

ABOUT THE CONVERGENCE OF THE FINITE ELEMENTS ANALYSIS

Dorian NEDELICU^{*}, Calin-Octavian MICLOȘINĂ^{}, Ioan PĂDUREAN^{***}**

^{*} “Eftimie Murgu” University Reșița, P-ta Traian Vuia, no. 1-4, d.nedelcu@uem.ro

^{**} “Eftimie Murgu” University Reșița, P-ta Traian Vuia, no. 1-4, c.miclosina@uem.ro

^{***} “Politehnica” University Timișoara, P-ta Victoriei no. 2, padurean58@yahoo.com

Abstract. The finite element analysis (FEA) is a numerical method for simulation of the real processes, which can generate accurate results instead of an exact solution. The present study will focus on the convergence of the finite elements computation, with strength calculation exemplification worked in Cosmos Design Star software [1], [2]. Specific conclusions about convergence and mesh technique were obtained [3], [4].

Keywords: convergence, finite element analysis, FEA, mesh.

1. Introduction

The study was made on both 2D and 3D geometry. The 2D geometry is presented in figure 1 [5]. The plate with 20 mm thickness, 200 mm length, 100 mm width and a central hole is fixed on left edge and loaded with pressure $p=5$ MPa on right edge. The plane geometry was converted to surface and shell mesh type was used with parabolic elements, which have 6 nodes: 3 corner nodes and 3 mid-side nodes. Shell elements are 2D elements capable of resisting membrane and bending loads. For structural studies, each node in shell elements has six degrees of freedom; three translations and three rotations.

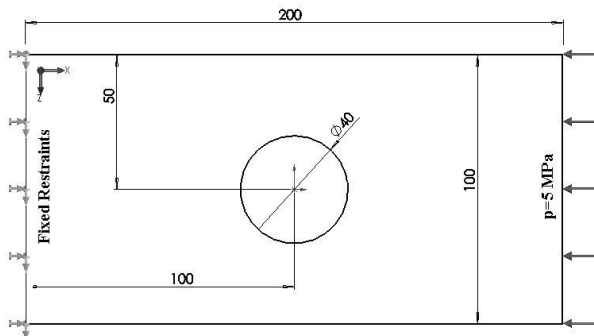


Figure 1 The 2D geometry

The material selected from Cosmos library for the plate was *Alloy Steel* with Elastic modulus 2.1×10^{11} N/m² and Poisson's ratio 0.28.

The 3D geometry is presented in figure 3. The *Fixed* restraints was applied on 4 holes and the solid was loaded with directional force $F_x=20000$ N on cylindrical zone of the top hole. Each solid element will have 10 nodes: 4 corner nodes and one node at the middle of each edge (a total of six mid-side nodes).

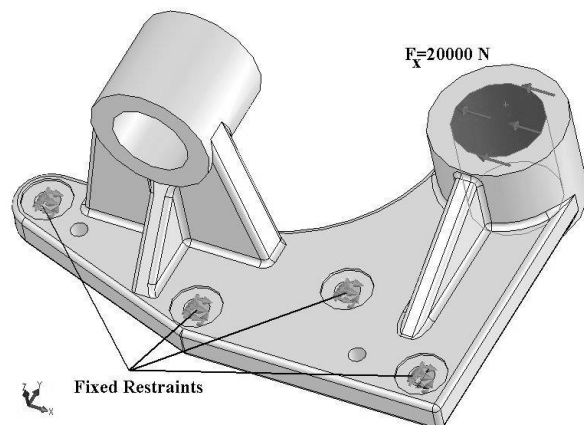


Figure 2 The 3D geometry

For structural problems, each node in a solid element has three degrees of freedom that represent the translations in three orthogonal directions. The material selected from Cosmos library for the 3D solid was *Cast Alloy Steel* with Elastic modulus $1.9003\text{e}+011 \text{ N/m}^2$ and Poisson's ratio 0.26.

For shell meshes the *Fixed* restraints will sets the translational and the rotational degrees of freedom to zero. For solid meshes, this restraint type sets all translational degrees of freedom to zero.

2. The mesh options

Meshing options are essential factors in determining the quality of the mesh and of the results. Results based on different preference settings should converge to each other if an adequately small element size is used.

The mesh is generated starting from the global average element size (GAES). The software suggests a default value based on the model volume and surface area.

It is possible to set the Mesh quality to *Draft* or *High*. A *draft* quality mesh does not have mid-side nodes and can be used for quick evaluation and in solid models where bending effects are small. *High* quality mesh is recommended in most cases, especially for models with curved geometry.

The *Automatic transition* (AT) option automatically applies mesh controls to small features, details, holes, and fillets. This option must be unchecked before meshing large models with many small features and details to avoid generating a very large number of elements unnecessarily.

The *Smooth surface* (SS) option will generate a slightly relocating of the boundary nodes to improve the mesh. It is recommended to check this option in most cases.

Local Mesh Control (LMC) can be used to specifying different element sizes at different regions in the model. A smaller element size in a region improves the accuracy of results in that region. It is possible to specify LMC at vertices, edges, faces, and components. The mesh control parameters are: *element size* (e) for the specified entities, *element growth ratio* (r), *number of layers* of elements (n). Assuming that the element size used for meshing an entity is (e), the average element size in layers radiating from the entity will be: e, $e*r$, $e*r^2$, $e*r^3$, ..., $e*r^n$. If the calculated average element size of a layer exceeds GEAS, the program uses GEAS instead. If the specified number of layers (n) is too small for a

smooth transition, the program adds more layers automatically. The mesh radiates from vertices to edges, from edges to faces, from faces to components, and from a component to connected components.

For a meshed model with a set of displacement restraints and loads, the linear static analysis program proceeds as follows:

- the program constructs and solves a system of linear simultaneous finite element equilibrium equations to calculate displacement components at each node.

- the program then uses the displacement results to calculate the strain components.

- the program uses the strain results and the stress-strain relationships to calculate the stresses.

Stress results are first calculated at special points, called Gaussian points or Quadrature points, located inside each element. These points are selected to give optimal numerical results. The program calculates stresses at the nodes of each element by extrapolating the results available at the Gaussian points.

After a successful run, nodal stress results at each node of every element are available in the database. Nodes common to two or more elements have multiple results. In general, these results are not identical because the finite element method is an approximate method. For example, if a node is common to three elements, there can be three slightly different values for every stress component at that node.

During viewing stress results, it is possible to select *element stresses* or *nodal stresses*. To calculate *element stresses*, the program averages the corresponding nodal stresses for each element. To calculate *nodal stresses*, the program averages the corresponding results from all elements sharing that node.

It is possible to use *symmetry* to model a portion of the model instead of the full model. The results on the un-modeled portions are deducted from the modeled portion. Taking advantage of symmetry can help to reduce the size of the problem and obtain more accurate results. Symmetry requires that geometry, restraints, loads, and material properties are symmetrical.

For *solid* models, every face that is coincident with a plane of symmetry should be prevented from moving in its normal direction. For *shell* models, symmetry restraints requires that faces coinciding with planes of symmetry should be prevented from moving in the normal direction and rotating about the other two orthogonal directions.

3. The 2D plate analyse

The analyse was made for *high* mesh, SS=ON and AT=OFF. The numerical results are presented in table 1 and graphical in figure 3.

Table 1 Numerical results for 2D plate

Case No.	Nodes number	Finite elements number	Medium mesh value (GAES)	Local Mesh Control (LMC) e/r/n	Von Mises Stress [MPa]	
					Element stress	Nodal stress
1	1208	558	8.2	-	13.3	17.08
2	2297	1087	6	-	14.87	17.49
3	4862	2340	4	-	15.65	18.19
4	19089	9363	2	-	17.15	18.71
5	75714	37494	1	-	17.92	18.81
6	154346	76654	0.7	-	18.19	18.19
7	20911	10333	8.2	5/1.5/5	17.62	18.78
8	24043	11899	8.2	5/1.5/10	17.62	18.78
9	28451	14103	8.2	5/1.5/20	17.62	18.78
10	21271	10505	6	5/1.5/5	17.62	18.78
11	24299	12019	6	5/1.5/10	17.61	18.78
12	28611	14175	6	5/1.5/20	17.61	18.78
13	22231	10969	4	5/1.5/5	17.62	18.78
14	25003	12355	4	5/1.5/10	17.61	18.78
15	28955	14331	4	5/1.5/20	17.61	18.78
16	630	281	8.2	-	13.3	17.09
17	1134	521	6	-	14.87	17.42
18	2306	1087	4.1	-	15.36	17.99
19	4361	2092	3	-	16.47	18.38
20	9639	4688	2	-	17.11	18.71
21	38002	18739	1	-	17.93	18.77
22	76927	38088	0.7	-	18.18	18.82
23	150987	74970	0.5	-	18.34	18.8

The case no. 1...6 correspond to uniform meshing, figure 4, with GAES decreasing. The case no. 7...15 correspond to some GAES values with local LMC applied, figure 5. The case no. 16...23 correspond to uniform meshing with GAES decreasing and symmetry conditions applied, figure 6.

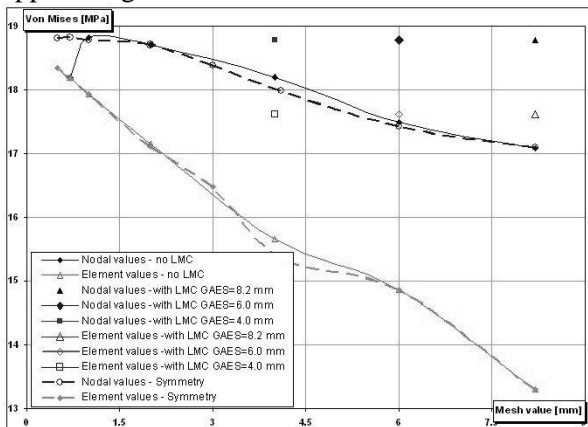


Figure 3 The VonMises distribution for 2D plate

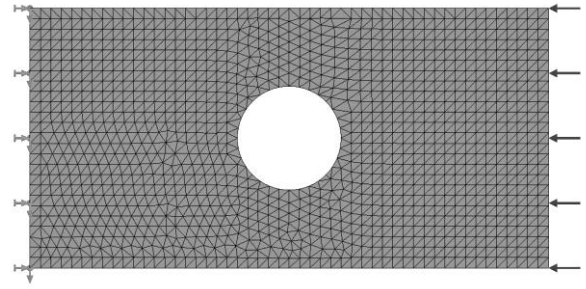


Figure 4 The 2D plate with uniform GAES mesh

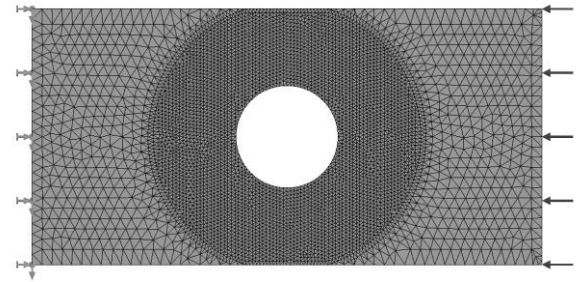


Figure 5 The 2D plate with LMC mesh

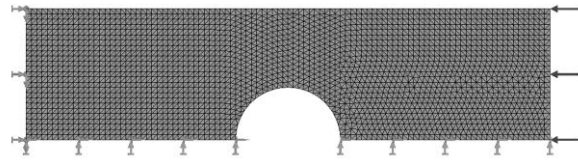


Figure 6 The 2D plate with symmetry conditions

For all cases the displacement was obtained at the same value 0.0059 mm. Because the software calculates first the displacement components at each node from the system of linear simultaneous finite element equilibrium equations this parameter is much insensible then stress parameter. For stress values it can be observed the following conclusions:

- the stress results for case no. 1...6 evolve to a convergence point (18.19 MPa) for *element stresses* and *nodal stresses*; if the solution is close to exact solution, all elements give the same stress values at the common node; but, because FEA is an approximate method, the stresses will be slightly different; so, the difference between *element stresses* and *nodal stresses* values is a measure of the error distribution throughout the model; the difference between minimal and maximal values is much bigger for *element stresses* than for *nodal stresses*; the computational effort increase for the smaller values of the GAES;

- the stress results for case no. 7...15 shows very good values for substantially reduced number of the finite elements; the values are relatively close and constant, even for large domain of the GAES; so the computational effort is reduced comparative with previous variant;

○ the stress results for case no. 16...23 shows the same conclusions like the case 1...6, but because the symmetry restraints will divide the geometry to the half, the precision is greater for the same number of the finite elements.

The figure 7 show the VonMises distribution for the case no. 3.

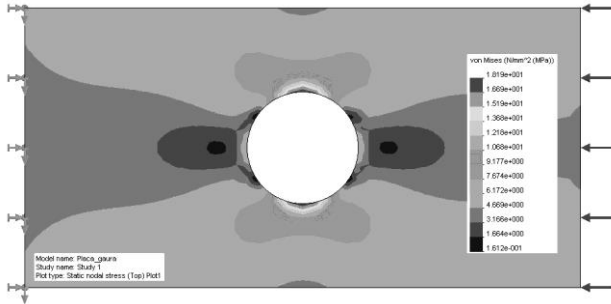


Figure 7 The VonMises distribution for 2D plate

4. The 3D solid analyse

The analyse was made for *high* mesh and SS=ON. The numerical results are presented in table 2 and graphical in figure 8 and figure 9.

Table 2 Numerical results for 3D solid

Case No.	Nodes number	Finite elements number	Medium mesh Value (GAES)	Local Mesh Control (LMC) e/r/n	Von Mises Stress [MPa]	
					Element values/ Position	Nodal values/ Position
1a	7891	4330	18	-	108.5/1	139.3/1
2a	10163	5649	14	-	103.7/1	136.1/1
3a	17100	9978	10	-	107/1	135.9/1
4a	28665	17443	8	-	117.4/1	142.4/1
5a	49967	31367	6	-	125.9/2	160.7/1
6a	103926	67862	4.5	-	136.7/1	179.7/1
7a	305970	207086	3	-	146.1/1	185.9/1
1b	65558	41990	18	-	133.1/1	162.6/1
2b	77407	50108	14	-	124.5/2	160.01/1
3b	93560	61178	10	-	136/1	163.3/1
4b	101748	66651	8	-	133.5/1	170.7/1
5b	100714	65797	6	-	128.6/2	167.7/1
6b	270970	183713	4.5	-	133.1/1	169.2/1
7b	334971	227649	3	-	147.3/1	187.5/1
1c	12164	7102	18	5/1.5/10	127.2/1	158.9/1
2c	14242	8351	14	5/1.5/10	128.6/1	164.4/1
3c	21126	12666	10	5/1.5/10	128.9/1	164.6/1
4c	31213	19137	8	5/1.5/10	125.2/1	156.5/1
5c	52027	32812	6	5/1.5/10	138.1/1	161.6/1
6c	104224	68043	4.5	5/1.5/10	145.9/1	174/1
7c	305678	206834	3	5/1.5/10	145/1	182.1/1

The case no. 1a...7a correspond to uniform mesh with GAES decreasing and AT=OFF, figure 10. The case no. 1b...7b correspond to uniform mesh with GAES decreasing and AT=ON, figure

11. The case no. 1c...7c correspond to uniform mesh with GAES decreasing, AT=OFF and LMC applied, figure 12. Point 1 or 2, figure 12, represent the position of the maximal stress value, also shown in VonMises column in table 2.

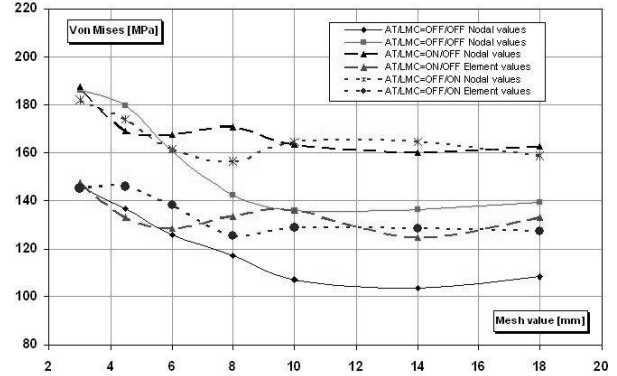


Figure 8 The VonMises distribution for 3D solid

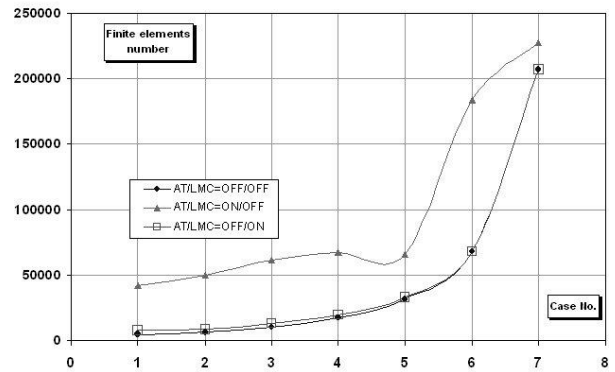


Figure 9 The finite elements number for 3D solid

For all cases the displacement was obtained inside the domain 0.1847...0.1915 mm. For stress values it can be observed the following conclusions:

○ the stress results for case no. 1a...7a evolve ascending for both *element stresses* and *nodal stresses*; the difference between *element stresses* and *nodal stresses* values are bigger than the shell mesh type; the difference between minimal and maximal values is also bigger for both *element stresses* and *nodal stresses*; the computational effort increase for the smaller values of the GAES, because the increase of the finite elements number;

○ the stress results for case no. 1b...7b evolve also ascending for both *element stresses* and *nodal stresses*, but the difference between minimal and maximal values is smaller for both *element stresses* and *nodal stresses* comparative with the previous cases; by activating the *Automatic transition* (AT) option, the software will increase the number of finite elements around the small entities, comparative with GAES value;

○ the stress results for case no. 1c...7c evolve also ascending for both *element stresses*

and *nodal stresses*; also, the difference between minimal and maximal values is smaller for both *element stresses* and *nodal stresses* comparative with the first cases; imposing a local mesh control around the small entities identified as possible location of the maximal stress value will increase the computation precision; the first or second variant can be used to identify those sensible locations of the model;

- the first and third second variant generate the same number of finite elements, figure 9;

- for solid elements the convergence between *element stresses* and *nodal stresses* is not obtained;

- for the first and second variant were obtained different position of the maximal stress values (point 1 or 2).

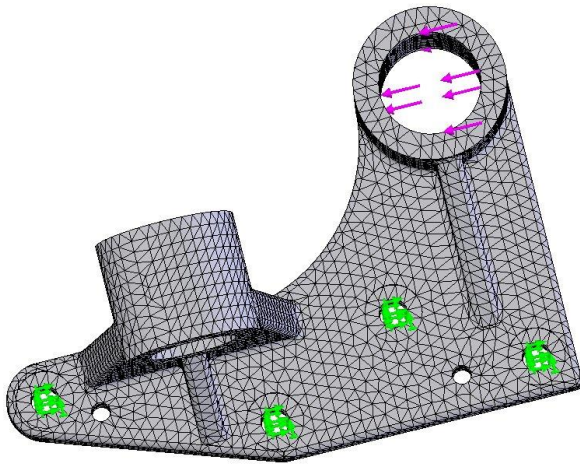


Figure 10 The 3D plate with uniform GAES mesh;
AT=OFF

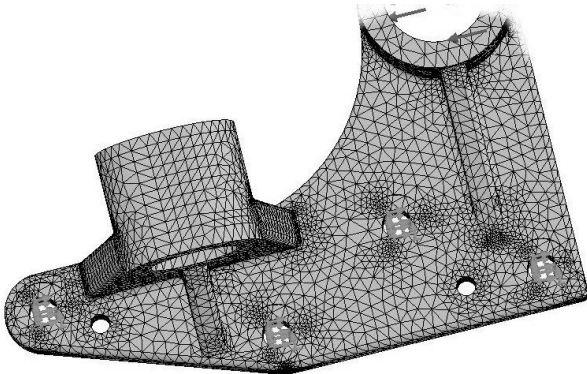


Figure 11 The 3D plate with uniform GAES mesh;
AT=ON

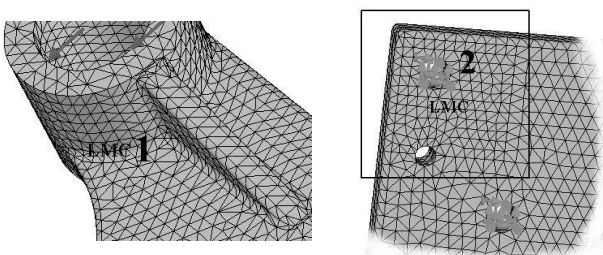


Figure 12 The 3D plate with uniform GAES mesh,
AT=OFF and LMC applied

The figure 13 show the VonMises distribution for the case no. 5c, where the location of maximal VonMises stress value is point 1.

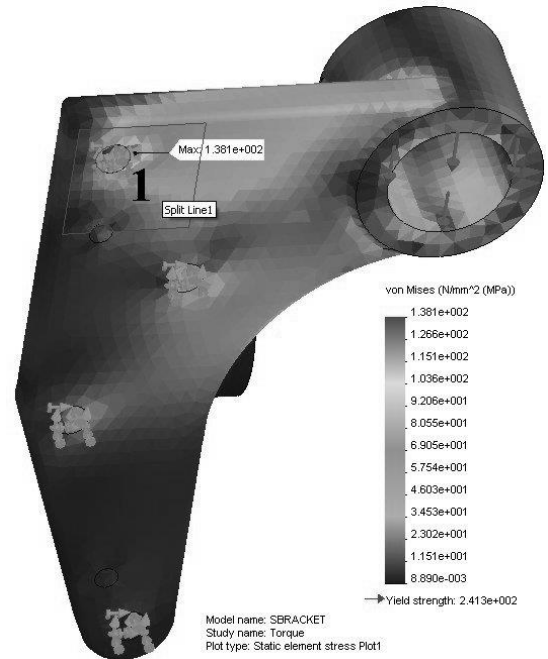


Figure 13 The VonMises distribution for 3D solid with
point 1 location of the maximal Von Mises stress value

The figure 14 show the VonMises distribution for the case no. 5a, where the location of maximal VonMises stress value is point 2.

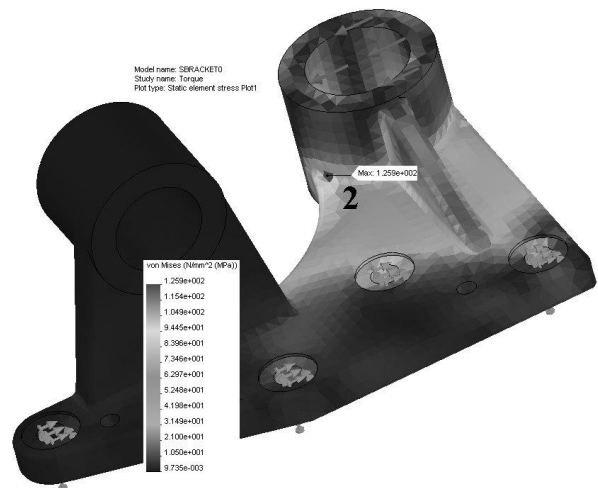


Figure 14 The VonMises distribution for 3D solid with
point 2 location of the maximal Von Mises stress value

5. The conclusions

The *Draft* mesh quality can be used in preliminary studies of very large problems. For the final results must be used *High* mesh quality option.

The *Smooth surface* option improves the quality of the mesh in most cases.

The *Automatic Transition* option must be used for a simple model with few small features.

Activating automatic transition may result in generating a very large number of elements unnecessarily when meshing models with many small features.

To improve results in important areas, it can be used *Local Mesh Control* to set a smaller element size. The important areas can be identified with uniform mesh in a first phase on the analyse.

The convergence study of the results must be verify by any of the following methods:

- increasing of the finite elements number; this method is limited by the computer hardware configuration;
- the comparison between *element stresses* and *nodal stresses* values; this method can be applied for *shell* type mesh;
- the use of the *Automatic Transition* option or *Local Mesh Control* option for sensible areas of the models
- the use of symmetry will improve the results; this method cannot be used for non symmetrical models.

References

1. Manescu, T.Ş., Nedelcu, D.: *Metoda Elementelor Finite*. Editura „Orizonturi Universitare” Timișoara, ISBN 973-638-217-6, Timișoara, 2005 (The Finite Elements Method)
2. Nedelcu, D., Manescu, T.Ş.: *Finite Element Through COSMOS M/Design STAR*. FME Transactions Volume 32, Number 1/2004, University of Belgrade, Faculty of Mechanical Engineering YU ISSN 1451-2092, 2004
3. ***: *Converging on an accurate solution*. <http://machinedesign.com/ContentItem/58789/Convergingonanaccuratesolution.aspx>. Accessed: 2008-11-01
4. ***: *Convergence*. <http://www.mae.ncsu.edu/courses/mae533/klang/Reference/Convergence.htm>. Accessed: 2008-11-01
5. ***: *2D Linear Static Finite Element Analysis Using AutoCAD Mechanical*. [http:// feadomain.com /content67.html](http://feadomain.com/content67.html). Accessed: 2008-11-01.

Considerații privind convergența analizei cu elemente finite

Rezumat

Analiza cu elemente finite (FEA) este o metodă numerică de simulare a proceselor reale, ce oferă o soluție aproximativă în locul uneia exacte. Prezentul studiu se concentrează asupra convergenței calculului cu elemente finite, cu aplicații concrete de rezistență derulate prin programul Cosmos Design Star [1], [2]. Se prezintă concluzii specifice referitoare la convergența și tehnica discretizării [3], [4].

Scientific reviewers: Marin TRUȘCULESCU^{*}, “Politehnica” University Timișoara
Mircea POPOVICIU^{**}, “Politehnica” University Timișoara
