*ISSN 1392 - 1207. MECHANIKA. 2005. Nr.1(51)*

# Application of adaptive finite elements for solving elastic-plastic problem of SENB specimen

**R. Kačianauskas*, E. Stupak**, S. Stupak*****

**Vilnius Gediminas Technical University, Saulėtekio al. 11, LT-10223 Vilnius-40, Lithuania, E-mail: rkac@fm.vtu.lt*

***Vilnius Gediminas Technical University, Saulėtekio al. 11, LT-10223 Vilnius-40, Lithuania, E-mail: Eugenius.Stupak@fm.vtu.lt*

****Vilnius Gediminas Technical University, Saulėtekio al. 11, LT-10223 Vilnius-40, Lithuania, E-mail: stupakas@adm.vtu.lt*

## 1. Introduction

A major part of fracture mechanics problems is related to stress strain fields in continuum with geometric discontinuities at rest [1-5]. In spite of significant achievements this issue remains still open.

Generally, for the investigation of field problems in non-linear fracture mechanics, analytical, numerical and experimental methods may be employed. The application of analytical solutions is still limited. The most popular analytical solution was proposed by Hutchinson, Rice and Rosengren [6-7] and the later corrected by Sih [8]. This solution considers elastic-plastic behavior of non-linear materials obeying Ramberg-Osgood hardening law. Such a material is applicable for some types of steels, for example duplex stainless steels [9].

Among the numerical methods the most usually used is the finite element method (FEM). However, it was investigated and shown earlier that it is also necessary to assess the quality of the computed results, which depends on the FE mesh. The unstructured FE meshes with proposed adaptive FE meshing strategy based on the stress criterion presented in [10-12] seem to be one of the most prospective numerical tools.

The advantages of the proposed technique were tested solving 2D elastic problems of fracture mechanics and crack propagation problems. The implementation of this technique to 3D has been presented in [13]. Recently adaptivity is also combined with remeshing technique using the remeshing technique with the transfer of state variables from an old FE mesh to new one [14-16]. These approaches suffer, however, from enormous computational expences.

The aim of this paper is the application of adaptive FE analysis technique for solving elastic-plastic problem of single edge notch bend (SENB) specimen. The main concept of this investigation is to use adaptive FE analysis without transfer of variables. In the proposed approach numerical analysis of material non-linear problem is carried out from the beginning till some fixed value of loading. After the comparison of results the new FE mesh was generated using maximum stress indicator and the analysis till the same value of loading is carried out.

## 2. Problem formulation

The analysis of a SENB specimen, see Fig. 1 (thickness $B = 15.8$ mm), with a central notch is considered below.

The external loading is given by a controlled quasistatic central displacement $U$, which is a function of time $t$, $U \equiv U(t)$ .

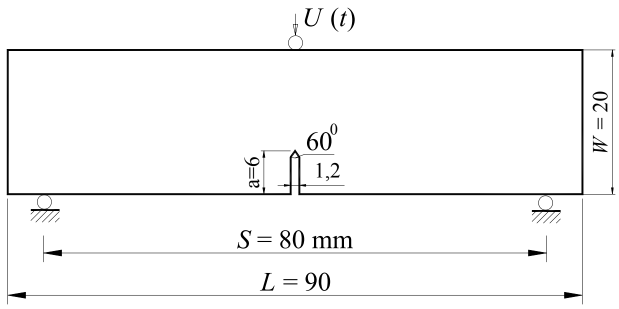



Fig. 1 Geometry and loading of a SENB specimen

The behaviour of the specimen's material is assumed to be elastic-plastic. The linear elastic part of the material diagram is characterized by the ratio $\sigma_0 / E = 0.002$ and Poisson's ratio $\nu = 0.3$ . Plastic behaviour of the material above the yielding limit $\sigma > \sigma_0 = 1160$ MPa is taken as a non-linear hardening material described by Ramberg-Osgood law

$$\frac{\varepsilon}{\varepsilon_0} = \left(\frac{\sigma}{\sigma_0}\right)^n \sigma_0 \qquad (1)$$

where $\varepsilon_0$ is yield strain, $n$ is hardening exponent.

This diagram for $n = 10$ is presented in Fig. 2.

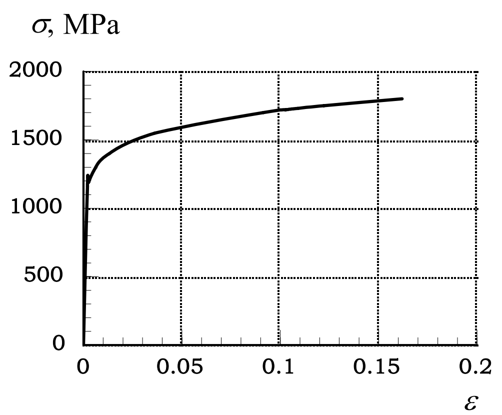



Fig. 2 Material stress-strain diagram for $n = 10$

### 3. Adaptive stress analysis

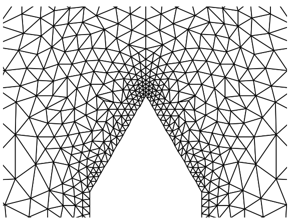
Triangle finite elements with an adaptive meshing for the simulation of plane stress problems are employed. An adaptive FE strategy based on the stress criterion is presented in [10-12]. For the adaptive meshing and computation of stress-based indicators original pre- and postprocessor software compatible with ANSYS [17] environment has been developed.

Three adaptive unstructured FE meshes were generated in each of 3 time steps: 1) at the end of lower loading, 2) in medium loading, 3) at maximum loading.

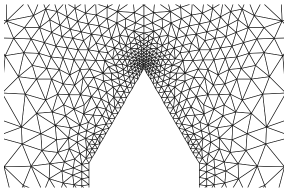
In the proposed approach, numerical analysis with the 1st FE mesh is carried out from the beginning till some fixed value of loading. At this point, the analysis of stress field at the vicinity of the crack tip is carried out and the new adaptive FE mesh is generated, using maximum stress indicator. At the next stage, the analysis till the same value of loading is carried out, using this new FE mesh and stress comparison. It allows avoiding the application of FE mesh remeshing technique with transfer of variables.

At the 1st time step ranging between 0 and 1.00 s displacement was applied with time increment equal $\Delta t_1 = 0{,}10$ s, in the 2nd time step ranging between 1.00 and 1.25 s time increment was $\Delta t_2 = 0.0025$ s and in the 3rd time step ranging between 1.25 and 1.50 s time increment was $\Delta t_3 = 0.001$ s.

At first, the adaptive stress analysis in elastic stage was performed. FE meshes of the 2nd and the 3rd step of the adaptive analysis are presented in Fig. 3, a, b.



a

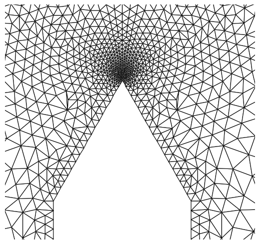


b

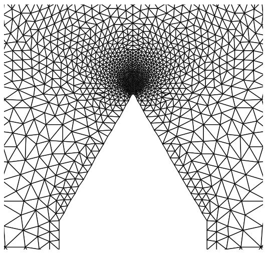
Fig. 3 Adaptive FE meshes: a –2nd (1439 nodes, 2652 FE); b – 3rd (1618 nodes, 3008 FE)

The first FE mesh of the second stage (4th FE model) is adequate to the 3rd FE mesh from the first stage, while details of the 5th and 6th adaptive FE meshes are presented in Fig. 4. The first FE mesh of the third (last) stage (7th FE model) is adequate to the 3rd FE mesh from the second stage (6th FE model) while details of the 8th and 9th adaptive FE meshes are presented in Fig. 5.

Numerical results may be validated by comparison with analytical solution. The simplest first order solution of linear problem is well-known and is given in the references [1-4]. The stress tensor components $\sigma_{ij}$ are expressed in the terms of the opening mode stress intensity factor $K_I$:

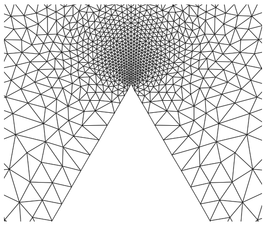


a

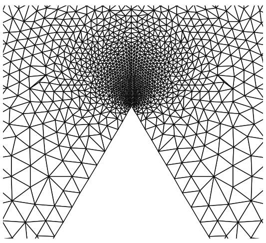


b

Fig. 4 Details of the adaptive FE meshes: a – 5th (2118 nodes, 3972 FE); b – 6th (2682 nodes, 5060 FE)



a



b

Fig. 5 Details of the adaptive FE meshes: a – 8th (3023 nodes, 5750 FE); b – 9th (3317 nodes, 6340 FE)

$$\sigma_{ij} = \frac{K_I}{\sqrt{2\pi r}} f_{\sigma ij}\left(\theta\right) + T\ \delta_{1i}\delta_{1j} \qquad (2)$$

here $r$ and $\theta$ are polar co-ordinates, $f_{\sigma ij}(\theta)$ is dimensionless function, $T$ is called "transverse stress" or "$T$ stress", $\delta$ is Kroneker symbol. For the SENB specimen $K_I$ is expressed as [1]

$$K_I = \frac{6F\sqrt{a}}{BW}\left[1.93 - 3.07\left(\frac{a}{W}\right) + 14.53\left(\frac{a}{W}\right)^2 - 25.11\left(\frac{a}{W}\right)^3 + 25.8\left(\frac{a}{W}\right)^4\right] \qquad (3)$$

where $F$ is applied load; $a$ is the length of the notch and crack; $W$ and $B$ are the width and thickness of the plate respectively.

Non-linear stress fields near the crack tip [1-4] are found on the basis of deformation theory for a finite exponent $n$ of plastic material. Hutchinson, Rice and Rosengren [6-7] solved the problem, while later Sih [8] proposed the following corrected expression. Analytical solution limited by first-order term is

$$\sigma_{ij} = \sigma_0 \left( \frac{J}{\alpha \varepsilon_0 \sigma_0 I_n r} \right)^{\frac{1}{n+1}} f_{\sigma ij}\left(\theta, n, M_p\right) \qquad (4)$$

where $\alpha$ and $n$ are constants of plastic material, dimensionless function $f_{\sigma ij}(\theta)$ of the polar angle $\theta$ depend on loading (fracture) mode, on the strain hardening exponent $n$ and plastic mixity of the modes defined by parameter $M_p$. It is proposed by Sih in the case of superposition of two modes.

## 4. Numerical results

The behaviour of SENB is characterised by opening stress coinciding with stress component $\sigma_x$ acting perpendicular to possible crack propagation direction. The normalised values $\sigma_x / \sigma_0$ are considered here for the sake of comparison. The first loading stage is supposed to be elastic. The results of this stage were compared in Fig. 6 with linear ones presented by Eq. (2).

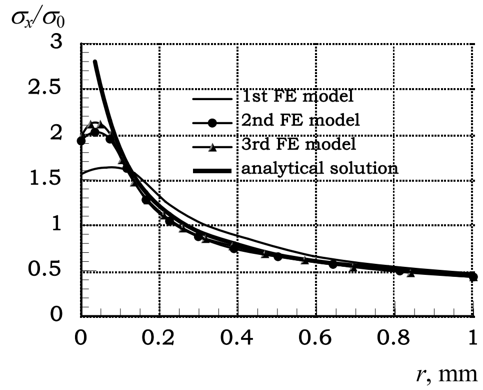


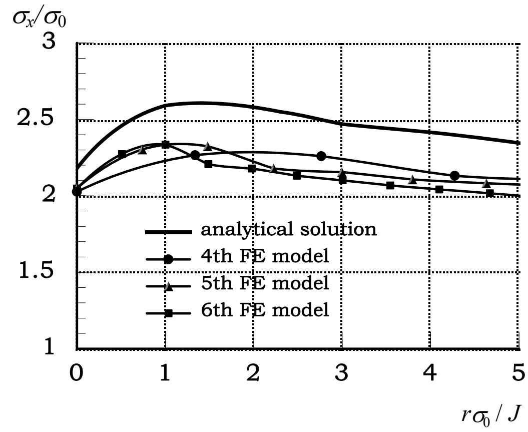

Fig. 6 Comparison of stresses for elastic stage

The curves illustrate elastic character of specimen behaviour, where curves are generally coinciding with linear solution. Results of 2nd and 3rd FE models show good convergence with minor 1 % difference. It allows concluding that such the verification test is enough. The nonlinear model allows, however, to detect existing of small plastic zone in the vicinity of notch tip with limited value maximal stress.

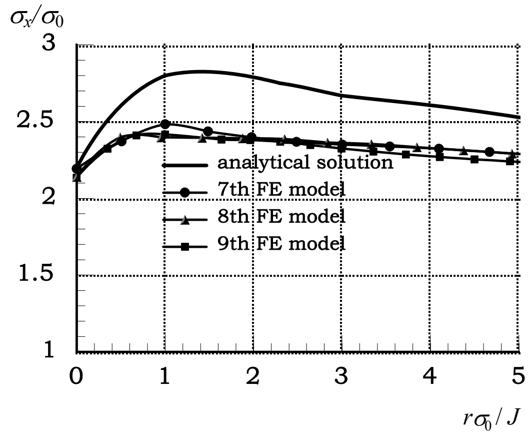
The development of plastic deformations possesses elastic-plastic behaviour of the specimen. These material nonlinear but geometrical linear models have been analysed incrementally in the first example during the second and the third stages of loading. Final solutions are reached using 110 and 360 equilibrium iterations, respectively. The results of adaptive analysis are presented in Fig. 7. The dimensionless stress values clearly illustrate the development of plastic zone in the vicinity of notch tip, which is characterised by occurring of unloading zone. The size of plastic zone is small enough in order to satisfy an assumption about small scale yielding.

Stress reach the maximum value when the relative distance $r\sigma_0 / J$ is equal 1 and decrease, when $r \to 0$. This value is equal to about double crack tip opening displacement, $\approx 2\delta$. It should be noted that the maximum of stresses becomes closer to the notch tip in the finer meshes 5th and 6th FE models in Fig. 7, a and the decreasing of stresses is quicker in some distance from the crack tip. After the comparison of stresses in 4th and 6th FE models it could be shown that the stress maximum in the 6th FE model is at 2 times closer distance than in the 4th FE model. Analysing the obtained results it can be concluded that the mesh refinement in the vicinity of crack tip provides convergent stress field.

Results of the maximum loading stage are compared in Fig. 7, b. The results show that the stress maximum is reached at the 7th model in this case, since stress curves in the 8th and 9th FE are not very smooth. Increasing of loading possesses increasing of maximal stress and reduction of unloading zone.




a




b

Fig. 7 Comparison of stresses for: a – medium loading; b – maximum loading

The second geometrical nonlinear model was solved to examine sensitivity of the problem to small geometrical changes. The comparison of stresses in the material and geometrical nonlinear models at the maximum loading is presented in Fig. 8. Results were calculated with the finest FE mesh – 9th FE model.

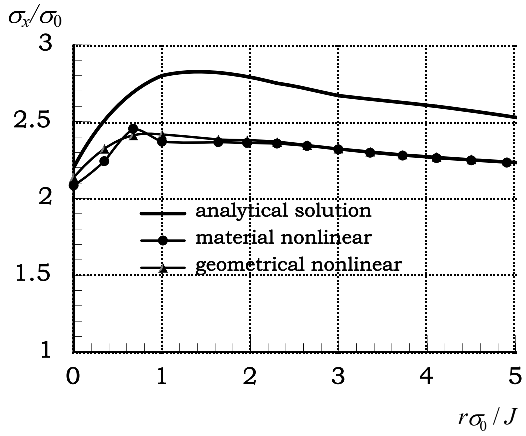
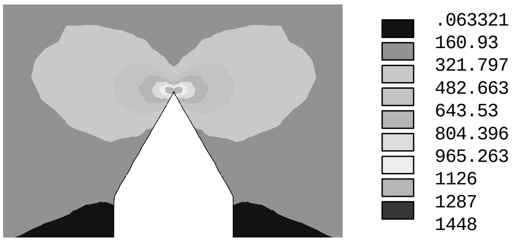
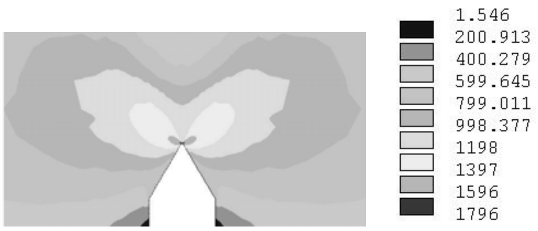



Fig. 8 Comparison of stresses

The distributions of equivalent stresses in the vicinity of crack tip are illustrated in Fig. 9.




a




b

Fig. 9 Stresses in plate for: a – elastic stage; b – maximum loading

The results show, stress and strain fields are obtained with the difference up to 5 % using the adaptive FE meshes. The investigation of geometrical nonlinearity provides that its influence in close vicinity of the crack tip is increasing stress values up to 5 %, while at relative large distance of 1.5 $r\sigma_0 / J$ its influence remains insignificant.

## 5. Discussion and conclusions

The adaptive FE technology without transfer of the state variables is proposed and adopted for the analysis of elastic-plastic problem of SENB specimen. The results of the simulation may be summarised as follows:

1. The proposed *h*-adaptive FE strategy is applicable for solving material nonlinear problems of SENB specimen. The results of the nonlinear solution show, that stress and strain fields are obtained with the difference up to 5 % using the adaptive FE meshes.

2. The investigation of geometrical nonlinearity provides that its influence in close vicinity of the crack tip is increasing the stress values by up to 5 %, while at relative large distance 1.5 $r\sigma_0 / J$ this influence remains insignificant.

## References


1. **Панасюк, В.В., Андрейкив, А.Е., Партон, В.С.** Основы механики разрушения материалов. Механика разрушения и прочность материалов: Справ. пособие / Под ред. Панасюка В.В.-Киев: Наукова думка, 1988, т.1.-488с.
2. **Anderson, T.L.** Fracture Mechanics. Fundamentals and Applications.-Boston: CRC Press, Inc., 1991.-793p.
3. **Miannay, D.P.** Fracture mechanics. -New-York: Springer, 1998.-337p.
4. **Kienzler, R., Herrmann, G.** Mechanics in material space with applications to defect and fracture mechanics.-Berlin, Heidelberg, New-York: Springer, 2000.-289p.
5. **Seweryn, A., Poskrobko, S., Mróz, Z.** Brittle fracture in plane elements with sharp notches under mixed-mode loading.-J. of Eng. Mechanics, 1997, v.123, No.6, p.535-543.
6. **Hutchinson, J.W.** Plastic stress and strains fields at crack tip.-J. Mech. Phys. of Solids, 1968, v.16(4), p.337-347.
7. **Rice, J., Rosengren, G.F.** Plain strain deformation near a crack tip in a power law hardening material.-J. Mech. Phys. of Solids, 1968, v.16 (1), p.1-12.
8. **O'Dowd, N.P., Shih, C.F.** Family of crack-tip fields characterized by a triaxiality parameter. Part I: Structures of fields.-J. Mech. Phys. of Solids, 1991, v.39, p.989-1015.
9. **Marrow, T.J., Humphreys, A.O., Strangwood, M.** The crack initiation toughness for brittle fracture of super duplex stainless steel.-Fatigue Fract. Eng. Mater. Struct., 1997, v.20, No.7, p.1005-1014.
10. **Stupak, E., Kačianauskas, R.** Adaptyviųjų baigtinių elementų tinklų taikymas įtempimų būviui nustatyti ties koncentratoriais. -Mechanika. -Kaunas: Technologija, 2000, No4(24), p.31-36.
11. **Kačianauskas, R., Stupak, E., Kienzler, R.** Macroscopic finite element simulation of brittle-to-ductile failure of SENB specimen.-Mechanika.-Kaunas: Technologija, 2002, No1(33), p.11-16.
12. **Stupak, E., Kačianauskas, R.** Adaptive FE analysis for crack propagation in SENB specimen.-Proc. of 1st CEACM Conf. on Computational Mechanics, 15th Int. Conf. "CMM-2003 –Computer Methods in Mechanics", June 3-6, 2003, Gliwice/Wisla, Poland, (Eds. T. Burczyński, P. Fedeliński, E. Majchrzak). -Silesian Technical University, Gliwice, Poland, 2003, p.[1-7].
13. **Kačianauskas, R., Žarnovskij, V., Stupak, E.** Three-dimensional correction of two-dimensional fracture criteria using a constraint factor. // ICTAM04 Abstracts Book and CD-ROM Proc., Warsaw, Poland, 15-21 August, 2004, (Eds. W. Gutkowski, T.A. Kowalewski). IPPT PAN, Warszawa, Poland, 2004, p.[1-2].

14. **Tradegard, A.**, **Nilsson, F.**, **Ostlund, S.** FEM-remeshing technique applied to crack growth problems. -Computer Methods in Applied Mechanics and Eng., 1998, v.160, p.115-131.
15. **Bouchard, P.O., Bay, F., Chastel, Y., Tovena, I.** Crack propagation modelling using an advanced remeshing technique.-Computer Methods in Applied Mechanics and Eng., 2000, v.189, p.723-742.
16. **Bouchard, P.O., Bay, F., Chastel, Y.** Numerical modelling of crack propagation: automatic remeshing and comparison of different criteria. -Computer Methods in Applied Mechanics and Eng., 2003, v.192. p.3887-3908.
17. ANSYS Release 8 Documentation.-SAS IP, Inc., 2003.

R. Kačianauskas, E. Stupak, S. Stupak

PRISITAIKANČIŲJŲ BAIGTINIŲ ELEMENTŲ TAIKYMAS LENKIAMO BANDINIO SU VIENPUSE ĮPJOVA TAMPRIAI PLASTINIAM UŽDAVINIUI SPRĘSTI

R e z i u m ė

Naudojant prisitaikančiųjų baigtinių elementų techniką be būvio kintamųjų perkėlimo, sprendžiamas lenkiamo bandinio su vienpuse įpjova tampriai plastinis uždavinys.

Fiziškai netiesinis uždavinys spręstas prieaugiais nuo apkrovimo pradžios iki tam tikros fiksuotos apkrovos, kuriai esant generuojamas naujas prisitaikančiųjų BE tinklas.

Įtempimai, gauti tankiausiuose BE modeliuose sprendžiant fiziškai ir geometriškai netiesinius uždavinius, lyginami tarpusavyje bei su literatūroje pateikiamais sprendiniais.

Modeliavimui atlikti naudota BE programa ANSYS.

R. Kačianauskas, E. Stupak, S. Stupak

APPLICATION OF ADAPTIVE FINITE ELEMENTS FOR SOLVING ELASTIC-PLASTIC PROBLEM OF SENB SPECIMEN

S u m m a r y

The proposed adaptive finite element analysis technique without transfer of variables is used for solving elastic-plastic problem of SENB specimen.

The material nonlinear problem has been solved in iterative manner from the beginning till the fixed value of loading where new adaptive FE mesh was generated.

Comparison of stresses, obtained solving material and geometrical nonlinear problems with the finest FE models, has been done with solutions presented in the literature.

The modelling was carried out using FE code ANSYS.

Р. Качянаускас, Э. Ступак, С. Ступак

ПРИМЕНЕНИЕ АДАПТИВНЫХ КОНЕЧНЫХ ЭЛЕМЕНТОВ ДЛЯ РЕШЕНИЯ УПРУГО-ПЛАСТИЧЕСКОЙ ЗАДАЧИ В ИЗГИБАЕМОМ ОБРАЗЦЕ С ОДНОСТОРОННИМ НАДРЕЗОМ

Р е з ю м е

В работе решается упруго-пластическая задача для изгибаемого образца с односторонним надрезом, используя технику адаптивных конечных элементов без переноса переменных напряженного состояния.

Физически нелинейная задача решалась, используя итерации от начала нагрузки до фиксированной нагрузки, где была генерирована новая сеть КЭ.

Напряжения, полученные решая физически нелинейную задачу и задачу больших деформаций используя самые мелкие сети КЭ, сравнивались между собой и с результатами решений, других авторов.

Для моделирования была использована программа КЭ ANSYS.

Received September 15, 2004