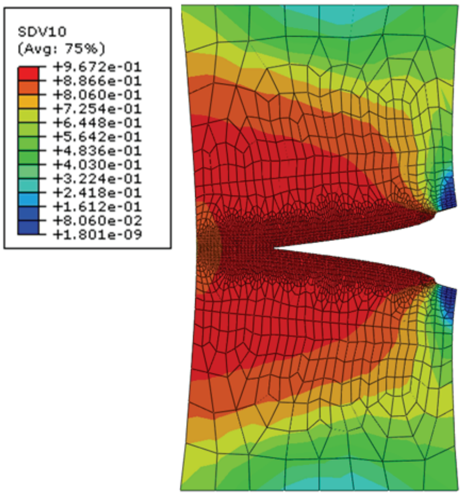

Fig. 4. *Martensite volume fraction contour.*

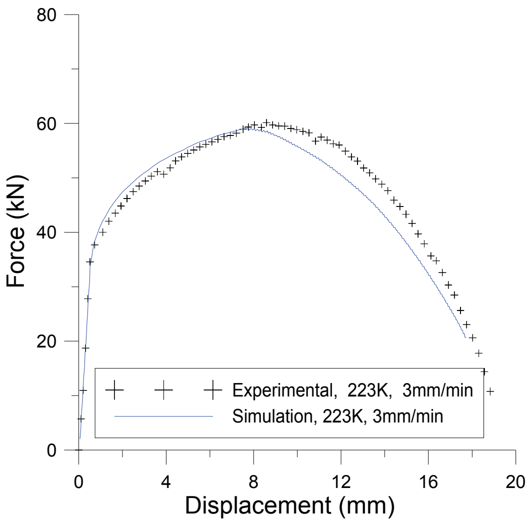

Fig. 5. *Force versus displacement curves of 223 K.*

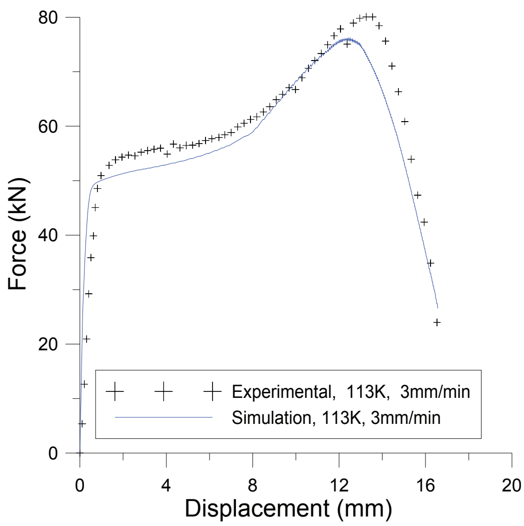

Fig. 6. *Force versus displacement curves of 113 K.*

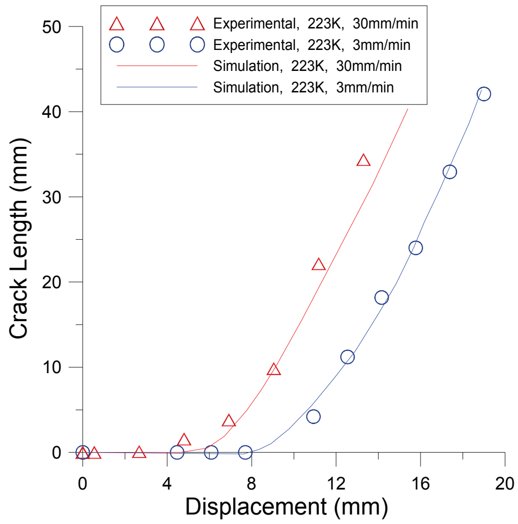

Fig. 7. *Crack length versus displacement curves of 223 K.*

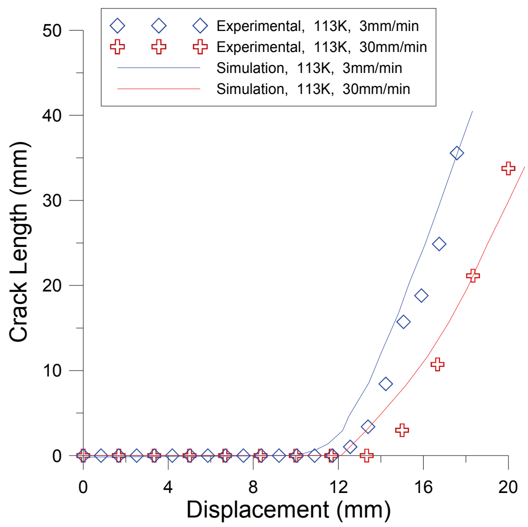

Fig. 8. *Crack length versus displacement curves of 113 K.*

5. CONCLUSION

A very useful computational algorithm based on special purpose user subroutine was developed in this study. ABAQUS user defined subroutine UMAT for stainless steel 304 was developed based on damage coupled visco-plastic constitutive model. The developed UMAT was verified using material tensile test and crack propagation test. The crack propagation characteristic of stainless steel was successfully predicted using the proposed computational approach. It was found that the complex material behavior, i.e., TRIP, can be described by the proposed model. The proposed method has been verified by comparing with stainless steel plate crack propagation test as well. It is our hope that this result can give useful information and/or guide to establish a suitable computational methodology for several industrial problems related to stainless steel applications.

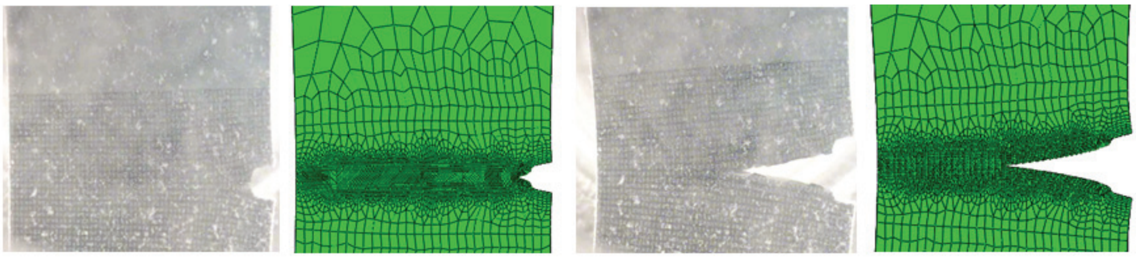

Fig. 9. Comparison between simulation results and experimental results of 113 K.

ACKNOWLEDGEMENTS

This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education, Science and Technology (20090068500) and Advanced Ship Engineering Research Center (ASERC) research program.

REFERENCES

ABAQUS User's Manual Version 6.9., 2009.

Andersson, H., 2002, An implicit formulation of the Bodner-Partom constitutive equations, *Computers & Structures*, 81, 1405-1414.

Bodner, S.R., 2002, *Unified plasticity for engineering applications*, Kluwer Academic/Plenum Publishers.

Bodner, S.R., Chan, K.S., 1986, Modeling of continuum damage for application in elastic-viscoplastic constitutive equations, *Engng. Fract. Mech*., 25, 705-712.

Hayhurst, D.R., 1972, Creep rupture under multi-axial state of stress, *J. Mech. Phys. Solids*., 20, 381-390.

Hayhurst, D. R., Leckie, F. A., 1977, Constitutive equation for creep damage. *Acta Metallurgica*, 25, 1059-1070.

McHugh, P. E., Connolly, P. J., 2003, Micromechanical modeling of ductile crack growth in the binder phase of WCCo, *Comput. Mater. Sci*, 27, 423-436.

Mishnaevsky Jr., L., 2005, *Numerical experiments in the mesomechanics of materials*, Habilitation Thesis, TU Darmstadt.

Ohio, 1976, *Source book on stainless steels*, American Society for Metals, Metals Park.

Park, W.S., Yoo, S.W., Kim, M.H., Lee, J.M., 2010, Strain rate effects on the mechanical behavior of the AISI 300 series of austenitic stainless steel under cryogenic environments, *Materials and Design*, 31, 3630-3640.

Yoo, S.W., 2010, *Development of temperature and strain-rate dependent damage model of trip steels for low temperature applications*, Master Degree Theses, Pusan National University, Pusan.

Yoshimura, H., Shimizu, T., 1982, Tensile and impact properties of 25mn-5cr-1ni austenitic steel at liquid helium temperature, *Iron Steel Institute Japan*, 22, 577-585.

Tomita, Y., Iwamoto, T., 2001, Computational prediction of deformation behavior of TRIP steels under cyclic loading, *International Journal of Mechanical Sciences*, 43, 2017-2034.

NUMERYCZNA METODA PRZEWIDYWANIA PROPAGACJI PĘKNIĘĆ W BLACHACH GRUBYCH ZE STALI NIERDZEWNEJ W OPARCIU O MECHANIKĘ PĘKANIA

Streszczenie

Celem pracy jest opracowanie numerycznej metody przewidywania propagacji pęknięć w blachach grubych ze stali nierdzewnej w temperaturach kriogenicznych. Stal nierdzewna jest jednym z najbardziej funkcjonalnych materiałów w szerokim zakresie temperatur. Charakteryzuje się mocną nieliniowością własności mechanicznych w temperaturach kriogenicznych, między innymi nieciągłym umocnieniem wywołanym przemianą fazową. Nieliniowe umocnienie wpływa znacząco na takie charakterystyki materiału jak wytrzymałość, odkształcalność i pękanie. Zjawisko to powoduje trudności w oszacowaniu możliwości konstrukcyjnych zastosowań stali nierdzewnych. W artykule przeprowadzono symulacje charakterystyk propagacji pęknięć w blachach grubych ze stali nierdzewnych w temperaturach kriogenicznych. Zastosowano w tym celu nową numeryczną technikę opartą o model konstytutywne sprzężony z pękaniem, wprowadzany jako procedura użytkownika do programu ABAQUS. Wyniki symulacji numerycznych porównano z doświadczalnymi próbami rozciągania w temperaturach kriogenicznych.

Received: September 30, 2010
Received in a revised form: November 8, 2010
Accepted: December 3, 2010