Mr Miloš Dobrojević, prof. dr Aleksandar Sedmak, mr Emhamed Argob, mr Olivera Popović

ANALIZA UTICAJA GEOMETRIJE I HETEROGENOSTI ZAVARENOG SPOJA NA PONAŠANJE ŠARPI EPRUVETE

THE ANALYSIS OF GEOMETRY AND WELDED JOINT HETEROGENEITY EFFECT ON CHARPY SPECIMEN BEHAVIOUR

Originalni naučni rad / Original scientific paper
UDK /UDC: 620.178.7:621.791.052
Rad primljen / Paper received: 01.12.2003.

Adresa autora / Author's address:
Mašinski fakultet Univerziteta u Beogradu, Srbija i Crna Gora

Ključne reči

- Heterogenost zavarenog spoja
- Mečing efekt
- Metoda konacnih elemenata
- Naponska analiza
- Standardno Šarpi ispitivanje
- Šarpi epruveta malih dimenzija

Keywords

- Heterogeneity of welded joint
- Matching effect
- Finite Elements Method
- Stress analyses
- Standardized Charpy testing
- Subsized Charpy specimen

IZVOD

U radu je analiziran uticaj geometrije (oblik zareza, dimenzije epruvete) i heterogenosti materijala (mismečing zavarenog spoja) na ponašanje Šarpi epruvete. Pimenjena je dvo- i trodimenzionalna naponska analiza metodom konačnih elemenata (MKE) za rešavanje ovog problema, čime je dobijen raspored naponskih polja i plastične deformacije u epruveti i prikazano njeno ponašanje.

ABSTRACT

The effect of geometry (notch shape, specimen size) and materials heterogeneity (mismatching in welded joint) on the Charpy specimen behaviour is analysed in the paper. Two- and three-dimensional stress analyses by finite element method (FEM) are applied in solving this problem, thus providing stress-fields and plastic deformation distribution in specimen, and presenting its behaviour.

UVOD

Ispitivanje po Šarpiju je uvedeno za ispitivanje malih zarezanih uzoraka savojnim udarnim opterećenjem. Imajući u vidu da prirodu loma čelika (duktilan ili krt) definišu tri parametra (brzina deformacije, raspodela napona u epruveti i temperatura ispitivanja), jasno je da mala promena parametara ispitivanja može dovesti do velikih promena u dobijenim osobinama materijala. Zato je uveden standardni postupak ispitivanja, u kome je usvojena veličina epruvete 10x10x55 mm sa "U" ili "V" zarezom (sl. 1), koja se postavlja na oslonac (sl. 2), i izlaže udarnoj energiji do 300 J. Ovi uslovi ne mogu biti uvek ispunjeni u praksi i dopuštena je njihova promena. Standard JUS EN 10045-1 preporučuje da se uz ispitivanje žilavosti (i određivanje energije udara) Šarpi klatnom obave i dopunska ispitivanja poput merenja kontrakcije ili fraktografske analize, a sve zarad dobijanja što više informacija o sklonosti ispitivanog materijala ka krtom lomu.

Instrumentirano Šarpi klatno. Dijagram sila-vreme

Rezultati dobijeni klasičnim Šarpi ispitivanjem su danas nedovoljni. Naime, ispitivani uzorak je suviše mali da bi dao podatke o karakteru loma ili odražavao ponašanje materijala u eksploataciji, ako karakteristike zavise od debljine. Takođe, nemoguće je primeniti realne uslove opterećenja, niti utvrditi geometrijski uticaj debljine (epruvete standardnih dimenzija), a veličina udarne žilavosti zavisi od vrste ispitivanog materijala. Problem je rešen ugradnjom osciloskopa i kompjutera na Šarpi klatno (sl. 3) tako da se opterećenje meri mernim mostom, a ugib potenciometrom /1/, /2/.

INTRODUCTION

The Charpy's test is designed for testing of small notched samples exposed to flexion by impact loading. Considering that the nature of steel fracture (ductile or brittle) is defined by three parameters (strain rate, stress distribution in the specimen and testing temperature), it is clear that even a small change in testing parameters may result in great changes in obtained material properties. Therefore, standard test method has been introduced, adopting specimen size of 10x10x55 mm with "U" or "V" notch (Fig. 1), which is positioned on the support (Fig. 2), and exposed to impact energy up to 300 J. These conditions can not be fulfilled always in practice, and their variation is allowed. Standard JUS EN 10045-1 recommends, when testing impact toughness (and defining impact energy) by Charpy's pendulum, to performed additional tests, i.e. contraction measurement or fractographic analysis, in order to obtain more information on tested material susceptibility to brittle fracture.

Instrumented Charpy's pendulum. Force-time diagram

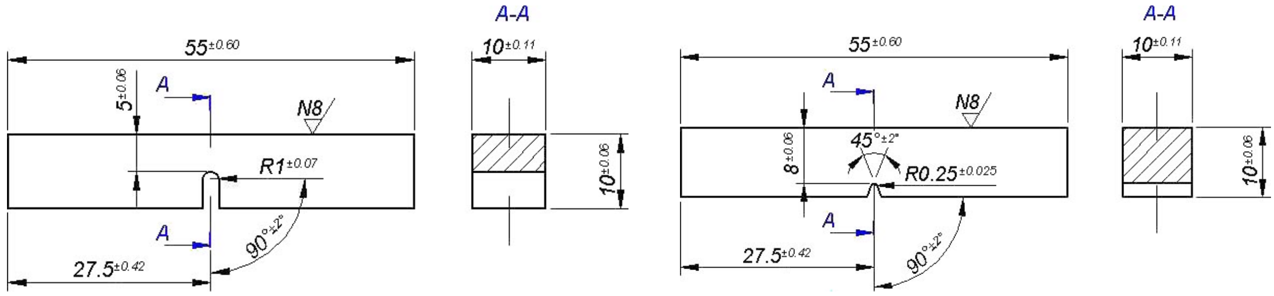
Results obtained by classical Charpy's test nowadays are insufficient. Namely, the tested specimen is too small to produce the data for fracture characterisation or to describe behaviour of material in service, if the characteristics depend on thickness. Also it is neither possible to apply the real loading condition, nor to establish the geometrical effect of thickness (standard specimen size), and the impact toughness value depends on the type of tested material. The problem is solved by involving oscilloscope and computer in Charpy's pendulum (Fig. 3) so that load is measured by measuring bridge and deflection by potenciometer /1/, /2/.

To je omogućilo razjašnjenje mehanizma loma epruvete, koje je osnova za precizniju karakterizaciju udarnih svojstava materijala. Analizom dijagrama sila-vreme moguće je odrediti vrednosti energije stvaranja prsline, vremena i brzine rasta prsline, maksimalne sile međudejstva klatna i epruvete, dinamičke sile na granici tečenja, sile na početku nestabilnog loma, sila zaustavljanja nestabilnog loma i maksimalnog ugiba epruvete.

Ako je potrebno, na osnovu dijagrama sila-vreme, mogu se dobiti i drugi različiti dijagrami, kao ugib-vreme, sila-ugib, energija-vreme, brzina klatna-vreme (sl. 4).

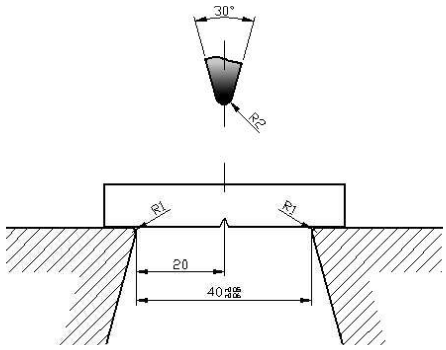
This has enabled the explanation of fracture mechanism, which is the base for more precise characterization of the materials impact properties. By analysing the force-time diagram it is possible to define the value of crack initiation energy, time and crack growth rate, maximum pendulum-specimen interaction force, dynamic force at the yield stress, force at the beginning of unstable fracture, force of unstable fracture arrest and specimen maximum deflection.

If necessary, based on force-time diagram, various diagrams can be obtained as well, such as deflection-time, force-deflection, energy-time, pendulum velocity-time (Fig. 4).



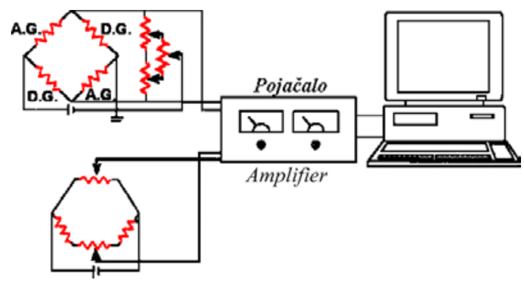
Slika 1. Oblik i dimenzije standardne epruvete (JUS EN 10045-1) sa "U" i " V" zarezom

Figure 1. Shape and dimensions of standard specimen (JUS EN 10045-1) with "U" and " V" notch



Slika 2. Postavljanje epruvete na oslonac (JUS EN 10045-1)

Slika 2. Positioning of specimen on support (JUS EN 10045-1)



Slika 3. Kompjuterizovano instrumentirano udarno ispitivanje (CAI)

Figure 3 -Computer aided instrumented impact testing (CAI)

Kompjuterizovano instrumentirano udarno ispitivanje

Ugradnjom kompjutera (PC) i elektronskih komponenti na instrumentirano klatno omogućeno je kompjuterizovano instrumentirano udarno ispitivanje (CAI metoda). Ova metoda olakšava obradu parametara mehanike loma (poput faktora intenziteta napona K_I i J-integrala), kao i analizu apsorbovane energije. Opterećenje se meri mernom trakom na tegu, a ugib potenciometrom na osovini klatna ili magnetnim senzorom na okviru mašine. Podaci o opterećenju i ugibu prolaze kroz visokofrekventno pojačalo i digitalni osciloskop do PC.

Energija udara

U klasičnom ispitivanju Šarpi, energija se definiše kao integralna veličina:

$$E = v_0 \int_0^t F(t) \cdot dt \quad [J]$$

gde je E ukupna energija udara, v_0 početna brzina klatna, $F(t)$ trenutno opterećenje, a t vreme.

Computer aided instrumented impact testing

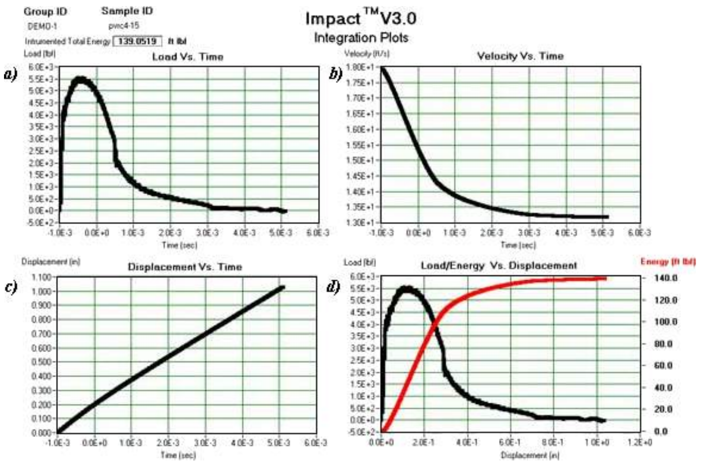
By involving computer (PC) and electronic components on the instrumented pendulum, computer aided instrumented impact testing (CAI method) is enabled. This method makes the processing of the fracture mechanics parameters (e.g. stress intensity factor K_I and J-integral), and the analysis of energy absorbed easier. The load is measured by strain gauges on the but and the deflection by potentiometer on the pendulum axle or by the magnetic sensor on the machine frame. The data on load and deflection pass through frequency amplifier and digital oscilloscope to PC.

Impact energy

In classical Charpy testing, impact energy is defined as an integral value:

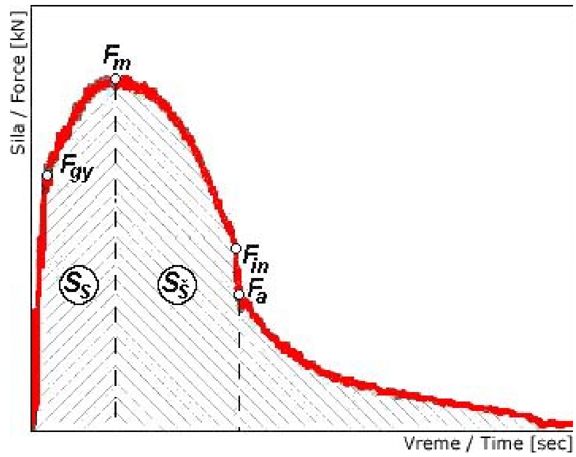
$$E = v_0 \int_0^t F(t) \cdot dt \quad [J]$$

where E represents total impact energy, v_0 initial pendulum velocity, $F(t)$ current load, and t time.



Slika 4. Tokom ispitivanja zapisani dijagrami: a) sila-vreme; b) brzina-vreme; c) ugib-vreme; d) sila-ugib i energija-ugib

Figure 4. During testing recorded diagrams: a) force-time; b) velocity-time; c) deflection-time; d) force-deflection and energy-deflection



Slika 5. Uvećan dijagram sila-vreme

Figure 5. Magnified diagram force-time

Kod CAI metode praćenje procesa udara je moguće preko već pomenutih dijagrama (sl. 4 i 5), koji pokazuju promenu sile otpora materijalu tokom procesa loma i udeo udarne energije utrošene na stvaranje prsline (E_i) i na njeno širenje (E_p) (najlakše pomoću epruvete sa V zarezom i zamornom prslinom). Poznata ukupna energija E (udarna žilavost) ispitivanog materijala može da se podeli na udeo za stvaranje (E_s) i širenje (E_p) prsline:

$$E = E_i + E_p$$

Energija stvaranja prsline (E_i) se ne menja mnogo sa promenom temperature ispitivanja (zbog smanjenja energije za plastičnu deformaciju u zoni vrha zareza), dok energija širenja prsline (E_p) zavisi od temperature ispitivanja. Granični slučaj je za E_p = 0, kada se ukupna energija troši na stvaranje prsline, što dovodi do brzog loma.

U skladu sa tim, pri projektovanju kritičnih delova konstrukcije korisna su sledeća dva pravila:
- energija širenja prsline u kritičnoj tački mora biti veća od energije stvaranja prsline na datoj temperaturi;
- od dva materijala iste ukupne energije udara, bolji je materijal sa manjom energijom stvaranja prsline.

Uticajni faktori na energiju udara

Na energiju udara utiču brojni faktori. Najvažniji su metalurški faktori (heterogenost, mikrostruktura, hemijski sastav, veličina zrna) i geometrija.

In CAI method, monitoring of the impact process is enabled thanks to above mentioned diagrams (Fig. 4 and 5), showing the change of material resistance force during fracture process and the fraction of impact energy spent for crack initiation (E_i) and its propagation (E_p) (most easily with precracked V notch specimen). Known total energy E (impact toughness) of the tested material can be divided into crack initiation (E_i) and crack propagation (E_p) fractions:

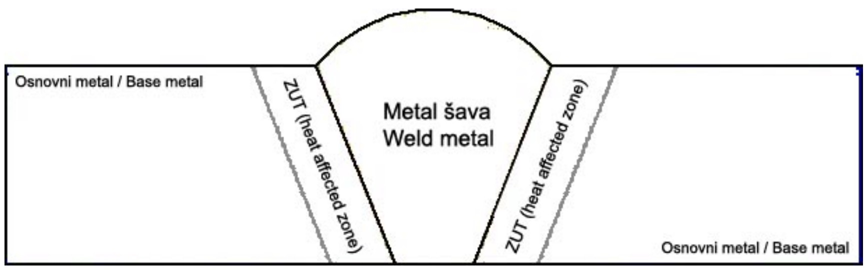
$$E = E_i + E_p$$

Crack initiation energy (E_i) doesn't vary significantly with the testing temperature change (due to decrease of energy for plastic deformation in the notch tip zone), while crack propagation energy (E_p) depends on testing temperature. Ultimate case is for E_p = 0, when the total energy is spent for crack initiation, resulting in fast failure.

Accordingly, two following two rules can be useful in design of critical structural components:
- crack propagation energy in critical point must be greater than crack initiation energy at the given temperature;
- between two materials with the same total impact energy, the material with smaller crack initiation energy is better:

The impact energy influencing factors

Numerous factors affect the impact energy. The most important are metallurgical factors (heterogeneity, microstructure, chemical composition, grain size) and geometry.

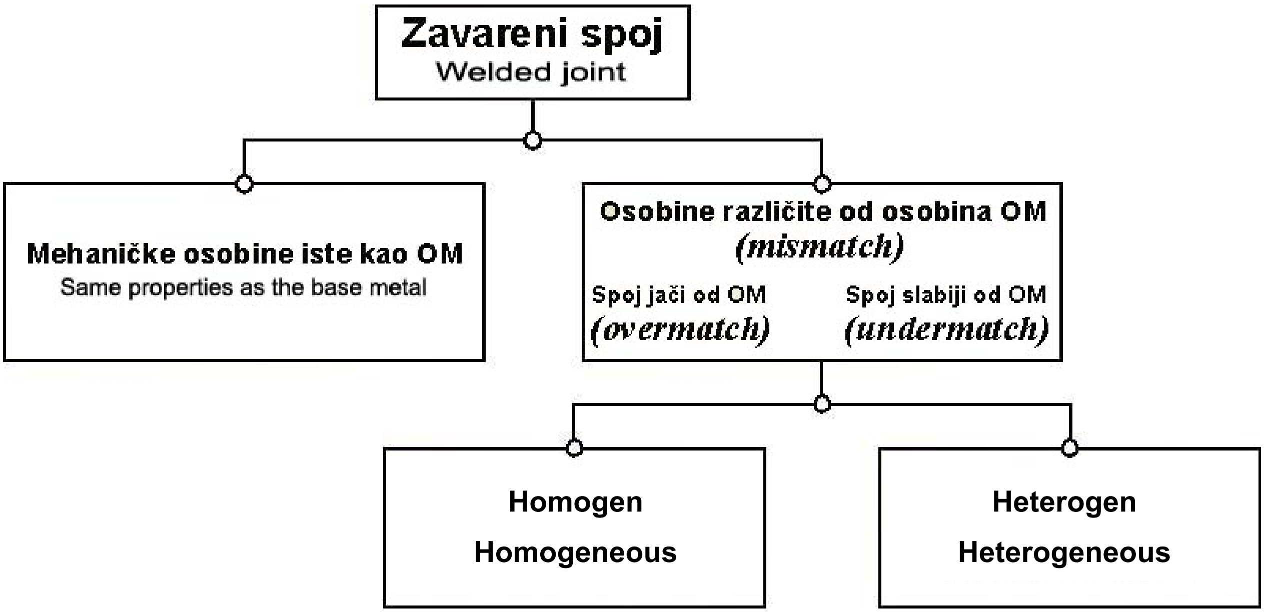


Slika6. Struktura zavarenog spoja

Figure 6. Structure of the welded joint

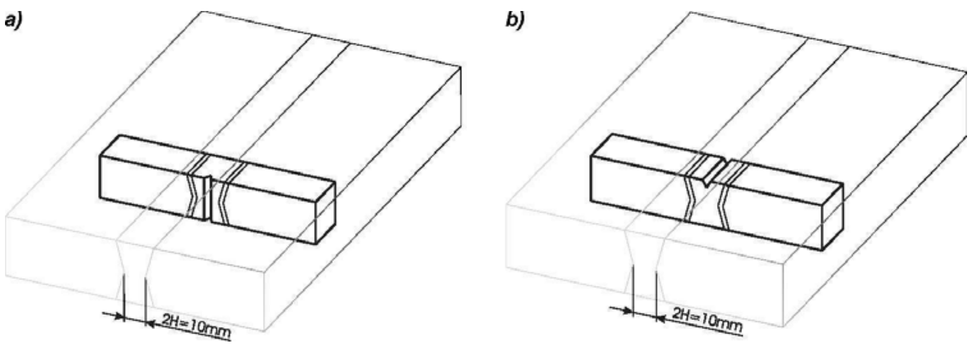
Uticaj heterogenosti materijala. Ovde razmatrana heterogenost se odnosi na strukturu zavarenog spoja, koji se sastoji od osnovnog metala (OM), metala šava (MŠ) i zone uticaja toplote (ZUT), (sl. 6). Svaki od ovih konstituenata ima različitu mikrostrukturu i različite mehaničke osobine. Mismečing (razlika u čvrstoći OM i MŠ) utiče na čvrstoću zavarenog spoja, ali ne i na njegovu udarnu žilavost /3/, /4/. Konstrukcijski oblik zavarenog spoja mora takođe da se uzme u obzir, jer su u tom pogledu moguće različite kombinacije (sl. 7 i 8) /5/.

Material heterogeneity effect. Here considered heterogeneity is related to the welded joint structure, consisting of the base metal (BM), the weld metal (WM) and the heat-affected-zone (HAZ) (Fig. 6). Each of these constituents has different microstructures and different mechanical properties. Mismatching (difference in the strength of BM and WM) affects the strength of welded joint, but not the impact toughness /3/, /4/. The design of the welded joint is also taken into consideration, for possible various combinations (Fig.7 and 8) /5/.



Slika 7. Podela zavarenih spojeva prema mikrostrukturi i mehaničkim osobinama u odnosu na OM

Figure 7. Classification of welded joints according to the microstructure and mechanical properties relating to BM



Slika 8. Uzimanje Šarpi uzorka iz zavarene ploče: a) Zarez prolazi kroz sve mikrostrukture zavarenog spoja (po dubini) i b) Zarez "pokriva" samo jednu zonu spoja (po površini)

Figure 8. Sampling from welded joint Charpy's specimen: a) The notch passes through all microstructures of the welded joint (in depth) and b) The notch "covers" only one zone of the joint (on the surface)

Uticaj geometrije. Dimenzije epruvete i oblik zareza su standardizovani, ali je dozvoljena i njihova promena. Uloga zareza je koncentracija napona na malu zapreminu epruvete. Oblik i dimenzije zareza bitno utiču na veličinu udarne žilavosti, koja je manja kod epruvete sa V zarezom od one sa U zarezom /6/. Energija udara se smanjuje sa povećanjem dubine zareza (pri istim dimenzijama epruvete), a odnos energija stvaranja i širenja prsline se menja sa promenom radijusa zareza pri konstantnoj temperaturi.

Influence of geometry. Specimen dimensions and notch geometry are standardised, but their variation is allowed. The role of the notch is stress concentration in the small specimen volume. The notch shape and dimensions greatly affect impact toughness, which is smaller for V notch specimens than for those with U notch /6/. Impact energy decreases with notch depth increase (for the same specimen size), and the ratio of initiation and propagation energies changes with notch radius variation at constant temperature.

NUMERIČKA SIMULACIJA

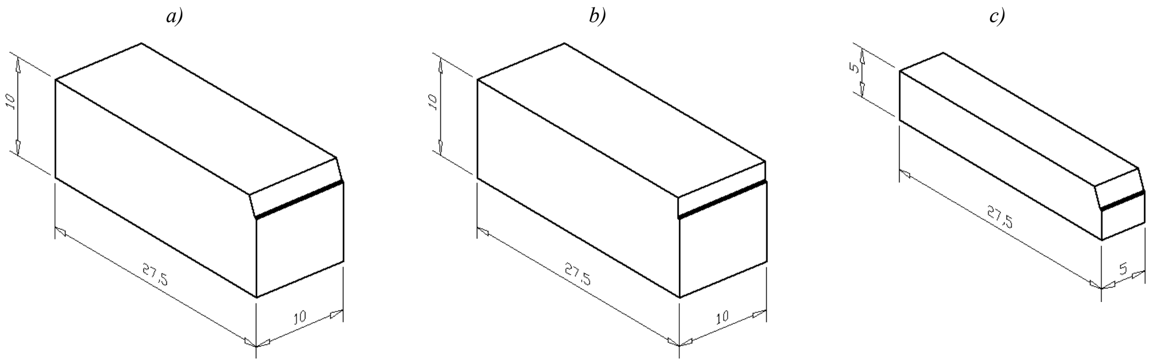
Geometrija epruvete i zareza

Da bi se stekao uvid u uticaj geometrije epruvete i zareza, za ispitivanje su korišćene epruvete 10x10x55 mm sa U nestandardnim i V standardnim zarezom, kao i epruvete malih dimenzija 5x5x55mm sa istim V standardnim zarezom. Dimenzije korišćenih epruveta su date na sl. 9, gde je radi jednostavnosti prikazana samo polovina epruvete. Geometrija zareza je data na sl. 10.

NUMERICAL SIMULATION

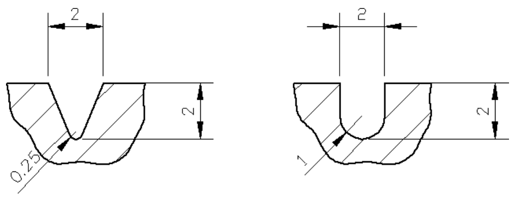
The specimen and notch geometry

To get the insight of specimen and notch geometry effect, the specimens 10x10x55 mm with nonstandard U and standard V notches, and the subsized specimens of 5x5x55 mm with the same standard V notch were tested. Dimensions of the used specimens are shown in Fig. 9, where only a half of specimen is shown for simplicity is shown. Notch geometry is given in Fig. 10.



Slika 9. Polovina epruvete a)10x10x55 mm V zarez, b) 10x10x55mm U zarez, c) 5x5x55mm V zarez

Figure 9. A half of the specimen a) 10x10x55 mm V notch, b) 10x10x55 mm U notch, c) 5x5x55 mm V notch



Slika 10. a) V zarez, b) U zarez

Figure 10. a) V notch, b) U notch

Heterogenost materijala

Uticaj heterogenosti materijala je već opisan ranije. Dakle, radi se o zavarenom spoju, gde su ispitane sledeće kombinacije:

1. Homogen zavareni spoj. Čvrstoće osnovnog i dodatnog metala su istog nivoa.
2. Homogen zavareni spoj veće čvrstoće metala šava. Dodatni materijal je veće čvrstoće od osnovnog metala.
3. Homogen zavareni spoj manje čvrstoće metala šava. Dodatni materijal je manje čvrstoće od osnovnog metala.
4. Heterogen zavareni spoj. Spoj izveden sa dva različita dodatna materijala (veće i manje čvrstoće od osnovnog metala), a zarez prolazi kroz njih.
5. Heterogen zavareni spoj veće čvrstoće metala šava. Zarez se nalazi u materijalu veće čvrstoće.
6. Heterogen zavareni spoj manje čvrstoće metala šava. Zarez se nalazi u materijalu manje čvrstoće.

Pri analizi su korišćena tri materijala čije su mehaničke osobine date u tab. 1, a dijagrami zatezanja na sl. 11. Niomol 490 je sitnozrni niskolegirani čelik visoke čvrstoće koji se koristi za izradu čeličnih konstrukcija. Dodatni materijali Fitub 75 i VAC 60, proizvodnje SZ Elektrode Jesenice, Slovenija, daju zavarene spojeve i metal šava veće i manje čvrstoće od osnovnog metala, respektivno.

Material heterogeneity

The effect of material heterogeneity is already described. Thus, it is a welded joint, where the following combinations have been tested:

1. Homogenous welded joint. The strength of basic and filler metal is of the same level.
2. Homogenous overmatched welded joint. Filler material is of higher strength than base metal
3. Homogenous undermatched welded joint. Filler material is of lower strength than base metal.
4. Heterogeneous welded joint. The joint performed with two different filler materials (of higher and lower strengths than base metal), with a notch passing through them.
5. Heterogeneous overmatched welded joint. The notch is located in the overmatched material.
6. Heterogeneous undermatched welded joint. The notch is located in the undermatched material.

Three different materials have been used in analysis, of mechanical properties given in Table 1 and tensile test records given in Fig. 11. Niomol 490 is a fine grained, low-alloyed, high strength steel for steel structures. With filler materials Fitub 75 and VAC 60, produced by SZ Elektrode Jesenice, Slovenia, overmatched and undermatched welded joints, respectively, can be obtained.

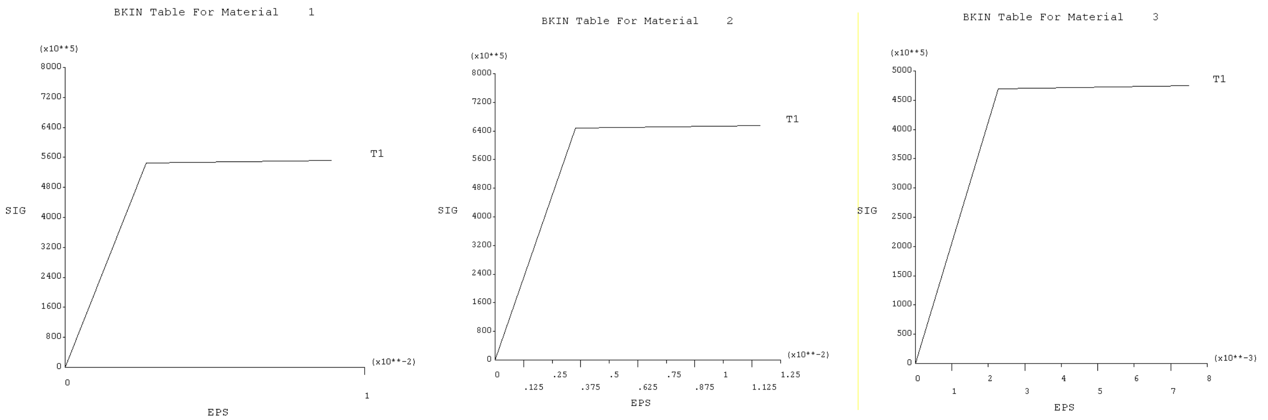
Tabela 1. Materijali korišćeni za simulaciju heterogenosti zavarenog spoja

Table 1. Materials used for simulation of welded joint heterogeneity

Materijal - Material		*E*, GPa	$R_{p0.2.}$ MPa	R_m, MPa	*H'*, MPa	$M = R_{p02M}/R_{p0.2OM}$
Osnovni metal –	NIOMOL 490	202	545	648	1030	
Overmeč – Overmatch	FITUB 75	184	648	744	960	1.19
Andermeč - Undermatch	VAC 60	206	469	590	1210	0.86

* *E* – modul elastičnosti; $R_{p0.2}$ – napon tečenja; R_m – zatezna čvarstoća; $H'=(R_m-)/0.1$ – koeficijent ojačavanja materijala; *M* – mečing faktor.

* *E* – elasticity modulus; $R_{p0.2}$ – yield stress; R_m – tensile strength; $H'=(R_m-)/0.1$ – material work hardening coefficient; *M* – matching factor.



NIOMOL 490 *FITUB 75* *VAC 60*

Slika 11. Dijagram napon-deformacija (σ-ε): a) osnovnog metala b) overmeč spoj i c) undermeč spoj

Figure 11. Stress-strain diagram (σ-ε): a) base metal, b) overmatched joint and c) undermatched joint

Modeliranje

Osnovna ideja je da se razvije model koji verno prikazuje reakciju tela na opterećenje. Pored 3D, urađena je i 2D analiza, jer se Šarpi ispitivanje može, uz određenu aproksimaciju, da predstavi kao dvodimenzionalni problem.

Žičani modeli epruvete (2D, sl. 12a i iz njega izvučen 3D, sl. 12b) su razvijeni u programima AutoCad / Solid Works, dok su generisanje mreže konačnih elemenata (sl. 12c,d) i proračun pomeranja, deformacije i napona izvedeni u ANSYS programu. Modelirana je samo jedna polovina uzorka, dok je za drugu polovina izvedena simuliracija uz odgovarajuće granične uslove. Zadatak je obavljen na dve konfiguracije:

- PC Pentium IV 2,4GHz / 256MB RAM, gde je prosečno vreme rešavanja ravnotežnih jednačina bilo 45 do 90 min za 3D i manje od 1 min za 2D model, zavisno od broja konačnih elemenata i broja iteracija.
- PC Celeron 366MHz / 512MB RAM, 210 do 480 min za 3D i 1 do 1,5 minut za 2D model.

Izneti podaci pokazuju značaj korišćenja brzog kompjutera pri radu sa konačnim elementima.

Mreža konačnih elemenata

Kako brzina proračuna direktno zavisi od broja konačnih elemenata, optimizacijom je na najopterećenijem mestu (zarez) mreža usitnjena (veći broj manjih elemenata); u zoni zavarenog spoja i na mestima oslonjanja epruvete elementi su nešto krupniji, dok je ostatak mreže ispunjen manjim brojem većih elemenata /7/, /8/.

Modelling

The basic idea is to develop a model which reliably represents reaction of the body to loading. In addition to 3D, 2D analysis has also been carried out, since Charpy's test, with some approximation, may be presented as 2D problem.

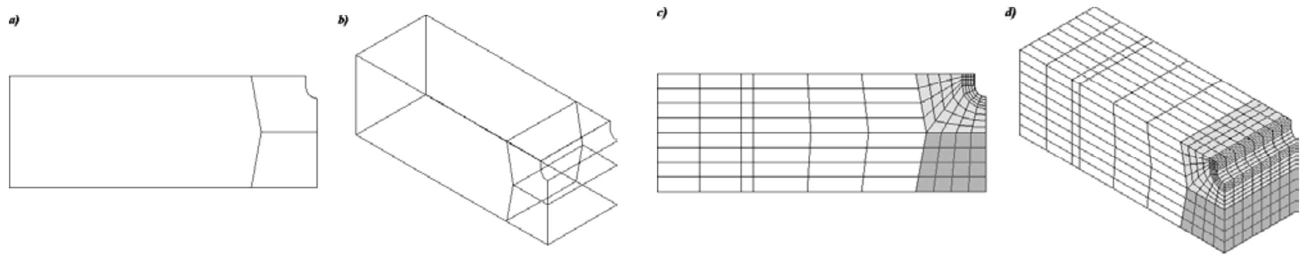
The wire frame models of specimen (2D, Fig. 12a and from it extruded 3D, Fig. 12b) have been developed using AutoCad / Solid Works softwares, while generation of finite elements (Fig. 12c,d) and calculation of displacement, strain and stress have been performed using ANSYS program. Only one half of the specimen has been modelled, the other one being simulated by appropriate boundary conditions. The task has been accomplished on two configurations:

- PC Pentium IV 2.4GHz / 256MB RAM, where average time for solving equilibriun equations was 45 to90 min for 3D and less than 1 min for 2D model, depending on the finite elements number and number of iteration.
- PC Celeron 366MHz / 512MB RAM, 210 to 480 min for 3D and 1 to 1.5 minute for 2D model.

Presented data show the imortance of using fast computer when dealing with finite elements.

Finite Elements Mesh

Since the processing speed is dependent directly on finite elements number, by optimisation in the most loaded point (notch) the mesh is refined (great number of small elements) in the zone of welded joint and specimen supporting points the elements are somewhat larger, while the rest of the mesh is filled with a small number of coarser elements /7/, /8/.



Slika 12. Žičani modeli epruvete 10x10x55mm sa U zarezom: a) 2D i b) 3D. Model sa konačnim elementima c) 2D i d) 3D

Figure 12. Wire frame models of specimen 10x10x55mm with U notch: a) 2D and b) 3D. Models with finite elements c) 2D and d) 3D

Pri izradi mreže korišćeni su 2D elementi sa 8 čvorova i 3D cigla elementi sa 20 čvorova. Svaki čvor ima tri stepena slobode (translacija po x, y i z pravcu). Element ovog tipa može zauzeti bilo koji položaj u ravni / prostoru, a omogućava simuliranje plastičnosti (uz nelinearno ponašanje materijala), puzanja, deformacijskog ojačavanja, savijanja i deformacije. Izuzetno su zahvalni za modeliranje mreža konačnih elemenata nepravilnog oblika, kakve obično generišu CAD/CAM sistemi.

Ulazni podaci potrebni za kreiranje ovakvih konačnih elemenata su geometrija, koordinate čvorova, koordinatni sistem samog elementa, mehaničke osobine materijala (uz definisanu anizotropnost, ako se razmatra), temperatura, opterećenje i granični uslovi. Izlazni podaci su pomeranje čvornih tačaka, temperatura i naponsko stanje. Pravci u kojima deluje napon odgovaraju koordinatnom sistemu elementa. Naravno, tu su i neizbežna ograničenja, te je važno poštovati sledeća pravila:

1. Površina (2D) / zapremina (3D) elementa ne sme biti manja ili jednaka nuli.
2. Element se ne sme uviti tako da formira dve nezavisne površine / zapremine (najčešće usled pogrešne numeracije elementa)
3. Ukoliko se ukloni čvor iz sredine ivice, podrazumeva se da je pomeranje po njoj linearno (a ne parabolično).
4. Pažnja je potrebna prilikom transformacije elementa u trougao / piramidu. Naime, dimenzije elementa u tom slučaju moraju biti dovoljno male da bi se obezbedio gradijent napona. Trougaona / piramidalna opcija je najbolje rešenje u prelaznim zonama modela.
5. Prilikom transformacije elementa u trougao (2D), tetraedar, prizmu ili piramidu (3D), i dalje se koriste jednačine originalnog oblika. One se naravno prilagođavaju novoj geometriji, ali ipak, rešenje se donekle razlikuje od onog koje bi se dobilo korišćenjem pravog trougla, tetraedra, prizme ili piramide.

Mreža konačnih elemenata je automatski generisana, mapiranjem.

Granični uslovi i parametri proračuna

Granični uslovi moraju obezbediti što realnije ponašanje modela, pa treba voditi računa o tome da je modelirana samo polovina uzorka, da se uzorak na mašini za ispitivanje postavlja na odgovarajuće oslonce, i da u njega u određenom trenutku udara teg i lomi ga. Problem je rešen uvođenjem potrebnih opterećenja i ograničenja pomeranja u odgovarajućim čvornim tačkama

Meshing has been performed using 2D elements of 8 nodes and 3D brick elements of 20 nodes. Each node is provided with three degrees of freedom (translation along x, y and z axis). Element of this type can take any position in the plane / space, and it enables simulation of plasticity (with nonlinear material behaviour), creep, work hardening, bending and deformation. They are exceptionally suitable for modelling finite elements meshes of irregular shape, as usually generated by CAD/CAM systems.

Input data necessary for creation of such finite element are geometry, coordinates of nodes, coordinate system of the element itself, material mechanical properties (with defined anisotropy, if considered), temperature, loading and boundary conditions. Output data are displacement of nodes, temperature and stress state. Directions of acting stress correspond to the elements coordinate system. There are, of course, inevitable limitations and it is very important to observe the following rules:

1. The surface (2D) / volume (3D) of element must not be less or equal to zero.
2. The element must not be bent in a way to form two independent surfaces / volumes (usually due to wrong numbering of elements).
3. If the node is removed from the middle of the edge, translation along the same is linear (not parabolic).
4. Transformation of element into triangle / pyramid must be done very carefully. In fact, dimensions of element in this case must be small enough to ensure stress gradient. Triangular / pyramidal option is the best solution in transition zones of the model.
5. When transforming an element into triangle (2D), tetrahedron, prism or pyramid (3D), the original form of the equations are still to be used. They have, of course, to be fitted to the new geometry, however, the solution differs somewhat from that which would be obtained by using the real triangle, tetrahedron, prism or pyramid.

The finite elemants mesh has been automatically generated, by mapping.

Boundary conditions and calculation parameter

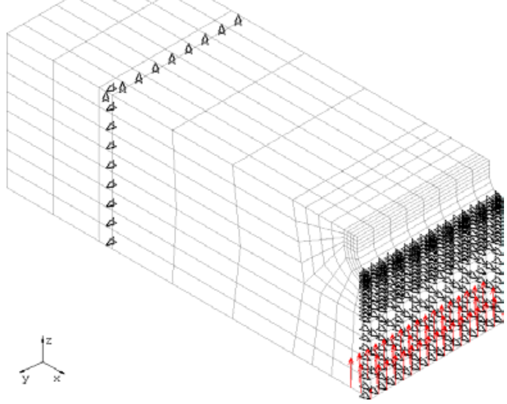
Boundary conditions must assure as realistic behaviour of the model as possible; therefore, care should be taken that only one specimen half is modelled, that the specimen is positioned on appropriate supports on the testing machine and that it will be hit by but at a certain moment and broken. The problem was solved by introducing necessary loading and limitation to displacement in the proper nodes.

Kod 2D modela, sl. 13, pomeranje čvornih tačaka na osloncu je blokirano u y pravcu, na osi simetrije u x pravcu, a na delu ivice gde udara teg je definisano opterećenje kao sila u y pravcu. Kod 3D modela, sl. 14, primenjen je isti princip, stim što su ograničenja uvedena po površini, a ne po ivici modela kao kod 2D modela. Prema tome: pomeranje dodirnih površina sa osloncem je sprečeno u y i z pravcu, površina simetrije u x pravcu, a sile koje simuliraju udarac tega su definisane u y pravcu.

In the 2D model, Fig. 13, nodes displacement on the support is blocked in y direction, on simetry axis in x direction, and on the edge part where the but hits loading is defined as a force in y direction. In 3D model, Fig. 14, the same principle is applied, but the limitations are defined at the surface, and not at the edge as in 2D model. Thus the displacement of contact surfaces on support is blocked in y and z direction, simetry surface in x direction and the forces simulating the weight hit are defined in y direction.

Slika 13. Granični uslovi na 2D modelu

Figure 13. Boundary conditions on the 2D model



Slika 14. Granični uslovi na 3D modelu

Figure 14. Boundary conditions on the 3D model

Važno je da se prilikom modeliranja pomeranja i sile rasporede i na susedne čvorove, jer u protivnom rezultati mogu biti pogrešni zbog velike koncentracije napona samo u jednom čvoru. Ovde je bitna raspodela napona po čitavom modelu, a na nju utiču heterogenost materijala i geometrija zareza / epruvete. Zbog toga je u numeričkom rešavanju problema usvojeno:
- da je opterećenje statičko;
- da je merodavno opterećenje 2500 N; ono je određeno na osnovu sukcesivnih aproksimacija polazeći od izabrane male sile, koja je povećavana sve dok se u homogenom uzorku ne pojavi plastična deformacija; ova vrednost sile je uvećana za 20% i usvojena za proračun; za epruivetu malih dimenzija usvojeno je opterećenje 833 N

Rezultati proračuna simuliranih epruveta

Rezultati opisane numeričke analize se odnose na sve kombinacije, kako sledi:

<u>Epruvete 10x10x55 mm, sa U i V zarezom</u>

1 Homogen uzorak

2;3 Homogen zavareni spoj sa metalom šava veće (2) i manje čvrstoće (3) od čvrstoće metala šava.

4;5;6 Heterogen zavareni spoj, izveden sa oba dodatna materijala, sa zarezom u metalu šava veće (4), odnosno manje čvrstoće (5) ili pak prolazi kroz oba dodatna materijala (6, sl. 15).

<u>Epruveta malih dimenzija 5x5x55 mm, sa V zarezom</u>

7 Homogena epruveta (sl. 16).

U tab. 2 su obuhvaćene maksimalne vrednosti napona po von Mizesu (S) i pomeranja (P), za 2D i 3D analizu i njihove tipične varijante.

In the process of modelling, it is important to distribute the displacements and forces to the adjacent nodes as well, since otherwise the results can be wrong due to high stress concentration in only one node. Here the stress distribution over entire model is substantial, and it is affected by the material heterogeneity and the geometry of notch / specimen. Therefore, in numerical solving of the problem the following has been adopted:
- that the loading is static;
- that appropriate loading is 2500N; this was defined based on successive approximation, starting from chosen small force, which has been increased up to the appearance of plastic deformation in homogenous sample; this value is increased by 20% and accepted for the calculation; for subsize specimen loading of 833 N is accepted..

Results of simulated specimens calculations

The results of the described numerical analyses refer to all combinations, as follows:

<u>Specimens 10x10x55 mm, with U and V notch</u>

1 Homogeneous specimen

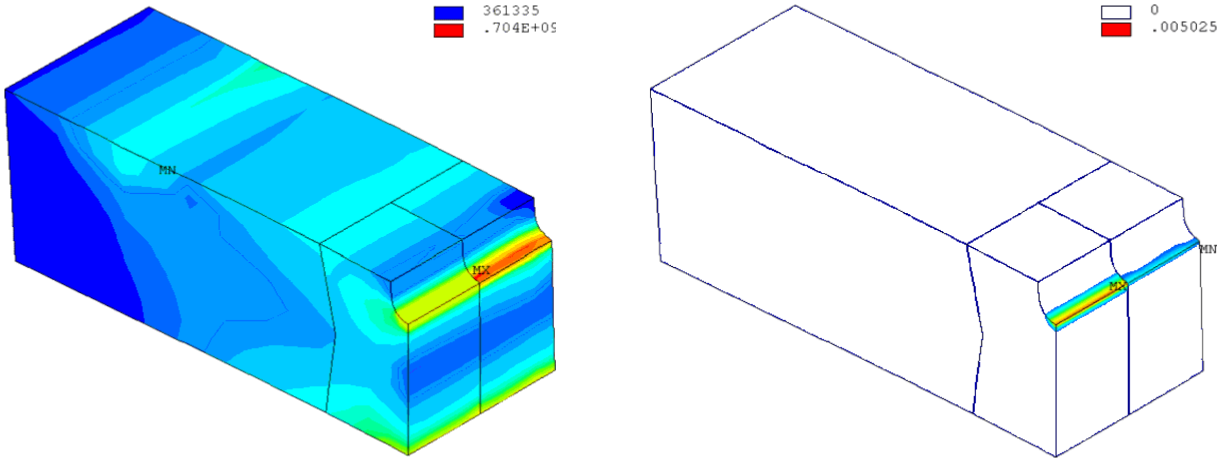
2,3 Homogeneous welded joint, overmatched (2) and undermatched (3).

4,5,6 Heterogeneous welded joint, performed with both filler materials, with notch in overmatched weld metal (4) and in undermatched (5) weld metal, or passing through the both filler materials (6, Fig. 15).

<u>Specimen of subsized dimension 5x5x55 mm, with V notch</u>

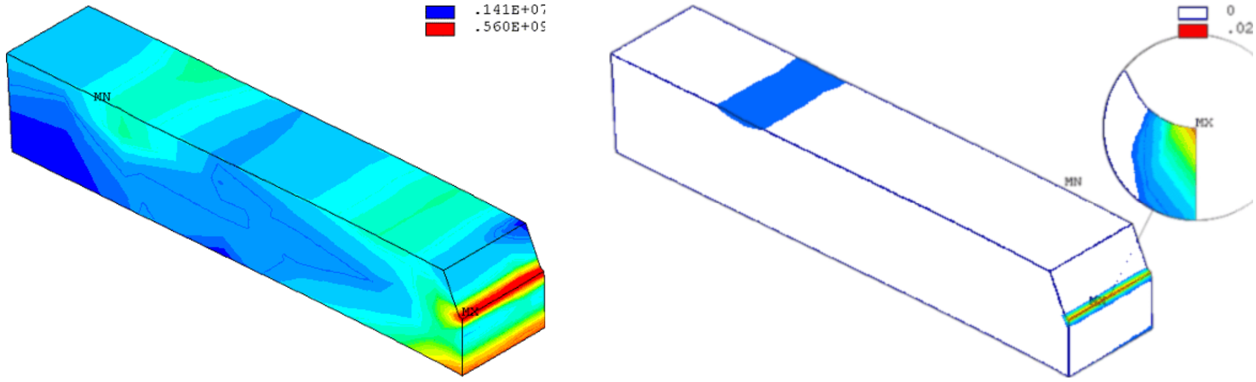
7 Homogeneous specimen (Fig. 16)

Table 2 includes maximum values of stress according von Mises (S) and displacement (P) for 2D and 3D analysis and their typical variations.



Slika 15. U zarez, 3D heterogen zavareni spoj metala šava manje čvrstoće, zarez prolazi kroz oba dodatna materijala

Figure 15. U notch, 3D heterogenous undermatched welded joint, notch passing through both of the filler materials



Slika 16. V zarez, 3D homogena epruveta malih dimenzija

Figure 16. V notch, 3D homogenous subsize specimen

Tabela 2. Rezultati proračuna simuliranih uzoraka

Table 2. Results of simulated specimens calculations

			2D		3D		3D/2D		2D/H		3D/H		2D V/U		3D V/U	
			S	P	S	P	S	P	S	P	S	P	S	P	S	P
STRUKTURA METALA ŠAVA / STRUCTURE OF WELD METAL	Homogen	U	548	0.004959	576	0.003218	1.05	0.65	X	X	X	X	X	X	X	X
		V	561	0.022899	595	0.009996	1.06	0.44	X	X	X	X	1.02	4.62	1.03	3.11
	Over	U	694	0.002993	699	0.001875	1.01	0.63	1.27	0.60	1.21	0.58	X	X	X	X
		V	699	0.016665	656	0.007868	0.94	0.47	1.25	0.73	1.10	0.79	1.01	5.57	0.94	4.20
	OvUnd	U	690	0.002501	689	0.001542	1.00	0.62	1.26	0.50	1.20	0.48	X	X	X	X
		V	695	0.015293	653	0.007187	0.94	0.47	1.24	0.67	1.10	0.72	1.01	6.11	0.95	4.66
	Under	U	527	0.006982	523	0.004597	0.99	0.66	0.96	1.41	0.91	1.43	X	X	X	X
		V	524	0.029082	518	0.012565	0.99	0.43	0.93	1.27	0.87	1.26	0.99	4.17	0.99	2.73
	UndOv	U	519	0.007689	515	0.005065	0.99	0.66	0.95	1.55	0.89	1.57	X	X	X	X
		V	516	0.031331	510	0.013467	0.99	0.43	0.92	1.37	0.86	1.35	0.99	4.07	0.99	2.66
	Horiz	U	X	X	X	X	X	X	X	X	1.01	2.68	X	X	X	X
		V	X	X	X	X	X	X	X	X	1.00	1.72	X	X	0.93	2.70
	Sub	V	580	0.050906	560	0.021943	0.97	0.43	1.03	2.22	0.81	2.20	X	X	X	X

DISKUSIJA REZULTATA

Diskusija rezultata je zasnovana na analizi raspodele izonaponskih polja i ostvarene plastične deformacije u epruveti. Simulirani epruvete imaju mali mečing faktor (1,18 za metal šava veće čvrstoće i 0,86 za metal šava manje čvrstoće). Numerička simulacija je pokazala sledeće:

RESULTS DISCUSSION

The results discussion is based on the analysis of iso-stress fields distribution and the achieved plastic deformation in the specimen. The simulated specimens have small mismatching factor (1.18 overmatch and 0.86 undermatch). Numerical simulation showed the following:

Kako je i očekivano, tri su mesta izražene raspodele napona na epruveti: koren zareza, mesto udara tega i područje kontakta epruvete sa osloncem. Raspodela i intenzitet napona zavise od materijala u zavarenom spoju.

Maksimalni naponi se kod 2D modela sa V zarezom razlikuju od vrednosti za U zarez za -1% (homogen i heterogen zavareni spoj sa metalom šava manje čvrstoće) do 2% (homogen zavareni spoj), a kod 3D modela od -7% (heterogen spoj sa zarezom koji prolazi kroz oba dodatna materijala) do 3% (homogen zavareni spoj).

Plastična deformacija u 2D analizi se razlikuje kod V zareza u odnosu na U zarez od 407% (heterogen spoj metala šava manje čvrstoće) do 611% (heterogen spoj metala šava veće čvrstoće), a u 3D analizi za 266% (heterogen spoj metala šava manje čvrstoće) do 466% (heterogen zavareni spoj metala šava veće čvrstoće).

Kod modela epruvete malih dimenzija uočeno je veće rasipanje polja napona oko oslonca, koje se proteže sve do krajnje (slobodne) ivice, a u području oko ose simetrije više nema karakterističnog pada napona. Mesto pojave maksimalnog napona kod ovog modela je isto kao i kod standardne epruvete sa V zarezom, ali je njegova vrednost za 5% veća. Plastična deformacija je za model malih dimenzija veća za 263%. Maksimalni napon u 3D modelu je manji za 19%, a plastična deformacija veća za 220%.

Kada se zarez nalazi u zavarenom spoju metala šava manje čvrstoće (kako za homogen tako za heterogen spoj, izuzimajući slučaj kada zarez prolazi kroz oba dodatna materijala), maksimalna vrednost napona je u području udarca tega iako je maksimalna deformacija u korenu zareza. U svim ostalim slučajevima, bez obzira na dimenzije epruvete, geometriju zareza ili primenjenu kombinaciju materijala, maksimalne vrednosti napona i plastične deformacije su u području korena zareza.

ZAKLJUČCI

Na osnovu diskusije dobijenih rezultata, mogu se izvesti sledeći zaključci:

U radu primenjena numerička simulacija omogućava proračun složenih realnih problema, i pruža precizne podatke o naponskom i deformacionom stanju unutar modela, kao i o uticaju geometrije i heterogenosti materijala na te veličine.

U ovakvim slučajevima 2D simulacija može dati rezultate koji su dovoljno pouzdani za približnu procenu. Ukoliko se žele precizni rezultati, koji odgovaraju eksperimentalnim vrednostima, mora se izvesti 3D analiza. Do razlike u rezultatima 2D i 3D analize dolazi zbog činjenice da se u 2D analizi ne uzima u razmatranje deformacija modela po debljini (po *z*-osi).

Potvrđeno je da zavisnost napona i deformacije ne mora biti linearno-elastična. Naime, ako zarez prolazi kroz metal šava manje čvrstoće (kako za homogen tako i za heterogen zavareni spoj), maksimum napona je u području udara tega, a maksimum deformacije u korenu zareza. U ostalim slučajevima, oba maksimuma su u korenu zareza.

Bilinearna kriva zatezanja može obezbediti dovoljno precizne rezultate, uzimajući u obzir da su modul ojačanja i

As expected, the stress distribution on the specimen is expressed in three locations: notch root, but impact point and specimen contact with support. Stress distribution and intensity depend on materials in the welded joint.

Maximum stresses in the 2D model with V notch differ from the values of U notch from -1% (homogeneous and heterogeneous undermatched welded joint) to 2% (homogeneous joint), and in 3D model from -7% (heterogeneous joint, the notch passing through both filler materials) to 3% (homogeneous welded joint).

Plastic deformation in the 2D analysis differs for the V notch in comparison to the U notch from 407% (heterogeneous undermatched joint) to 611% (heterogeneous overmatched joint), and in 3D analysis from 266% (heterogeneous undermatched joint) and 466% (heterogeneous overmatched welded joint).

In the subsize specimen model larger scattering of stress field has been observed, extended to the very end (free) edge, while in the region of axes of symmetry there is no more typical stress drop. The location of maximum load in this model is same as in V notched standard specimen, but its value is 5% higher. Plastic deformation in the subsized model is higher for 263%. Maximum stress in 3D model is smaller for 19% and plastic deformation is higher for 220%.

When the notch is in the undermatch welded joint (for the homogeneous joint as well as for heterogeneous one, except the case when the notch passes through both filler materials), maximum stress value is in the region of the but impact, although maximum deformation is in the notch root. In all other cases, regardless specimen dimensions, notch geometry or the applied combination of the materials, maximum values of stress and plastic strain are in the region of notch root.

CONCLUSIONS

Based on the discussion of obtained results, the following conclusions may be derived:

Numerical simulation, applied in the paper, enables calculation of complex, real problems, and provides precise data about stress and deformation state within the model as well as about the effect of geometry and material heterogeneity on those parameters.

In such cases, the 2D simulation can provide the results, which are sufficiently reliable for approximative assessment. If precise results are required, that corresppond to the experimental values, 3D analysis should be performed. The differences between the results of 2D and 3D analysis are due to the fact that in the 2D analysis the strain of the model along the thickness (*z*-axis) has not been considered.

It is confirmed that stress - strain dependance need not be linear - elastic. In fact, when notch passed undermatched weld metal (for both homogeneous and heterogeneous welded joints), the stress maximum is in the but impact region, and strain maximum in the notch root. In other cases, both maxima are in the notch root.

Bilinear tensile curve may provide sufficiently precise results, taking into account that the work hardening module veličina plastične deformacije obrnuto proporcionalne.

Uticaj geometrije koncentratora napona na stanje napona i deformacije modela je veliki. Malim povećanjem radijusa korena zareza (sa 0,5 mm na 2mm, pri istoj dubini zareza), plastična deformacija se smanjuje nekoliko puta, iako je polje napona samo neznatno promenjeno.

Homogen zavareni spoj metala šava veće čvrstoće je ocenjen kao najbolja kombinacija, jer je ostvaren maksimalni napon, a veličina plastične deformacije je mala.

Plastična deformacija se dalje može smanjiti ako se kombinuju dodatni materijali veće i manje čvrstoće, sa koncentratorom napona u prvom slučaju, a mestom udara tega u drugom slučaju.

Kod modela malih dimenzija, zbog malog poprečnog preseka velika su rasipanja polja napona na osloncima, što bitno utiče i na vrednosti napona i plastične deformacije.

and value of plastic strain are in inverse proportion.

The effect of stress riser geometry on stress and strain state of the model is significant. By small increase of the notch root radius (from 0.5mm to 2 mm at the same notch depth), plastic strain is decreased several times, although the stress field change is negligible.

Homogeneous overmatched welded joint has been evaluated as the best combination. exhibiting maximum stress, followed by small plastic strain value.

Plastic strain may additionally be reduced combining overmatched and undermatched filler materials, with stress riser is the first case, and but impact location in the second case.

In the subsized model, due to small cross-section great scattering of the stress field over the supports occurr, with substancial effect on the values for both the stress and the plastic strain.

LITERATURA – REFERENCES

1.From Charpy to present impact testing, ESIS Publication 30, © ELSEVIER Science 2002, Historical Background and Development of the Charpy Test, p3-21, L. Tóth, H.P. Rossmanith, T.A. Siewart, Development in The Instrumented Impact Test, Computer Aided Impact Testing System -, p165-173, Toshiro Kobyashi, Toyohashi University of Technology, Japan

2.Procedures And Problems Associated with Reliable Control of the Instrumented Impact Test, Ireland, D.R, ASTM STP 563

3.Primena mehanike loma na procenu integriteta konstrukcije, A. Sedmak, Monografija, Mašinski fakultet u Beogradu 2002.

4.Analiza uticaja geometrije i heterogenosti materijala na ponašanje Šarpijeve epruvete, magistarski rad, M. Dobrojević, Mašinski fakultet u Beogradu, 2003.

5. Effect of Strength Mis-match Weld Metal Portion on Charpy Impact Toughness, N. Gubeljak, M. Kozak, J. Predan, M. Oblak, 2002.

6. Uticaj oblika zareza na probnoj epruveti pri određivanju žilavosti materijala, V. Keserović, diplomski rad, Tehnološko-metalurški fakultet, Beograd, 1972.

7. Preporuke za primenu metode konačnih elemenata u mehanici loma, VI Međunarodna letnja škola mehanike loma, Vrdnik, N. Savović, A. Sedmak, TMF-"GOŠA"-ECMR, 1994

8. Primena parametara elasto-plastične mehanike loma na procenu integriteta zavarenih konstrukcija sa površinskom prslinom - doktorska teza, D. Jaramaz, Mašinski fakultet u Beogradu, 2002.

INTERNET:

www.yeji21.com	© 1999-2001. Yeji Corporation. All rights reserved
www.mpmtechnologies.com	MPM Technologies, impact test machines manufacturer
www.matweb.com	© 1999-2001 by Automation Creations, Inc. MatWeb Property Values for Plastic, Metal Alloys and other Engineering Materials
www.nist.gov	© 2002 National Institute of Standards and Technology NIST, agency of the U.S. Commerce Department's Technology Administration.