Zero stiffness method for fail-safe analysis

Raul CORMOS*,[1], Catalin Andrei NEAGOE[2,3]

*Corresponding author
[1]INCAS – National Institute for Aerospace Research "Elie Carafoli",
B-dul Iuliu Maniu 220, Bucharest 061126, Romania,
cormosr@yahoo.com*
[2]Department of Strength of Materials, University POLITEHNICA of Bucharest,
Splaiul Independentei 313, Bucharest 060042, Romania,
catalin.neagoe@upb.ro
[3]The Institute of Solid Mechanics of the Romanian Academy,
Constantin Mille 15, Bucharest 030167, Romania

DOI: 10.13111/2066-8201.2023.15.2.3

Received: 23 March 2023/ Accepted: 07 April 2023/ Published: June 2023

Abstract: *One of the major engineering tasks is to evaluate the structural behavior when one of the components has failed. In such cases it is necessary to carry out fail-safe analysis to evaluate if the structure can be used safely in loading conditions. Thus, fail-safe analysis is a vital and important task to properly validate the mechanical structure. The implementation of the fail-safe analysis using the finite element method is usually done by eliminating the given component from the finite element model and carrying out the given analysis. But when due to finite element modeling issues such an approach cannot be carried out without causing singularities in the model, another approach should be used to perform the fail-safe analysis. One such method, presented in this paper, is the zero stiffness method, which applies near-zero stiffness to the structural component that is removed from the finite element model. The zero stiffness method is used by applying close to zero values to the material and element properties, and thus reducing the load that is in the given structural component.*

Key Words: *Fail-safe analysis, zero stiffness, finite element, aerospace structures*

1. INTRODUCTION

It is very important to carry out a fail-safe analysis of primary high-performance structural components, such as those found in aerospace structures. Numerous studies have been realized on the use of fail-safe analysis in aerospace applications, and a general definition has been given. Fail-safe analysis means that the structure has been evaluated to ensure that a catastrophic failure or excessive structural deformation, which could adversely affect the flight characteristics of an aircraft, is not likely after a fatigue failure or obvious partial failure of a single major structural element [1].

The fail-safe analysis method has been investigated in numerous research works and it has been concluded that structural components to fail are influenced by many factors such as loading case scenarios and the complexity of the structure.

Herein, the more complex a structure is, the less harmful impact the failure of a structural component will have on the operation of that given structure.

Local failure of the structure can cause a redistribution of the load pattern, weakening the local structure, but not necessarily failure, which means that the structure can take the load without complete structural failure [2-5]. Economic reasons are also very important to take into consideration. It is vital to predict the amount of damage a structural component can withstand and when it should be considered replaceable, or situations where just a local repair is sufficient to reduce damage.

Computer algorithms were built to simulate the effect of local failure and to give a better understanding of local failure processes. In recent decades, reduced and zero stiffness characteristics were introduced in finite element solvers with the purpose of structural optimization. On that account, material stiffness is reduced taking into consideration the load pattern, and the mechanical structure is optimized by reducing the local structural stiffness of the part via material mechanical properties, more specifically the Young's modulus and Poisson's ratio. The structures optimized in this way are then tested in the given loading conditions and the optimum form is determined by the critical load case scenario.

However, we must not make a confusion between structural optimization and fail-safe analysis. By structural optimization, a given mechanical part is computed in such ways that an optimum shape is determined, usually by means of economic reasons but also to reduced weight, which is especially important in the field of aerospace engineering or material cost estimation. But that does not mean that a local failure of the mechanical part cannot cause total or partial failure, rendering, thereby, the results of the optimization with little or no impact in the safe exploitation of the structure for the given critical loading scenario.

Fail-safe analysis, on the other hand, does not mean that a structural component or structural system is economically optimized. It means that the mechanical system has been designed in such a way that a local or partial failure of the structure does not affect the safety of the whole structure, but that is economically conservative since it does not lead to the loss of the whole mechanical structure. Thereby, a structure computed in a fail-safe analysis is not necessarily the optimum solution since it must reduce the chance of catastrophic failure by increasing the complexity of the structure, by adding more parts to it. A fail-safe analysis can be further complicated by considering additional computational parameters like stress concentration factors, which make the computation more expensive, in both time and resources.

2. PURPOSE OF THE ARTICLE

The main purpose of the article is to present an alternative way of applying the fail-safe analysis using the zero stiffness method for a fail-safe analysis application. The problem presented in this paper is based on a mechanical structure which was designed after a real world application, but only the concept of the method application is presented.

The mechanical system in question is made out of two plates held together by six fasteners. A load is applied to the structure and it is checked if, by taking out the middle two fasteners, by elimination or by applying zero stiffness to them, the results are comparable in terms of:

-shear forces;

-local structural stresses around the holes using fracture mechanics principles.

The obtained numerical results are then validated by classical hand calculation methods from Strength of materials and Machine elements handbooks.

A detail of the application of the method is presented in this article by showing the NASTRAN card declaration for zero stiffness using the PBUSH card. The following application is compared with a finite element model simulation using the Nastran finite element solver. The finite element model was built using Altair HyperMesh and the results were processed using MSC Patran.

3. MODEL DESCRIPTION

The employed geometric model is made up of two bodies: a plate and a plate wall support. The two plates are linked together using 6 steel fasteners, as presented in Fig. 1.

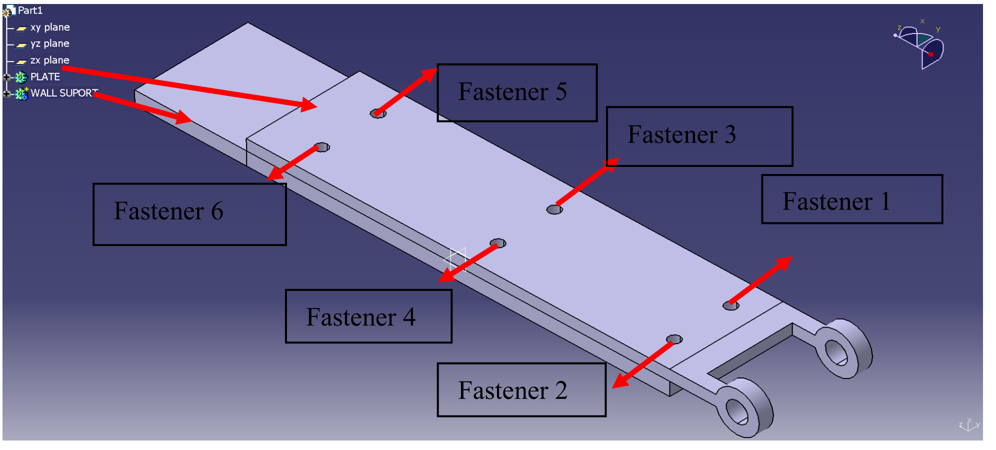

Fig. 1 – CAD model

The two plates are made of standard aluminum and the fasteners are made of standard steel. The wall support has all translational degrees of freedom locked. The geometry of the two parts is presented in Fig. 2.

To be noted that the wall support has no particular purpose in the model besides serving as support.

The wall part has a thickness of 10 mm, the top part 8 mm and the fasteners have a diameter of 8 mm. A higher thickness is considered for the wall so as to increase its stiffness.

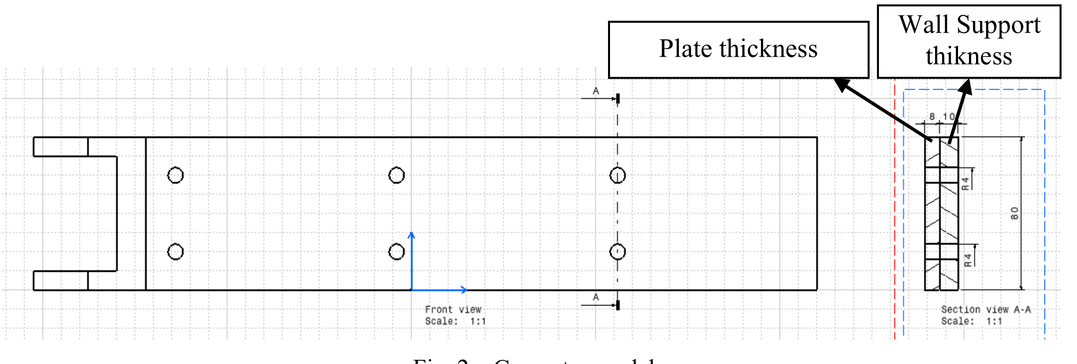

Fig. 2 – Geometry model

The top plate is modeled using hexa8 elements, where three rows of elements are used to capture the bending behavior.

The support plate is made with quad4 elements. The two parts are joined together with elastic CBUSH elements linked with the wall support by RBE3 elements. A representation of the finite element model is shown in Fig. 3.

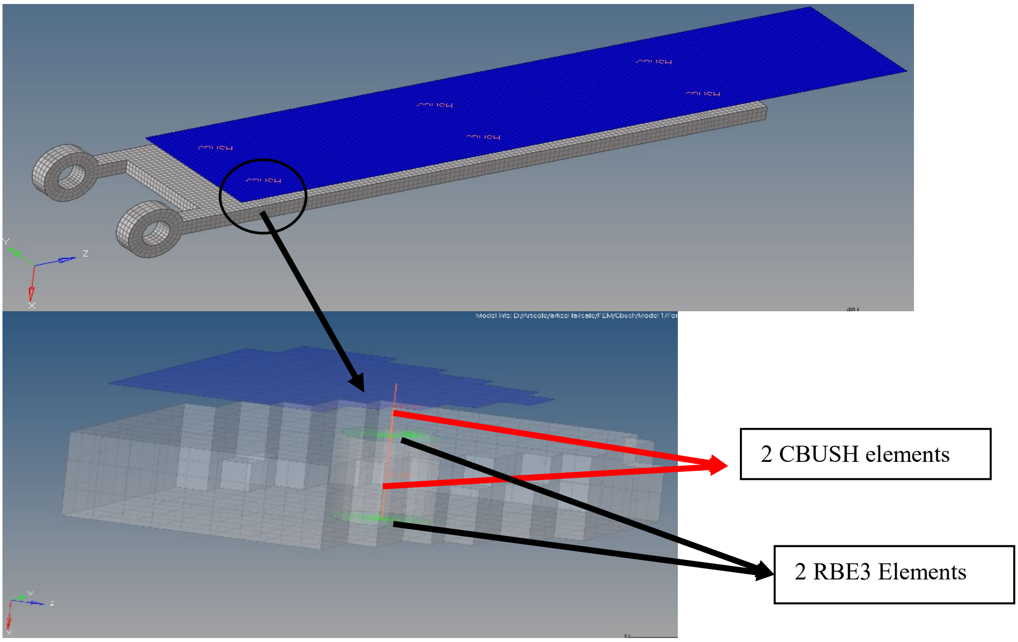

Fig. 3 – Finite element model

The load conditions were considered by applying a 68000 N load on the CBEAM elements which are simulating the bolts. The loading condition is presented in Fig. 4.

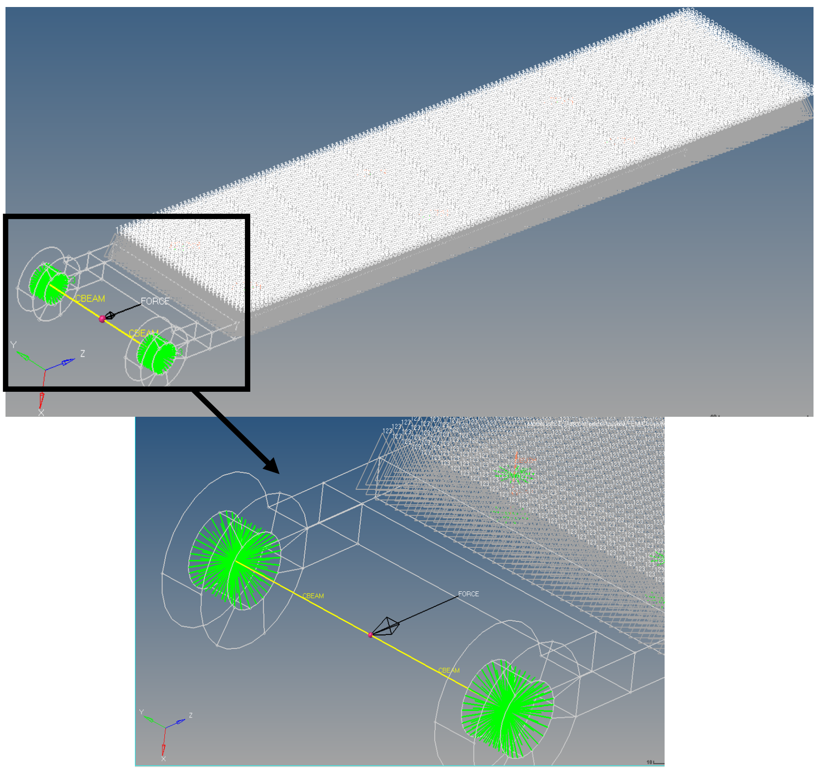

Fig. 4 – Finite element loading conditions

4. MATERIAL PROPERTIES

There are two materials used in the analysis: steel and aluminum. The material data used [6] are presented in Table 1.

Table 1 – Material characteristics data

Material name	**Young's modulus [MPa]**	**Poisson's ratio**
Steel	200000	0.29
Aluminum	72000	0.33

The Plate and Wall Support have the aluminum material applied, while the steel material properties are used to compute the translational stiffness components of the fasteners.

5. IMPLEMENTATION OF THE ZERO STIFFNESS METHOD

To validate the usage of the zero stiffness method, three finite element models were made which all have different properties applied on the two middle stiffness CBUSH elements. The ID and locations of the beams are presented in Fig. 5.

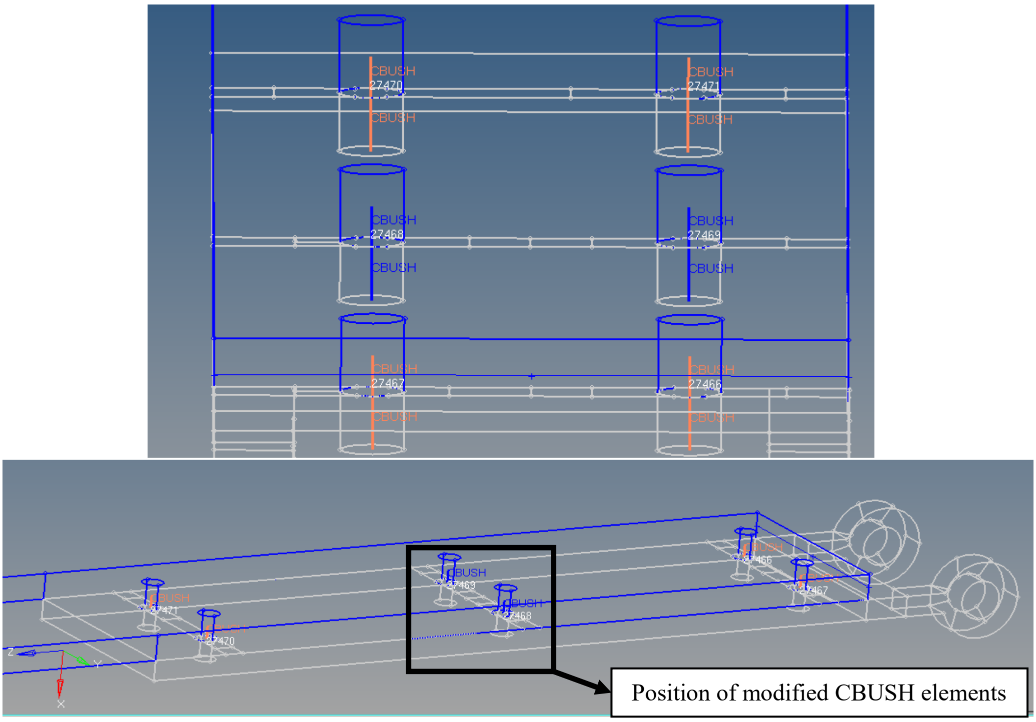

Fig. 5 – Result extraction CBUSH locations

Using this description, the shear loading on the beams takes place on the upper fasteners, between the two parts.

Thus, three finite element models are made. In the first model, all fastener elements have steel stiffness properties applied. In the second, near zero stiffness is applied on middle four CBUSH elements. In the third model, the middle four elements are deleted and the model is examined.

The zero stiffness method is implemented by applying near zero stiffness on the four beams, by making the stiffness of the PBUSH elements equal to 0.001 N/mm.

6. ANALYTICAL CALCULATION OF STIFFNESS

For implementing the zero stiffness method, the CBUSH element was used due to the fact that it allows the controlled introduction of the stiffness in the model. As a short description, CBUSH is a Nastran spring element allowing manual definition of stiffness components, namely the three translational stiffnesses, K1, K2, K3 (on X, Y, Z directions) and the three rotational ones K4, K5, K6.

In the PBUSH entry, a local coordinate system is defined in order to specify CBUSH stiffness orientation, as presented in Fig. 6. For the present model, only the translational stiffnesses were considered [7], [8].

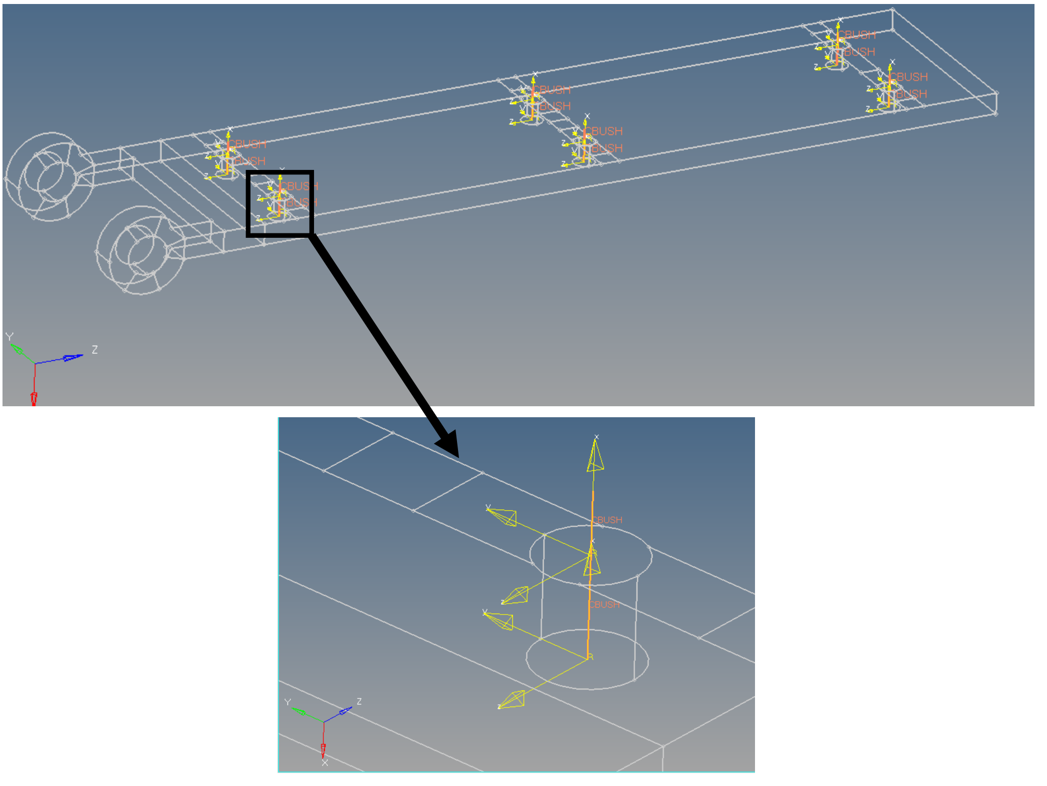

Fig. 6 – CBUSH element and local axes system

The stiffness of CBUSH element is computed using the Huth formulation. This formulation is used in industrial applications and it is estimated that it gives a small margin of error. Other methods are also presented for estimation of stiffnesses. A short summary of these methods can be found in refence [9].

The Huth method calculates the stiffness using the following analytical formulas:

For the longitudinal stiffness K1:

$$K_1 = \frac{E \cdot S}{L} \tag{1}$$

where: K_1 – longitudinal stiffness [N/mm]; E – Young's modulus [N/mm^2]; S – transversal area [mm^2]; L – element length [mm].

According to the Huth formulation, the translational stiffness for direction two and three are equal to one another:

$$K_2 = K_3 = \frac{1}{C} \tag{2}$$

where C is the elasticity of the fastener [mm/N].
Using Huth's approach, the elasticity of the fastener is computed with the following formula:

$$C = \left(\frac{t_1 + t_2}{2 \cdot d}\right)^a \cdot \frac{b}{n} \cdot \left(\frac{1}{t_1 \cdot E_1} + \frac{1}{n \cdot t_2 \cdot E_2} + \frac{1}{2 \cdot t_1 \cdot E_2} + \frac{1}{2 \cdot n \cdot t_2 \cdot E_3}\right) \tag{3}$$

where: t_1 – thickness of plate 1 [mm]; t_2 – thickness of plate 2 [mm]; E_1 – Young's modulus of plate no. 1 [N/mm2]; E_2 – Young's modulus of plate no. 2 [N/mm2]; E_3 – Young's modulus of fastener [N/mm2]; d – fastener diameter [mm]; n – number of shear sections; a – experimental constant, the value for bolted joint is $2/3$; b – experimental constant, for metallic material it is equal to 3.
Using the material data and the formulas presented above, the stiffness values can be computed. The obtained results are presented in Table 2.

Table 2 – Stiffness calculation for CBUSH element

Part	**Material**	**Diameter**	**t_1 [mm]**	**t_2 [mm]**	**L [mm]**	**E_3 [MPA]**	**E_1=E_2 [MPA]**	**Area [mm^2]**
Fastener	Steel	8	8	10	80	200000	72000	50.26

Element length	**K_1 [N/mm]**	**n**	**a**	**b**	**C [mm/N]**	**K_2=K_3 [N/mm]**
5	2010619	1	0.67	3	1.37744E-05	72598.53

For the second computational case, the stiffness value was considered to be 0.001 N/mm. A description of the PBUSH Nastran card is presented in Fig. 7 and 8. Fig. 9 displays the PBUSH application on the CBUSH element.

```
$$  PBUSH Data
$
$HMNAME PROP                  1"CBush" 2
$HWCOLOR PROP                 1       11
PBUSH          1       K2010619.72598.5372598.53
```

Fig. 7 – PBUSH Card for First and third case analysis

```
$$  PBUSH Data
$
$HMNAME PROP                  1"CBush" 2
$HWCOLOR PROP                 1       11
PBUSH          1       K2010619.72598.5372598.53
$
$HMNAME PROP                  5"CBush_0" 2
$HWCOLOR PROP                 5       11
PBUSH          5       K0.001   0.001   0.001
```

Fig. 8 – PBUSH Card for second case analysis

```
$$  CBUSH Elements
$$
CBUSH      27460    1   28019   28018                                   11
CBUSH      27461    1   28021   28020                                    9
CBUSH      27462    5   28022   28024                                    8
CBUSH      27463    5   28025   28023                                    5
CBUSH      27464    1   28028   28027                                    2
CBUSH      27465    1   28029   28026                                    1
CBUSH      27466    1   28018   23183                                   12
CBUSH      27467    1   28020   23188                                   10
CBUSH      27468    5   28023   23184                                    7
CBUSH      27469    5   28024   23185                                    6
CBUSH      27470    1   28026   23187                                    4
CBUSH      27471    1   28027   23186                                    3
```

Fig. 9 – CBUSH Card for second case analysis

For the purpose of this article, the rotational stiffness is not considered because it has relatively minor influence on results.

7. ANALYTICAL COMPUTATION OF LOADS

To validate the results, the classical computation used for mechanical components is used. Thereby, for both cases the following loads will be applied on each fastener [10], [11].
Shear load for case 1:

$$F_{1i} = \frac{68000}{6} = 11333 \text{ N} \tag{4}$$

Shear load for case 2 and 3:

$$F_{23i} = \frac{68000}{4} = 17000 \text{ N} \tag{5}$$

8. POSTPROCESSING OF THE RESULTS

After completing the finite element simulation on the three cases, the following results were obtained.

The areas of interest are the beams on the shear portion of the model [12]. For the first model with all six fasteners, the following results were gathered:

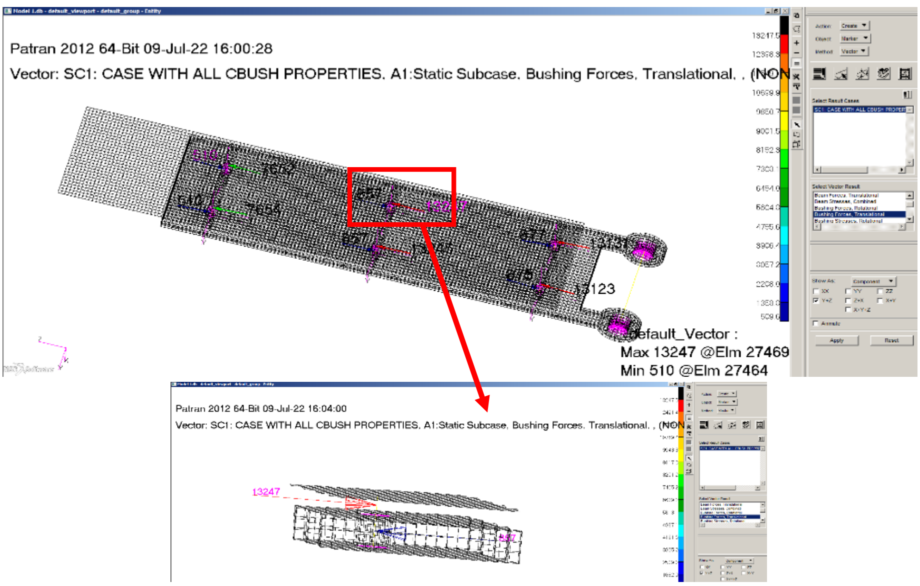

Fig. 10 – First model run results

The computed error is:

$$Error_{Model\_1} = \left|\frac{11333 - 13247}{11333}\right| \cdot 100 = 16.88\ \% \tag{6}$$

For the second model with near zero stiffness on the middle fasteners, the results are displayed in Fig. 11.

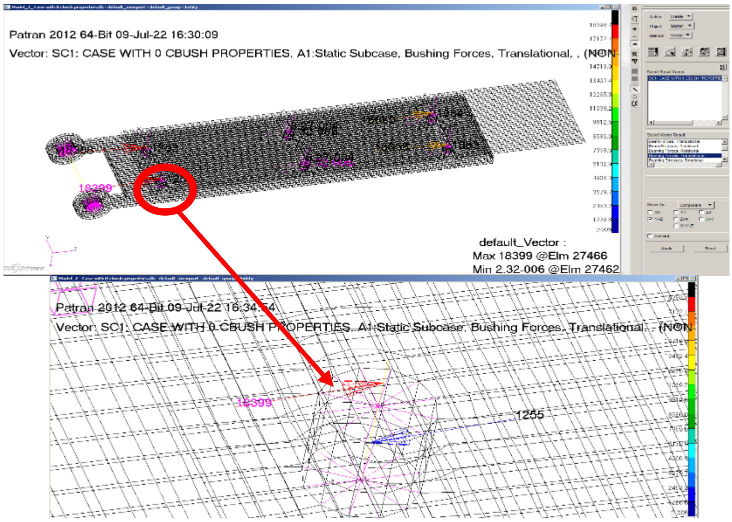

Fig. 11 – Second model run results

The computed error in this case is:

$$Error_{Model\_2} = \left| \frac{17000 - 18398.7}{17000} \right| \cdot 100 = 8.23\ \% \tag{7}$$

For third model with near zero stiffness on middle fasteners.

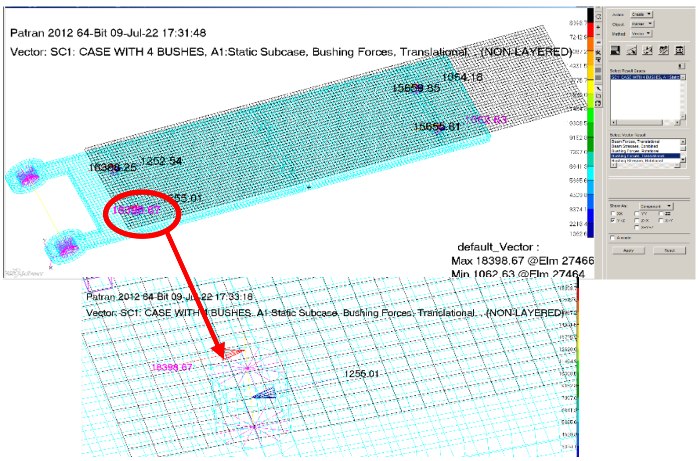

Fig. 12 – Third model run results

In the third case, the computed error is equal to the one in model 2.

9. VALIDATION OF THE METHOD USING FRACTURE MECHANICS PRINCIPLES

Stress around a hole in a plate can be approximated as 3 times the nominal stress, $k_t = 3$. The local nominal stress is composed of the tension stress in an unnotched specimen summed with the bearing stress acting on the hole [13], [14]. Thus, the nominal stress can be computed as:

$$\sigma_{nom} = \sigma_t + \sigma_b \tag{8}$$

where: σ_{nom} – nominal stress [MPa]; σ_t – tension stress in unnotched specimen [MPa]; σ_b – bearing stress [MPa].

The finite element stress around the hole is computed as a product between the nominal stress and the stress concentrator coefficient. Such a computation is an engineering hypothesis. The scientific literature in fracture mechanics offers more detailed methods for approximating the value of k_t. One of the formulas proposed is:

$$k_t = F \cdot \sqrt{r \cdot \pi} \tag{9}$$

where: F- computational coefficient, usual value between 1-1.12; r – hole radius [mm].

Eq. (8) can be written, expanding the terms:

$$\sigma_{nom} = \frac{T}{S} + F_{sh} \cdot d \cdot t \tag{10}$$

where: T – tension force in plate [N]; S – transversal area [mm^2]; F_{sh} – shear force [N]; d – hole diameter [mm]; t – plate thickness [mm].

The finite element stress around the hole is computed using the formula:

$$\sigma_{fem\_hole} = k_t \cdot \sigma_{nom} = 3 \cdot \sigma_{nom} \tag{11}$$

For the first case of computation, the analytical calculation of the stress around the hole is:

$$\sigma_{fem\_hole\_1} = \left(\frac{68000}{8 \cdot 80} + \frac{11333}{8 \cdot 8}\right) \cdot 3 = 850 \text{ MPa} \tag{12}$$

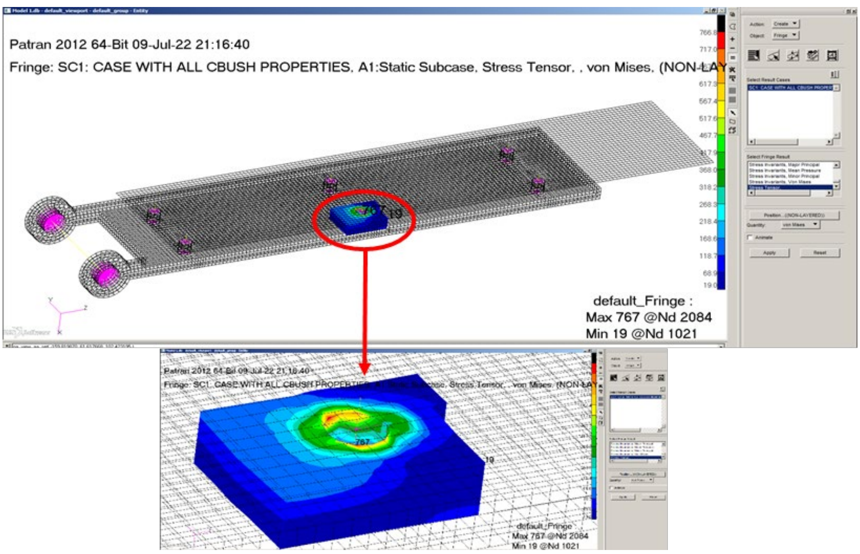

Fig. 13 – Stress around hole – Case 1

The computed error is:

$$Error_{Model\_1} = \left|\frac{850 - 767}{850}\right| \cdot 100 = 9.7\ \% \tag{13}$$

For the second and third cases where there are four fasteners, the stress value is:

$$\sigma_{fem\_hole\_23} = \left(\frac{68000}{8 \cdot 80} + \frac{17000}{8 \cdot 8}\right) \cdot 3 = 1115 \text{ MPa} \tag{14}$$

The stress around the hole for the second case is presented in Figure 14.

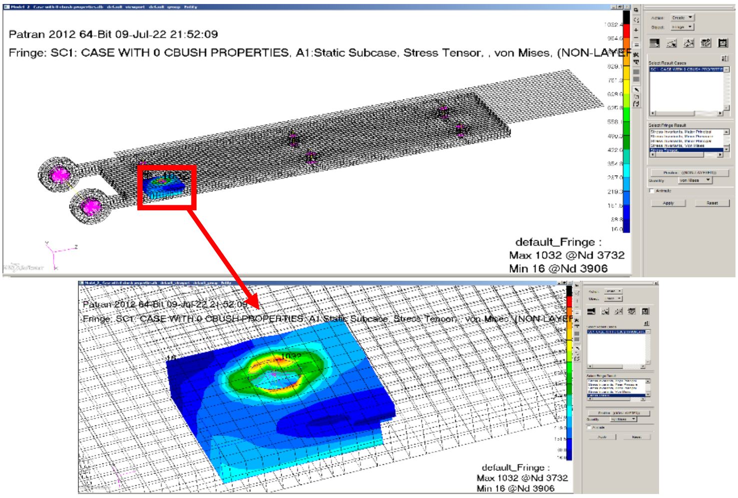

Fig. 14 – Stress around hole – Case 2

$$Error_{Model\_2} = \left|\frac{1115 - 1032}{1115}\right| \cdot 100 = 7.4\ \% \tag{15}$$

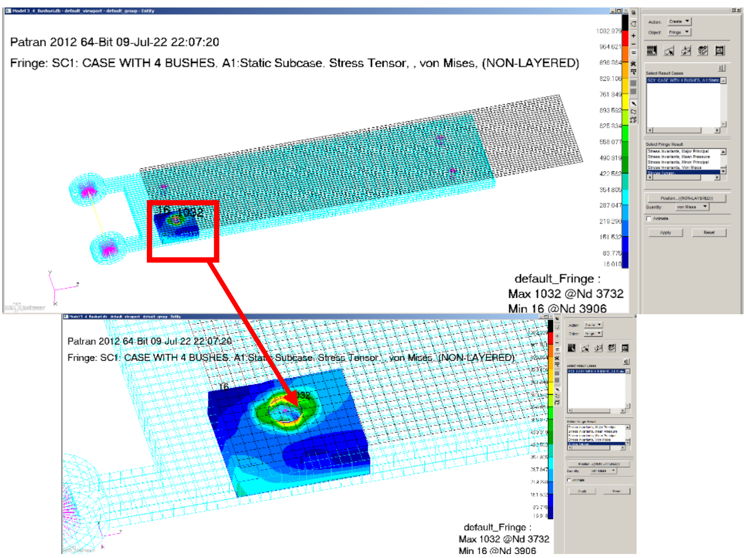

Fig. 15 – Stress around hole – Case 3

The error computed in case 3 is equal to the error in case 2.

10. CONCLUSIONS AND REMARKS

The current study has presented in a comparative manner the application of the zero stiffness method in the fail-safe analysis of mechanically-fastened joints in aerospace structures.

After analyzing the results obtained, the following conclusions can be drawn:

1. For the linear static analysis, the zero stiffness method can be very well used for fail-safe analysis if/in case the finite element model proves to be unstable.

2. The results obtained from the usage of zero stiffness method and classical fail-safe analysis for fastener design yields identic results.
3. The implementation of the zero stiffness method is very accessible in the case of the MSC Nastran solver. It was made by reducing to almost zero the values of the stiffnesses, by using CBUSH elements.
4. *bdf* files allow for quick and easy evaluation and or modification of the solver data, thus such changes can be made even without the usage of a graphical preprocessor finite element program.
5. Using the Huth formula allows the analytical declaration of stiffness matrix.
6. Two methods were used to validate the results obtained, one using the shear force and the other based on the calculation of stress around holes.
7. The accuracy of the validation lies between 8.23-16% for the shear force method and 7.4-9.4% for the second method, thereby the best results were obtained for the second method based on fracture mechanics principles.
8. Using the presented engineering hypotheses, the best results were obtained for the second method.

ACKNOWLEDGEMENTS

This work was supported by the POCU/993/6/13-153178 grant, co-financed by the European Social Fund through the Sectorial Operational Programme Human Capital 2014-2020.

REFERENCES

[1] W. G. Heath, Dip.Ae.(Hull), A.F.R.Ae.S., Structural Design Part II—Fail-Safe Aspects, *Aircraft Engineering and Aerospace Technology*, Vol **37**, issue 3, ISSN: 0002-2667, 1965.

[2] J. K. Lüdeker, B. Kriegesmann, Fail-safe optimization of beam structures, *Journal of Computational Design and Engineering*, Volume **6**, Issue 3, Pages 260–268, July 2019, https://doi.org/10.1016/j.jcde.2019.01.004

[3] K.-J. Bathe, *Finite Element Procedures*, ISBN-10 : 097900490X, ISBN-13: 978-0979004902, 2007.

[4] M. Stolpe, Fail-safe truss topology optimization, *Structural and Multidisciplinary Optimization*, https://doi.org/10.1007/s00158-019-02295-7, 2019.

[5] N. F. Harpur, Fail-Safe Structural Design, *The Aeronautical Journal*, Volume **62** , Issue 569, pp. 363 - 376, May 1958, https://doi.org/10.1017/S0368393100068826I958.

[6] * * * Mat Web, https://search.library.wisc.edu/database/UWI12433.

[7] * * * Altair Hypermesh help file, version 2017.

[8] * * *Msc Nastran Quick reference guide, version 2018.

[9] R. Omar, M. N. Abdul Rani, M. A. Yunus, D. P. Starbuck, W. I. I. Wan Iskandar Mirza, and M. S. Mohd Zin, *Different flexibility formulae for finite element modelling and analysis of the dynamic behavior of a structure with bolted joints,* AIP Conference Proceedings 2059, 020004 (201,; https://doi.org/10.1063/1.5085947 2059, 020004

[10] Gh. Manea, O. Bosianu, A. Chisu, L. Saveanu, *Machine design exercise book volume 1 and 2*, Tehnical publisher, Bucharest 1958.

[11] I. Draghici, A. Jula, E. Chisu, and others, *Mashine design exercice book*, Didactic and pedagogica publisher, Bucharest 1980.

[12] * * * Msc Patran post procesing guide, version 2012.

[13] N. E. Downling, *Mechanical Behavior of Materials*', Pearson edition, ISBN 10: 0-273-76455-1, ISBN 13: 978-0-273-76455-7, 2013.

[14] W. D. Pilkey, Z. Bi, D. F. Pilkey, *Peterson's stress concentration factors 4th edition*, Wiley, ISBN: 978-1-119-53252-1, 2020.