

COMPUTER SOFTWARE FOR THE STRESS ANALYSIS OF REPAIRED AIRCRAFT STRUCTURES WITH THE METHOD OF COMPOSITE PATCH REPAIR

UDC 629.73 624.042.1

George Tsamasphyros, Zaira Marioli-Riga, Chris Vrettos

Department of Mechanics, General Section, National Technical University of Athens

Abstract. *A computer program for the solution of fracture mechanics problems is presented. The software includes a mesh generator routine for the creation of the geometry and a finite element routine for the calculation of values under interest such as Stress Intensity Factors, Stresses etc. The program is capable of dealing with the stress analysis of cracked or corroded metallic aircraft structures. One of the most modern methods of repairing such types of damage is the composite patch repair method. The principal of the method is the application of a patch made of composite material (e.x. carbon, boron) over the damaged area. If the damage type is a crack the scope of the repair is the reduction of the Stress Intensity Factor in the metallic structure. In the case of a corrosion, the scope of the repair is the reduction of the Maximum Von Mises Stress in the metallic structure. The program described herein – assisted by its user friendly interface- is a tool for the quick analysis of the cases described above.*

1. INTRODUCTION

Up to now the most popular method for repairing damaged aircraft structures has been the application of metallic stiffening surfaces over the damaged area. The role of these stiffening surfaces is to carry the loads that cannot be carried by the main damaged structure anymore with safety. The case of a cracked aircraft skin is a typical case. The most common way of repairing this kind of damage is riveting a stiffening plate over the cracked area. Eventhough this is an acceptable repair method, yet it is not the optimum for many reasons: New cracks may initiate at the proximity of the rivet holes, disassembly of the structure is often necessary, a significant amount of machining work is usually required etc.

For the reasons mentioned above, research is taking place by several researchers and aircraft industries into developing alternative repair methods. As a result of these efforts the method of "Composite Patch Repair" has been developed eventhough it is not yet applied to real aircraft since it has not yet been established as an "official" repair procedure.

The basic concept of this repair method is that instead of riveting a metallic patch over the damaged area, a patch made of composite material may be bonded onto the damaged area. The patch material is usually carbon or boron pre-preg.

A typical repair sequence for repairing a crack is the following:

- Cleaning / surface preparation of the area to be repaired
- Cutting of the pre-preg material at the desired size and shape
- Precuring of the patch
- Applying an adhesive film over the area to be repaired
- Applying the composite patch over the adhesive film
- Curing the patch / metallic structure together
- Repair is complete

Other types of damage such as corrosion may also be treated with the "Composite Patch Repair" method. In that case a suitable material such as a structural epoxy resin may be used to fill the corroded area.

The "Composite Patch Repair" method has the following advantages over the conventional repair methods:

- There are no rivet holes. New cracks are unlikely to be created and propagate.
- Because the patch is continuously joined with the metallic structure the stress field is quite smooth.
- Machining work is usually negligible.
- Disassembly of the metallic structure is very often not necessary. The bonding of the patch onto the metal may be performed with portable devices.

A very important stage of the repair procedure is the design phase where the patch configuration must be determined. A correct patch design must assure that after the repair no failure will take place in either the patch, or the adhesive film, or the metallic structure under the most severe loadcase. For the stress analysis analytical as well as numerical methods have been developed. The computer software that is described herein is orientated towards the stress analysis of typical damages of aircraft structures such as center cracks, edge cracks, cracks at the proximity of holes, and corrosions. With this software it is possible to automatically create the finite element mesh of the geometry under interest and solve for the displacements and the stresses with the finite element method. The user has also the capability to import the model into the Ansys software and view it. The main benefit of the current software is that within a few seconds the engineer can very easily build and solve a finite element model which if it was to be built with another software it could take a few hours.

2. MESH GENERATION

The present software is capable of dealing with the following cases which are most typical of aircraft skin damage:

- Center crack in a finite sheet (patch may be elliptical or orthogonal)
- Edge crack in a finite sheet (orthogonal patch)
- Radial crack at the proximity of a hole in a finite sheet (elliptical patch)
- Corrosion in a finite sheet (orthogonal patch)

In all the above cases a finite element mesh is created. The mesh is created using conformal mapping or/and laplace equations as described below.

2.1. Center or edge crack in a finite sheet – mesh generation

Consider an elastic and isotropic body containing a center crack of length $2a$. Our interest is focused at the vicinity of the tip of the crack [1]. Let us introduce both cartesian coordinates (x,y) and polar coordinates (r,θ) (figure 1.a) with the origin placed at the crack tip. Using the transformation

$$z = \frac{a}{2} \left(\zeta + \frac{1}{\zeta} \right), \quad z = x + iy, \quad \zeta = \xi + i\eta \quad (1)$$

or in real variables

$$x = \frac{a}{2} \left(\xi + \frac{\xi}{\xi^2 + \eta^2} \right), \quad y = \frac{a}{2} \left(\xi - \frac{\eta}{\xi^2 + \eta^2} \right) \quad (2)$$

the vicinity of the crack tip (the enclosed region in figure 1.a) is transformed to a region (figure 1.b) where both lips of the crack are mapped to the same straight line. At this point it must be noted that points O', A', B', C' (figure 1.b) are the images of points O, A, B, C (figure 1.a) respectively.

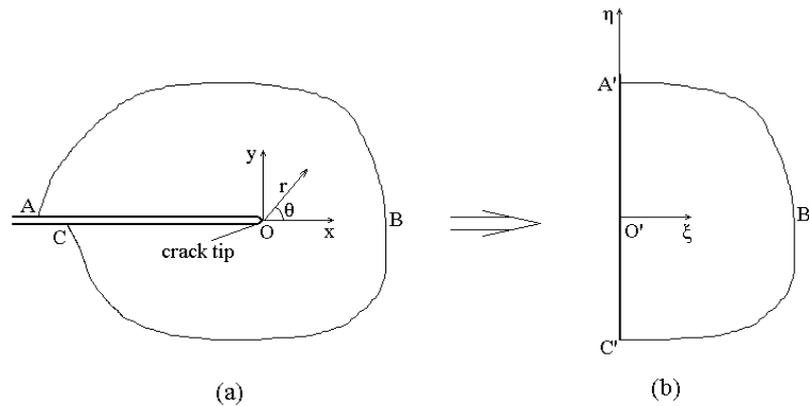


Fig. 1. Mapping of crack tip region

The next step is to define the quantities ξ, η . By using the transformation

$$\xi = r \cos \theta, \quad \eta = r \sin \theta \quad (3)$$

we finally get

$$x = \frac{a}{2} r \left(\cos \theta + \frac{\cos \theta}{r^2} \right), \quad y = \frac{a}{2} r \left(\cos \theta - \frac{\sin \theta}{r^2} \right) \quad (4)$$

A typical mesh created using the previous transformation is presented in figure 2.

In the case of an unrepaired crack, the geometry is flat and two dimensional elements may be used. Most suitable elements for this purpose are the 8 node quadrilateral plane stress element and the 6 node triangular plane stress element. In the case of a repaired crack the geometry is no longer flat and the use of three dimensional elements is necessary. For that purpose the 20 node solid brick and the 15 node solid wedge elements may be used (see figure 3).

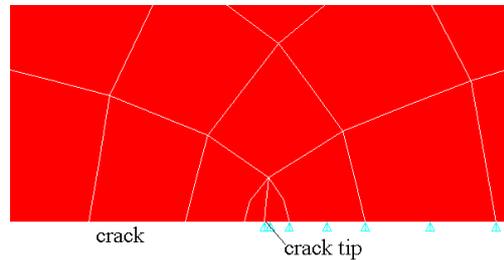


Fig. 2. Mesh around crack tip area (1/4 FE model)

By making use of the proper boundary conditions it is possible to create an edge crack model or a center crack model.

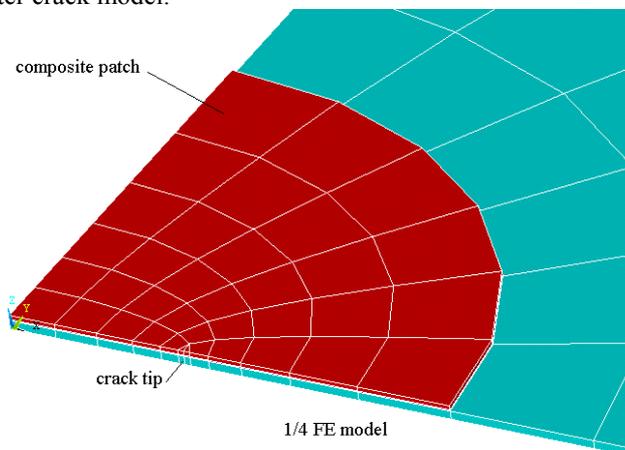


Fig. 3. 3d model of composite patch repair

2.2. Radial crack at the proximity of a hole in a finite sheet– mesh generation

Consider a finite plate with a central circular hole of radius R and a radial crack of length L emanating from the hole. The geometry of the internal boundary can be described by considering the plate as the complex z plane where $z = x + iy$. The center of the hole is chosen at $z = 0$. Figure 4 shows the geometry of the radial crack. Additionally let us introduce an auxiliary plane, the ζ plane where $\zeta = \xi + i\eta$ [2]. Again the transformation $\xi = r \cos\theta, \eta = r \sin\theta$ is required in order to define the variables ξ, η . The finite element mesh may be generated by making use of the following transformation:

$$z = x + iy = \omega(\zeta) = \left\{ \frac{\zeta + \frac{1}{\zeta} + 1 + \varepsilon + \left(1 + \frac{1}{\zeta}\right) \left(\zeta^2 + 2\varepsilon\zeta + 1\right)^{\frac{1}{2}}}{1 - \varepsilon} \right\} R \tag{5}$$

where
$$\epsilon = 2\left(\frac{\lambda}{\lambda + 2}\right)^2 - 1 \tag{6}$$

and
$$\lambda = \frac{L}{R} \tag{7}$$

By separating the real and imaginary parts of Eq (5) the coordinates x, y can be found. A typical mesh created using the previous transformation is presented in figure 5.

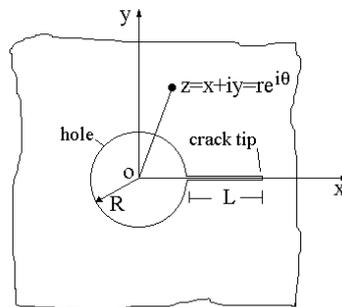


Fig. 4. Geometry of radial crack

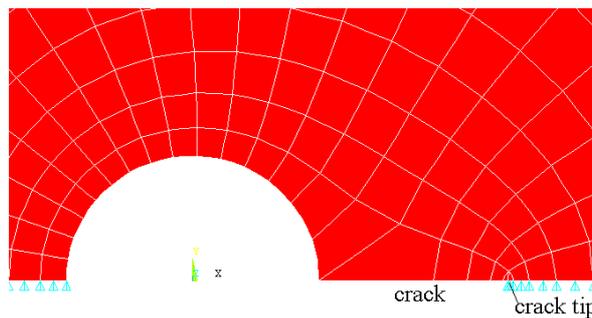


Fig. 5. Mesh around crack tip in the case of radial crack (1/2 FE model)

In the case of an unrepaired crack, the geometry is flat and two dimensional elements may be used. Most suitable elements for this purpose are the 8 node quadrilateral plane stress element and the 6 node triangular plane stress element. In the case of a repaired crack the geometry is no longer flat and the use of three dimensional elements is necessary. For that purpose the 20 node solid brick and the 15 node solid wedge elements may be used.

2.3. Corrosion in a finite sheet– mesh generation

Consider a finite plate of finite thickness t containing a corrosion area of depth d where $d < t$. The corrosion may be approximated as an ellipse with axes a and b . The equation of the ellipse is $x^2/a^2 + y^2/b^2 = 1$. Due to the existence of the corrosion, the geometry is not flat, therefore the finite element mesh is constructed by using 2 layers of 3d solid elements. The first layer of elements represents the healthy metal under the corrosion

area of thickness $t - d$ while the second layer of elements represents the corroded metal of thickness d . The finite element mesh inside the corrosion area as well as outside is a structured mesh and is generated by making use of the elliptical laplace equations. As a result a smooth mesh is created (figure 6).

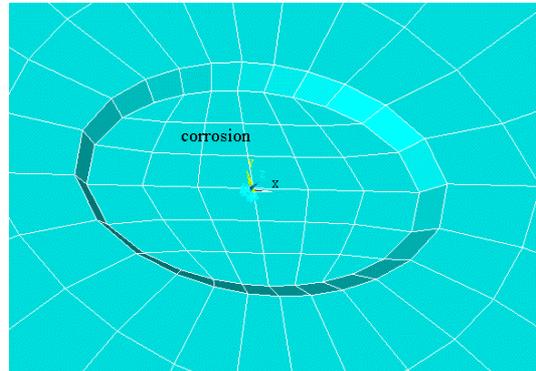


Fig. 6. Detail of mesh around the corrosion area

When a corrosion defect is to be repaired with a composite patch the corrosion cavity may be filled with a filler material such as a structural epoxy resin as shown in figure 7.

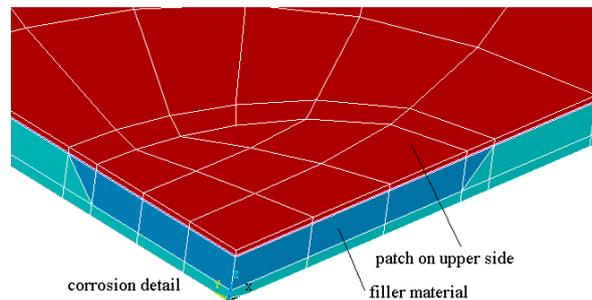


Fig. 7. Detail of repaired corrosion

3. OTHER FEATURES OF THE SOFTWARE

3.1. Calculation of the stress intensity factor

Consider the crack of figure 8.

The stress intensity factors K_I and K_{II} may be obtained by the difference of the displacements between the points B and B' as well as C and C' on the upper or the lower lip of the crack according to the following equations which are valid for a plane stress problem:

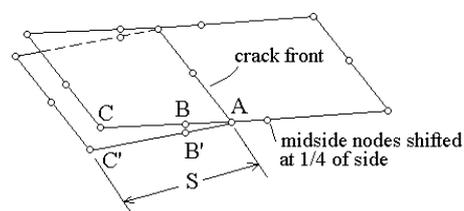


Fig. 8. A typical crack

$$K_I = \sqrt{\pi} \frac{E[8(u_y^B - u_y^{B'}) - (u_y^C - u_y^{C'})]}{6\sqrt{2S}} \quad (8)$$

$$K_{II} = \sqrt{\pi} \frac{E[8(u_x^B - u_x^{B'}) - (u_x^C - u_x^{C'})]}{6\sqrt{2S}}$$

where: E is the cracked plate young modulus
 u_x, u_y are the nodal displacements in the xy plane
 S is the distance between nodes A and C (or C')

3.2. User friendly interface

One of the main advantages of the software described herein is the user friendly interface. The user is asked to provide the input data such as problem configuration, material properties, geometry, loading through a series of dialog forms. As soon as all the input data is provided to the program, the user can create the finite element model and solve it. The input file of the model as well as the FE solution are stored in the hard disk. By default every time a model is created, an input file for the "Ansys" finite element software is also created and stored in the hard disk. In figure 9 the reader may take a look of what the interface looks like.

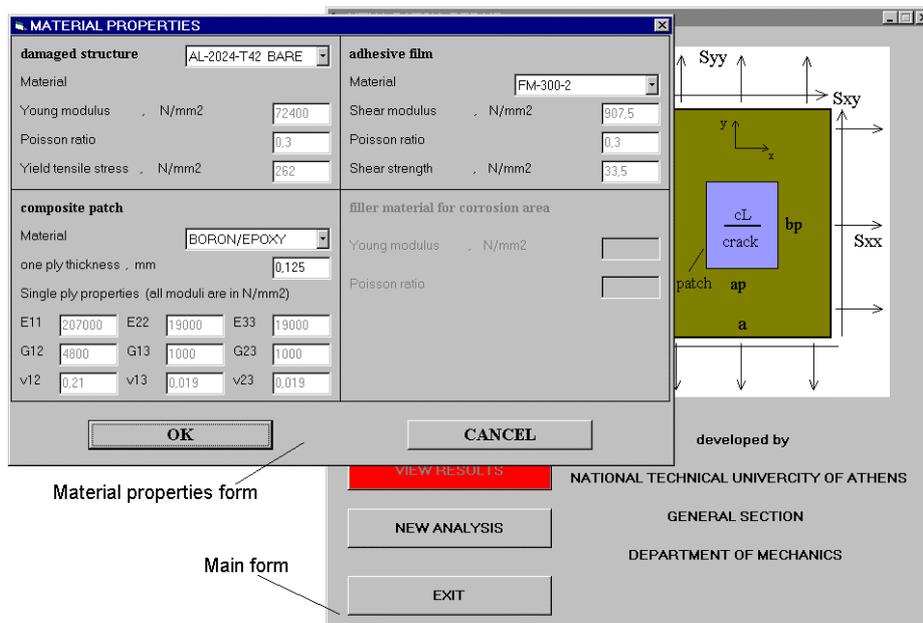


Fig. 9. Software environment

4. NUMERICAL IMPLEMENTATION

In order to demonstrate the accuracy of the software several test cases were run and the results are presented herein. For crack cases the Stress Intensity Factor K_I will be used

as an accuracy criterion while for corrosion cases the Maximum Von Mises Stress will be used as an accuracy criterion. Wherever applicable, the results are compared with exact analytic solutions. In cases where no analytic solution exists, the results are compared with the results produced by the Ansys finite element software.

Unrepaired structure

1. Central crack in a finite rectangular sheet, $a/b = 1$, uniform, uniaxial tensile stress perpendicular to crack

ro/a	K_I/K_0 exact	K_I/K_0 ntua	K_I/K_0 ansys	Error ntua %	Error ansys %
0.02	1.010	0.982	1.014	-2.85	0.19
0.1	1.025	0.995	1.027	-1.80	1.40
0.2	1.055	1.035	1.073	-1.89	1.70
0.3	1.123	1.101	1.142	-1.95	1.69
0.4	1.216	1.192	1.236	-1.97	1.64
0.5	1.334	1.306	1.355	-2.09	1.57
0.6	1.481	1.449	1.503	-2.16	1.48
0.7	1.680	1.636	1.699	-2.61	1.13
0.8	2.016	1.894	1.952	-6.05	-3.17

2. Edge crack in a finite rectangular sheet, $a/b = 1$, uniform, uniaxial tensile stress perpendicular to crack

ro/a	K_I/K_0 exact	K_I/K_0 ntua	K_I/K_0 ansys	Error ntua %	Error ansys %
0.05	1.172	1.128	1.168	-3.72	-0.34
0.10	1.246	1.204	1.247	-3.37	0.08
0.15	1.343	1.314	1.363	-2.16	1.49
0.20	1.470	1.454	1.508	-1.09	2.58
0.25	1.625	1.618	1.679	-0.43	3.32
0.30	1.806	1.806	1.873	0.00	3.71
0.35	2.018	2.020	2.095	0.10	3.81
0.40	2.271	2.268	2.352	-0.13	3.57
0.45	2.582	2.557	2.658	-0.97	2.94

3. Central hole with radial crack in a finite rectangular sheet, $a/b = 1$, uniform, uniaxial tensile stress perpendicular to crack, $a, b \gg cL, R$

cL/R	K_I/K_0 ntua	K_I/K_0 ansys	K_I/K_0 exact	Error ntua %	Error ansys %
0.1	2,764	2,822	2,786	-0,79	1,29
0.2	2,371	2,431	2,380	-0,38	2,14
0.3	2,068	2,107	2,098	-1,43	0,43
0.4	1,868	1,903	1,885	-0,90	0,95
0.5	1,732	1,796	1,728	0,23	3,93
0.6	1,614	1,684	1,605	0,56	4,92
0.8	1,448	1,517	1,428	1,40	6,23
1.0	1,323	1,383	1,306	1,30	5,89
1.5	1,156	1,204	1,128	2,48	6,70

4. Corrosion of elliptic shape in a finite rectangular sheet, uniform, uniaxial tensile stress perpendicular to corrosion x axis, corrosion depth is half the sheet's thickness (results are compared with ansys solution)

a/b (see paragraph 2.3)	Ntua Stress Concentration Factor	Ansys Stress Concentration Factor	Ntua error %
0,40	4,062	3,719	9,24
0,80	2,763	2,613	5,75
1,00	2,425	2,328	4,17
1,25	2,142	2,084	2,79
2,50	1,586	1,575	0,68

Repaired structure

5. Central crack in a finite sheet, $a/b = 1$, double orthogonal patch, uniform, uniaxial tensile stress perpendicular to crack, (no analytic solution exists, results are compared with ansys solution)

ro/a	ntua K_I/K_0 (middle)	ntua K_I/K_0 (top)	ansys K_I/K_0 (middle)	ansys K_I/K_0 (top)	Error ntua (middle) %	Error ntua (top) %
0.1	0,159	0,154	0,167	0,162	-4,91	-4,44
0.2	0,107	0,107	0,111	0,112	-4,01	-4,16
0.3	0,082	0,081	0,086	0,085	-4,20	-4,70
0.4	0,069	0,069	0,072	0,072	-4,08	-4,28

6. Edge crack in a finite sheet, $a/b = 1$, double orthogonal patch, uniform, uniaxial tensile stress perpendicular to crack, (no analytic solution exists, results are compared with ansys solution)

ro/a	ntua K_I/K_0 (middle)	ntua K_I/K_0 (top)	ansys K_I/K_0 (middle)	ansys K_I/K_0 (top)	Error ntua (middle) %	Error ntua (top) %
0.1	0,138	0,134	0,146	0,142	-5,65	-5,36
0.2	0,091	0,092	0,096	0,096	-4,41	-4,57
0.3	0,073	0,073	0,076	0,076	-4,30	-4,80
0.4	0,064	0,064	0,067	0,067	-4,02	-4,22

7. Central hole with radial crack in a finite rectangular sheet, $a/b = 1$, uniform, uniaxial tensile stress perpendicular to crack, $a, b \gg cL, R$, double elliptical patch (no analytic solution exists, results are compared with ansys solution)

cL/R	ntua K_I/K_0 (middle)	ntua K_I/K_0 (top)	ansys K_I/K_0 (middle)	ansys K_I/K_0 (top)	Error ntua (middle) %	Error ntua (top) %
0.3	0,252	0,255	0,229	0,259	10,33	-1,63
0.4	0,224	0,225	0,215	0,238	4,10	-5,77
0.5	0,192	0,196	0,183	0,218	4,75	-10,12
0.6	0,166	0,176	0,163	0,201	1,51	-12,24

In the case of a corroded sheet the accuracy criterion that is used is the maximum stress inside the metal sheet. As it can be seen from table 4 there is some error in the calculation of the nodal stresses compared with the ansys solution. This may be due to the fact that ansys may perform a smoothening of the nodal stresses in comparison to the ntu software where the stress at each node is the mean value of the surrounding element stresses. As far as the calculation of the nodal displacements is concerned, the error norm of the displacements takes values at the order of $10e-6$ which means that the displacement vector is completely identical for both softwares.

5. CONCLUSIONS

The software presented herein, is a very fast and very easy to use tool for fast analysis of repaired aircraft structures. Because of the automatic mesh creation it is also ideal for parametric analyses. The engineer can –very quickly- create different models with varying repair configurations (number of plies, stacking sequence, patch dimensions) and choose the optimum one. The accuracy of the software is within reasonable limits.

REFERENCES

1. George Tsamasphyros, (1987), "Singular element construction using a mapping technique", International Journal for Numerical Methods in Engineering, Vol. 24, 1305-1316
2. Y.C.HSU, (August 1975), "The infinite sheet with cracked cylindrical hole under inclined tension or in-plane shear", International Journal of Fracture, Vol. 11, 571-581, No. 4

KOMPJUTERSKI SOFTVER ZA ANALIZU NAPONA POPRAVLJANIH VAZDUHOPLOVNIH STRUKTURA METODOM POPRAVKE MEŠOVITIM KRPLJENJEM

George Tsamasphyros, Zaira Marioli-Riga, Chris Vrettos

Prikazan je kompjuterski program za rešavanje problema mehanike loma. Softver uključuje rutinu okastog generatora za stvaranje geometrije, i rutinu konačnog elementa za izračunavanje vrednosti koje su ispod interesa, kao što su faktori intenziteta napona, naponi itd. Program je u stanju da se bavi analizom napona naprslih ili korodiranih metalnih vazduhoplovnih struktura. Jedna od najmodernijih metoda popravke takvih vrsta oštećenja je metoda mešovite popravke krpljenjem. Osnova metode je postavljanje zakrpe načinjene od mešovitog materijala (npr. ugljenik, bor) preko oštećene oblasti. Ako je vrsta oštećenja prslina, popravka se sastoji u smanjenju faktora intenziteta napona u metalnoj strukturi. U slučaju korozije, popravka se sastoji u smanjenju maksimalnog Von Mises napona u metalnoj strukturi. Ovde opisani program – uz pomoć njegovog user friendly okruženja- orudje je za brzu analizu gore opisanih slučajeva.