Theoret. Appl. Mech., Vol.32, No.2, pp. 97–112, Belgrade 2005

Limit load solution for mismatched welded plate and pressure vessel with a surface crack

Emhamed Argoub [*] **Aleksandar Sedmak** [†]
Stojan Sedmak [‡] **Gorgi Adziev** [§]

Abstract

Fracture behavior of a structure having a crack in the middle of a weld is influenced by the mechanical properties of the welded joint constituents (weld metal, base metal, heat-affected-zone) and their geometry. Therefore, defect assessment procedures have been developed in order to take into account all those affecting parameters. One of the key point for all those defect assessment procedures is to have limit load solutions of the mismatch structure. Such limit load solutions have been obtained by using 3D finite element solutions for different configurations such as plate with surface crack in tension or pressure vessel with axial external surface crack exposed to internal pressure, since those limit load have been obtained for non-hardening material exponent.

Keywords: Limit load, overmatching, undermatching, FEM - finite elements method, 3D, surface crack plate, pressure vessel, center cracked tensile specimen

[*]Faculty of Mechanical Engineering, University of Belgrade, Serbia and Montenegro
[†]Faculty of Mechanical Engineering, University of Belgrade, Serbia and Montenegro
[‡]Faculty of Technology and Metallurgy, University of Belgrade, Serbia and Montenegro
[§]Faculty of Mechanical Engineering, University of Skopje, Macedonia

Introduction

Generally, metallic materials harden progressively with plastic strain, but reach limit states where additional hardening becomes impossible. Structures constructed from such materials also have a maximum load limit at which they lose the ability to bear an additional increase of loading [1]. The applied load at this limit is called the limit load and plays an important role in the prediction of ductile fracture in various ways. Usually, limit loads can be obtained by analytical or numerical evaluations assuming rigid or elastic-perfectly plastic material behavior. They may be determined by experimental methods. In some cases, the limit load solutions are directly used for prediction of fracture strength. The yield load is an important input parameter of defect assessment procedures using engineering approaches, such as the Engineering Treatment Model (ETM) proposed by Schwalbe and Cornec [2] and modified for welded structures (ETM-MM) by Schwalbe [3] and the ARAMIS method proposed by Gilles and Frano [4], developed for analysis of cracked components. The limit loads are an important input to defect assessment methods not only because they impose a limit on the load that can be applied, but also because they are useful in predicting response at lower loads, as it is clear from the Fracture Analysis Diagram (FAD) approach [1]. In this paper elastic-plastic 3D finite element analysis calculation are used by increasing the load until convergence is no longer possible. The limit load for plate and pressure vessel with surface crack have been compared with limit load for centre-crack tensile (CCT) plate obtained by Schwalbe.

Use of Finite-Element Analysis

The flawed component may be analyzed using a small displacement finite-element method with an elastic-perfectly plastic material model [5]. The analysis is performed for a monotonically increasing load; the maximum load attained corresponds to the limit load. The elastic properties assumed in the analysis do not affect the plastic yield load in a small displacement analysis.

In the finite-element method, convergence problem arises as the load approaches the limit load. Therefore, a finite-element package, which has been validated for plastic collapse analysis, must be used when this approach is adopted. However, a lower estimate of plastic yield load is obtained by taking any load at which the analysis remains convergent and in many cases may lead to a satisfactory assessment.

The finite-element method has not been widely used for obtaining plastic yield loads of flawed components, but has been used to determine limit loads by the methods of limit analysis rather than by performing an incremental elastic-plastic solution. However, results have been obtained largely for plate geometries and for complex components. With the ever-increasing power of digital computers such numerical solutions are likely to become more widely available.

Numerical simulation

Geometry of specimen and pressure vessel

To get more insight in the effect of geometry, specimen with surface crack, Table 1 and Fig. 1, and pressure vessel with axial external surface crack, Table 2, are used in this study.

Table 1: Input data for FEM analysis of cracked plate

Item	Denotation	Dimensions, mm
Length	$2L$	500
Width	$2W$	100
Thickness	B	50
Width of welding	$2H$	4-10-20-50
Crack length	$2a$	50
Crack depth	c	25

Materials properties

The materials of same properties have been used for plate and pressure vessel with surface crack, for homogenous and heterogeneous welded joints. These properties are shown in Table 3.

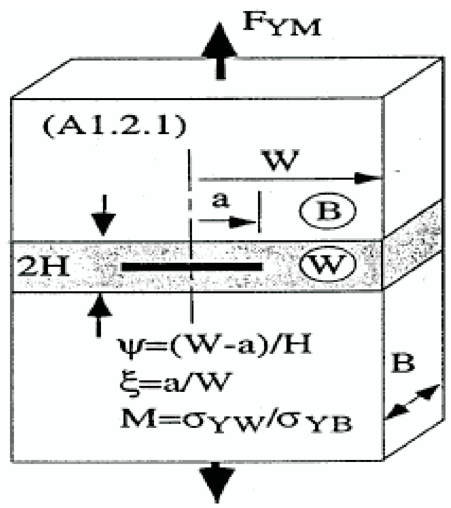

Figure 1: Mis-matched surface crack plate

Table 2: Input data for FEM analysis of pressure vessel with axial surface crack in BM and WM

Item	Denotation	Dimensions, mm
Outside diameter	D	200
Inside diameter	d	190
Thickness	B	5
Crack depth	a	2.5
Crack length	$2c$	40
Crack tip		0.2
Weld metal length	H	2; 5; 10; 25

Table 3: Input data for FEM analysis of mechanical properties of BM and WM

	Yield strength	Elasticity modulus	Poason's ratio
Item	σ_Y, MPa	E, GPa	ν
Base metal	560	200	0.3
Overmatched weld metal	M=1.3; 1.5; 1.75; 2; 3	200	0.3
Undermatched weld metal	M=0.35; 0.5; 0.6; 0.75	200	0.3

Matching factor M is defined as ratio of weld metal and base metal yield strength.

Modeling

The basic idea was to make a model, which will exactly represent the reaction of the body to loading. The models of specimen and pressure vessel r shown in Fig.2 and 3. Calculation of displacement, deformation and stress have been done by using ANSYS [6]. In all alternatives, only one quarter of the specimen for plate with surface crack and only 1/8 of pressure vessel with axial surface crack has been used due to symmetry.

Finite Element Mesh

Since the processing speed is dependent directly on the number of FE, optimization has been done, ensuring that the mesh is the finest at the crack tip (greater number of small elements), within the welded joint zone, and the rest of network being filled with small number of greater elements.

The region with crack required special manual and semi-manual procedure. First 2D eight-node (thin shell) elements were generated, which are then "deformed" to take the singularity into account, i.e. to form the crack tip and model elastic-plastic behaviour at the same time. Then, on the basis of these plane elements, 3D region of elastic-plastic singularity has been formed by using the appropriate mapping contour (Fig. 4). After three-dimensional singular elements formed the crack front, they are merged with the surrounding uniform mesh, consisting of hexaedric elements (standard 20 node element). Each node is provided with three degree of freedom (translation along x, y and z axis). Element of such a type can take any position in the plane/space, and enables simulation of plasticity.

Boundary conditions

Boundary conditions have to correspond to real behavior of a model, therefore it should be taken into consideration that only quarter of a specimen is modeled and only 1/8 of pressure vessel is modeled. The specimen is positioned on appropriate supports of testing machine and pressure vessel adopted in proper way for pressurizing by hydraulic pump. The problem was solved by introducing necessary loading and prescribed displacement in the suitable nodes and the boundary condition are shown in Fig.5a for plate with surface crack and in Fig. 5b for pressure vessel.

Static loading

At first, low load values have been applied and increased until appearance of plastic deformation in homogenous plate and vessel. Since there is no equation for surface crack to define limit load, the load has been increased up to maximum load when the plastic zone reaches the interface as shown in Fig. 6. Above this load the convergence is no longer possible, but for both over and under-matching case the load has been chosen by taken the ratio of F_Y^{WM}/F_Y^{BM} (yield load F_Y for weld metal WM and base metal BM) from result for central cracked tensile (CCT) specimen (Schwalbe) [5] as the start point, then the load increased or decreased depending on the solution convergence.

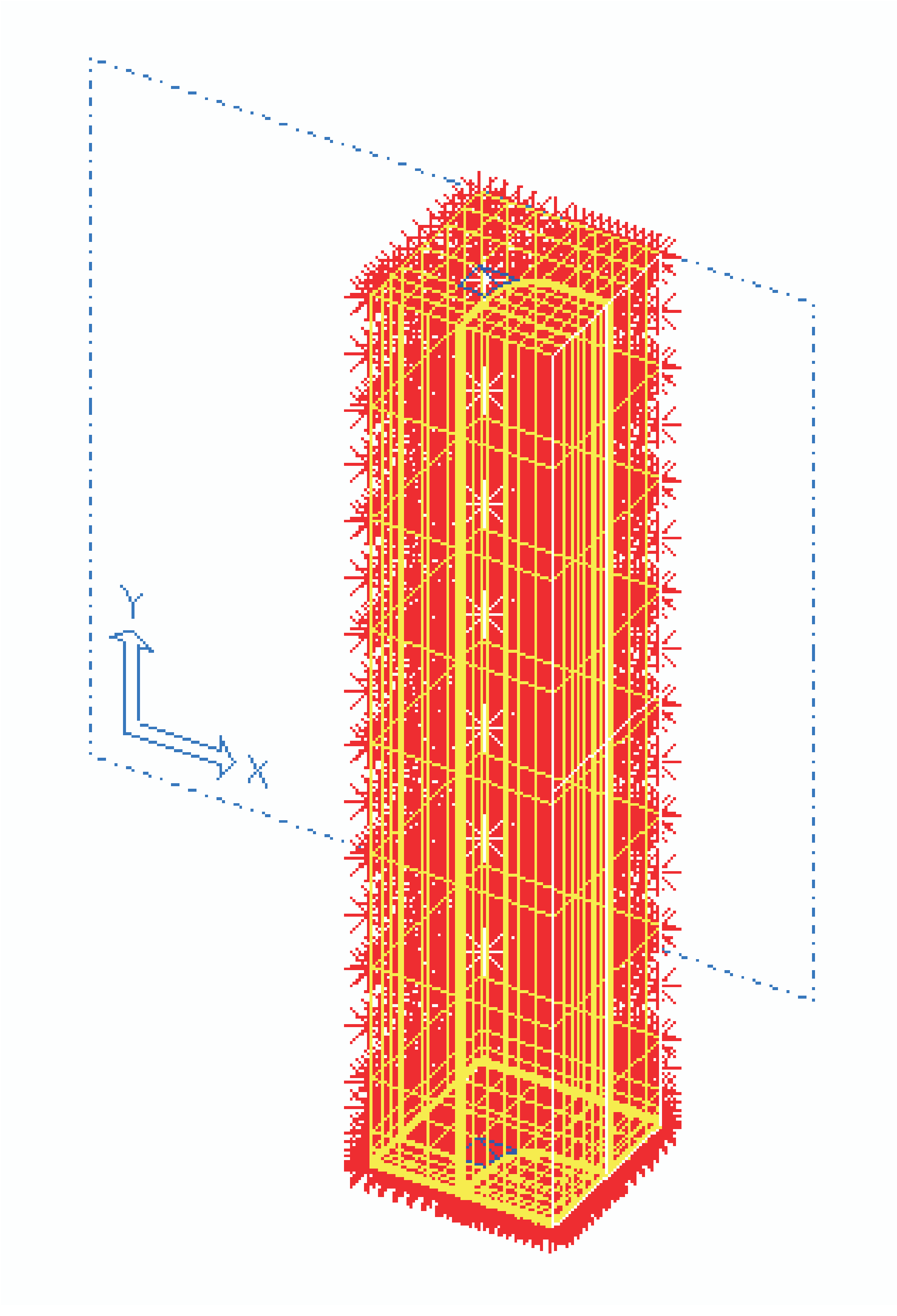

Figure 2: 3D model for plate

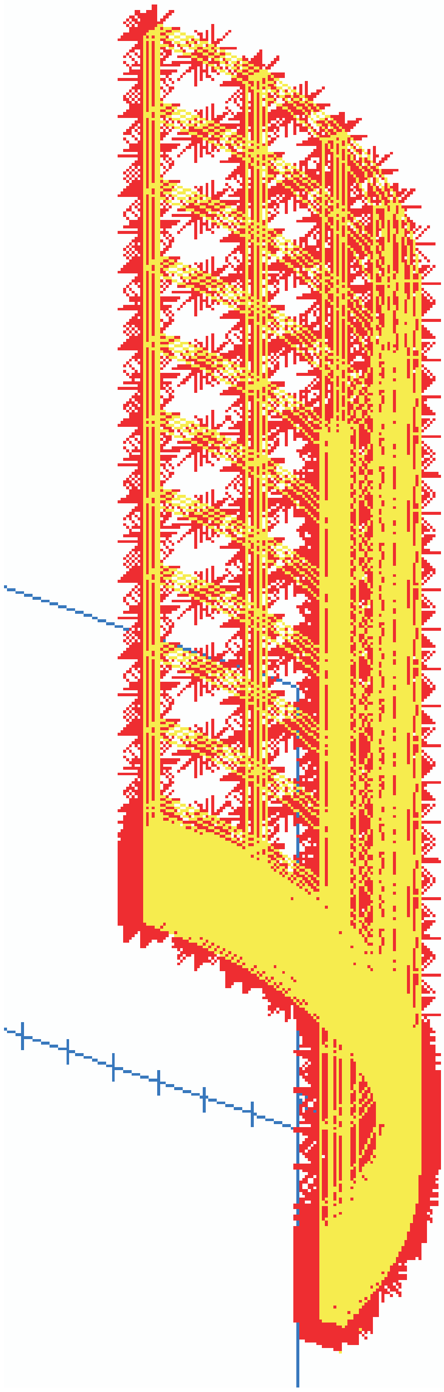

Figure 3: 3D model for pressure vessel

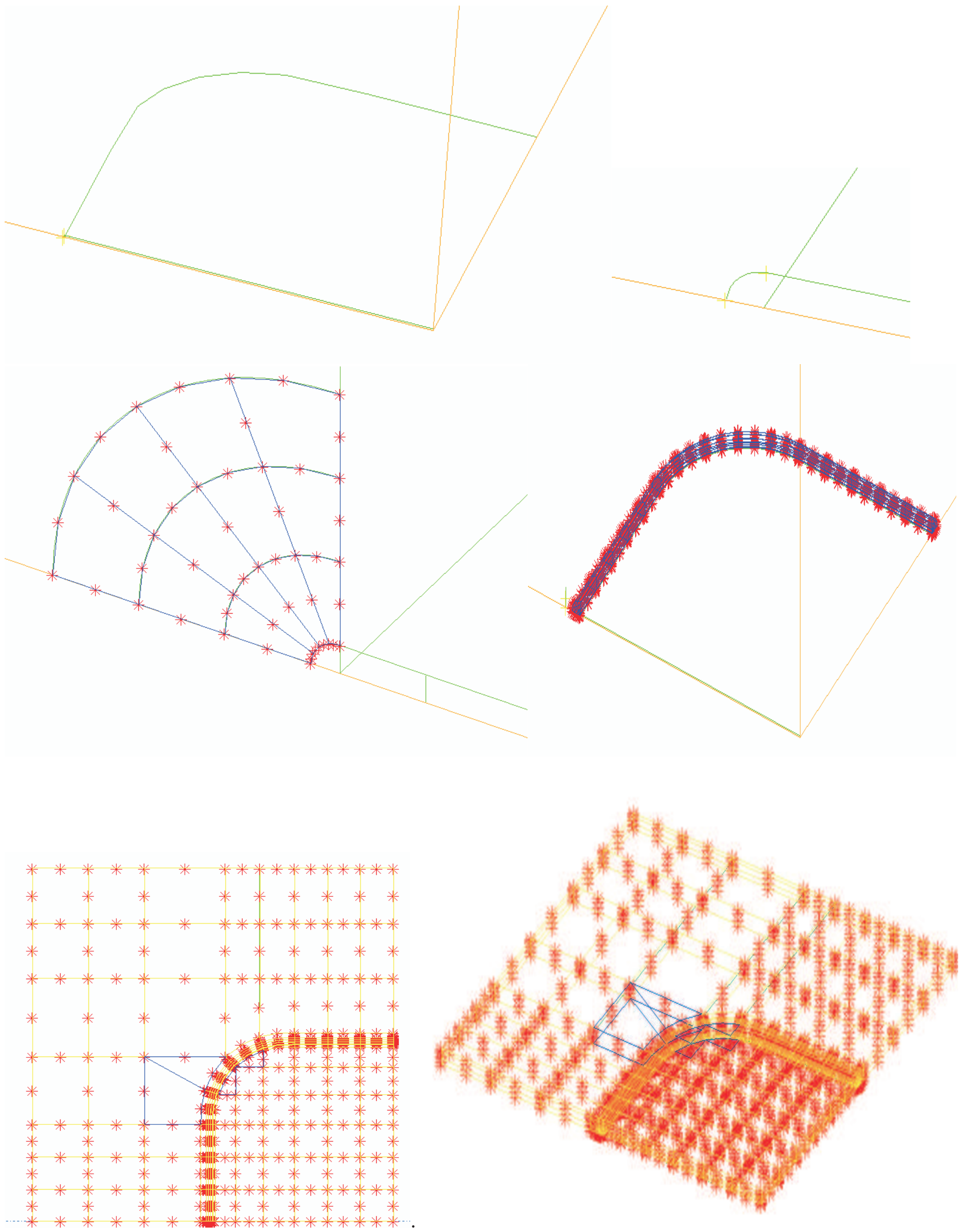

Figure 4: Mapping contour for surface crack in plate

Results

The results of performed analyses are compared in Figs. 7 to 9 for plate with surface crack and CCT specimen, in Fig. 10 for plate with surface crack and pressure vessel and in Fig. 11 for pressure vessel and CCT specimen. Maximum limit load of plates with surface crack and vessels for homogenous material are given in Table 4.

Table 4: Maximum limit load of plates with surface crack and vessels for homogenous case

	CCT Plate	*Surface crack in plate*	*Pressure vessel*
Limit load (MPa)	320	475	25

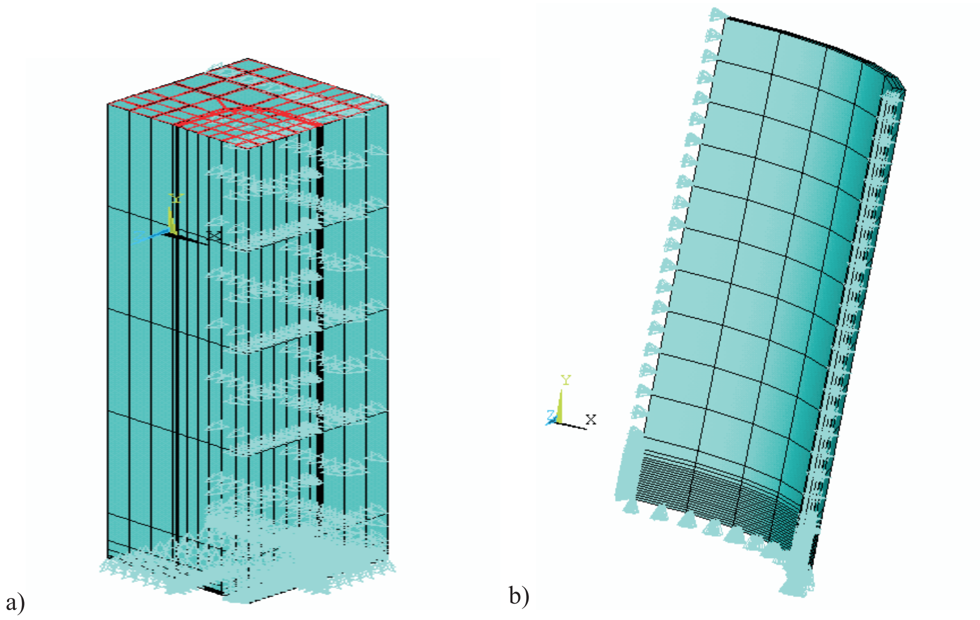

Figure 5: Finite element mesh for plate with surface crack (a) and pressure vessel (b)

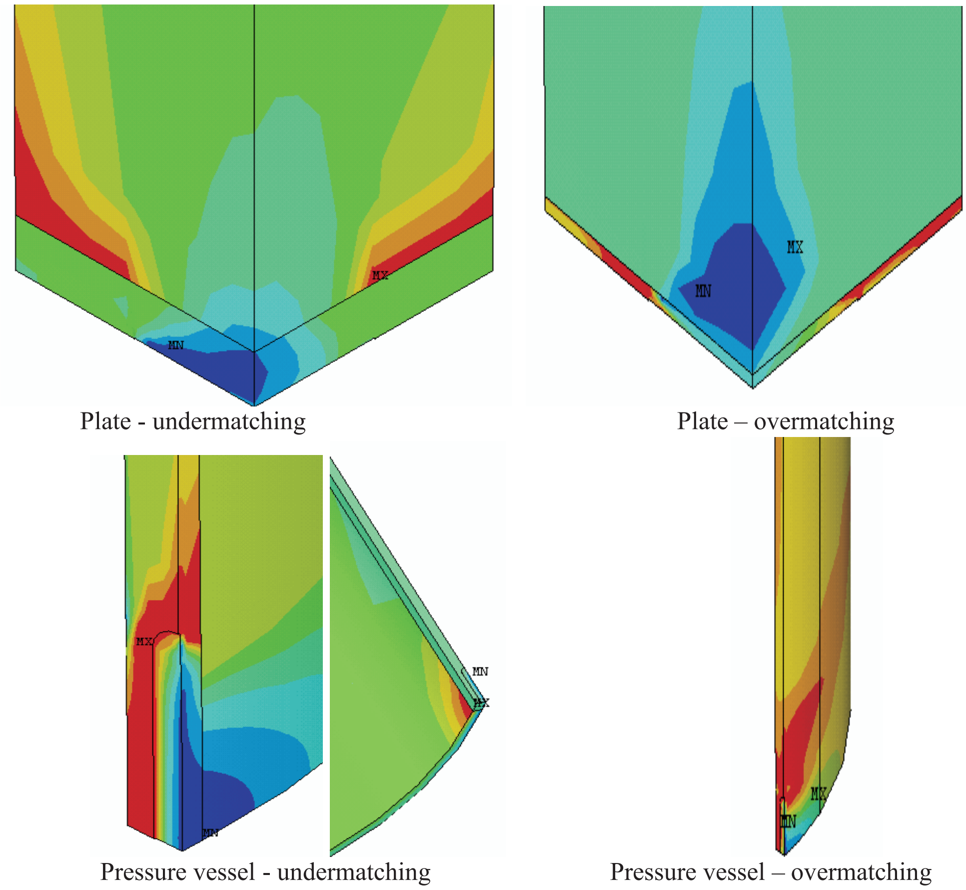

Figure 6: Final stage in increasing load: plastic zone reaches the interface

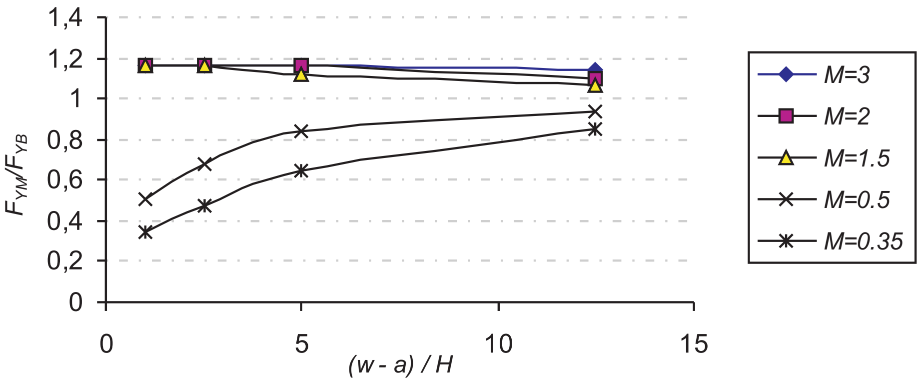

Figure 7: Limit load for different mismatch factor M of plate with surface crack for a/w= 0.5

Discussion and conclusions

Based on the results, the following conclusions may be drawn:

- Numerical simulation enables calculation of complex, real problems, which often couldn't be solved in an analytical manner. What is even more important, numerical simulation may provide precise information on the stress and deformation state within the model and effect of geometry and heterogeneity of material.
- Figure 8 shows good agreement between limit load ratio for plate with surface crack and central cracked plate for undermatched case

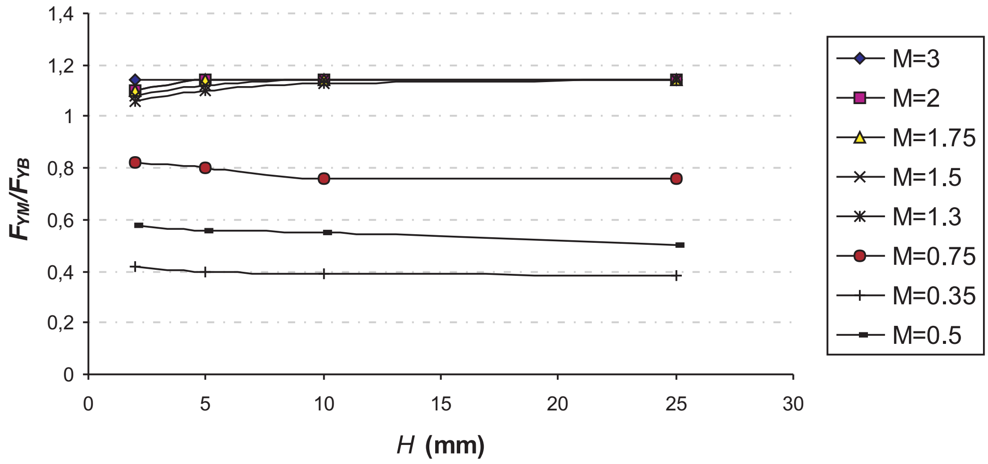

Figure 8: Limit load for different mismatch factor M for pressure vessel for t/a = 0.5

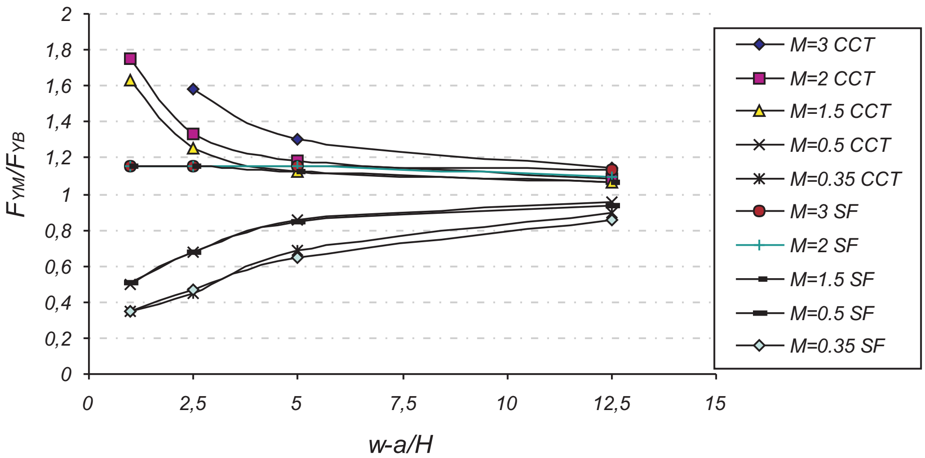

Figure 9: Limit load for different matching of CCT plate in plane strain and plate with surface crack for $a/W = 0.5$

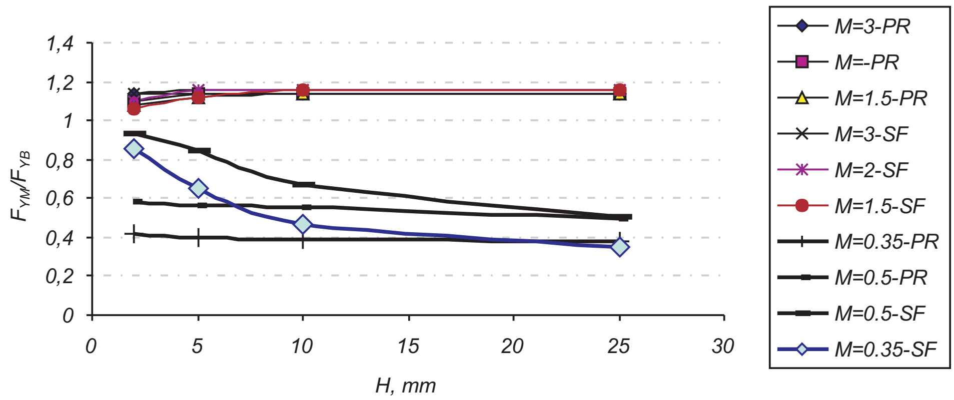

Figure 10: Limit load for different matching of pressure vessel for t/a = 0.5, and plate with surface crack

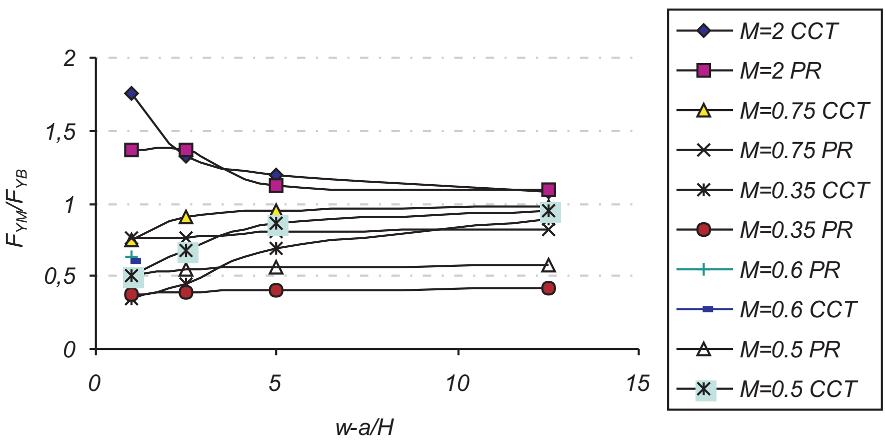

Figure 11: Limit load for different matching of CCT plate in plane strain for a/W = 0.5 (2D) and pressure vessel (3D)

and the ratio from CCT can be used. Anyhow, for overmatched welded joint the agreement between them are very close for higher values of ligament ψ*=(W-a)/H,* and not so good for lower values of ψ. It also shows small difference for limit load ratio for overmatching for different values of ψ for plate with surface crack.

- Figure 10 shows that in overmatched case the limit load ratio between plate with surface crack and pressure vessel the agreement is very good, with only small difference for lower values of weld metal width H, but for undermatched case the agreement is obtained only for higher values of H.

- Figure 11 shows that in overmatched case the limit load ratio between pressure vessel and CCT, at lower values of H, is in the very close agreement, but at higher values of H there is no agreement. For the undermatched joint the results are opposite, i.e. for the lower values of H there is no agreement, whereas for higher values of H, agreement is very good.

- This difference in results between the three models, CCT, pressure vessel and plate with surface crack, can be attributed to different constraint effect, different geometry and applied load.

References

[1] Comprehensive Structure Integrity, Practical Failure Assessment Methods Handbook, by Milne, Karihaloo, Ritchie, Volume 7, 2003

[2] Schwalbe K-H. Cornec A. The engineering treatment model (ETM) and its practical application. Fatigue Fract Engng Mater Struc 1991: 14(4):405-12

[3] Schwalbe K-H. Effect of weld metal matching on toughness requirments: some simple analytical considerations using the engineering Treatment Model (ETM). Int J Fracture 1992:56:257-77

[4] Gilles Ph. Franco Ch. A new J-estimation scheme for cracks in mismatching welds - the ARAMIS method in: Schwalbe K-H. Kocak M. editors. Mis-matching of welds. ESIS 17. London: mechanical Engineering Publications. 1994.p. 661-83

[5] Structural Integrity Assessment Procedure For European Industry, SINTAP, project No. BE95-1426,1999

[6] ANSYS Theory Reference

Submitted on May 2005.

Rešenje graničnog opterećenja za nesrodnu zavarenu ploču i sud pod pritiskom sa površinskom prslinom

UDK 539.42

Na ponašanje strukture pri lomu, koja ima prslinu na sredini vara, utiču mehaničke osobine sastavnih delova zavarenog spoja (metal šava, osnovni materijal, zona uticaja toplote) i njihova geometrija. Zbog toga su razvijeni postupci procene oštećenja koji treba da uzmu u obzir sve te uticajne parametre. Jedna od ključnih stvari u ovom postupku je da se dobiju rešenja graničnih opterećenja nesrodne strukture. Ova rešenja su dobijena korišćenjem $3D$-rešenja metodom konačnih elemenata za različite konfiguracije kao što su: zategnuta ploča sa površinskom prslinom ili posuda pod pritiskom sa spoljnom osnom površinskom prslinom izložena unutrašnjem pritisku, pošto su ova granična opterećenja dobijena materijalnim eksponentom za slučaj bez ojačanja.