Proceedings of the International Conference on Manufacturing Systems – ICMaS
Vol. 4, 2009, ISSN 1842-3183
University POLITEHNICA of Bucharest, Machine and Manufacturing Systems Department
Bucharest, Romania

THEORETICAL CONSIDERATIONS REGARDING THE FINITE ELEMENTS ANALYSIS OF A MODULAR SYSTEM FOR MILLING PROCESSING

Raluca Magdalena NIȚĂ, Nicoleta RACHIERU

Abstract: *The paper deals with the study using the finite elements method of the behavior under the action of clamping and process forces of a modular system used for clamping the pieces during the cutting process. The performance, reliability, and durability of the system functioning represent the fundamental characteristics having a great influence on the final results of processing.*

Key words: *modular system, structural analysis, FEM analysis, CAD, milling.*

1. INTRODUCTION

The finite elements method is an approximate numerical method of calculation used when it is necessary to analyze problems related to mechanical deformable structures. The static analysis, component part of the finite element analysis, is used to calculate specific strains, displacements, stress, for well defining the dimensional geometry, and loading data for support better conditions specified and for knowing materials characteristics [1].

When a complex system is analyzed, it is necessary to specify the contact between its component parts, because the way the contact is defined has a great influence on the results of the simulation operation.

The contact is a very complex phenomenon and depends on the elastic properties of the bodies that come in contact, their geometry, and also the way that the forces are applied, because the dimensions and form of the surfaces in contact may change under the force action. Also, it can modify the stress distribution on this surface [1, 5]. For a more accurate simulation of the contact between bodies, the contact technology is available, helping the user in the simulation. The results becomes more accurate, even in the case of particularly complex results [4].

2. CAD-FEM INTERFACE

In the design stage, the CAD programs can be used, their results being data files accompanying the design.

These files of the used CAD program represent among others (material, tolerances, etc.) and spatial geometry (3D), full or part of the structure, parts, etc. that are designed. The CAD file is transferred toa FEM program, although the model calculation with FEM does not always correspond to the actual geometry of a structure. Moreover, the transfer may not be complete due to the incompatibilities between the data storage and interpretation in the two programs (file recognized by every program) [4, 6].

For these reasons, the transfer of the data in the FEM program, the geometry of the calculation model, have to be corrected or adjusted according to the program.

The databases created in the CAD programs are specific for each program and can be converted into standard formats. One of the most commonly used types of files used to transfer databases between programs is the file type IGES (Initial Graphics Exchange Specification), which has "igs" extension [5].

3. THE MODELATION OF THE FLEXIBLE ANALYZED SYSTEM

3.1. CAD Model

To achieve the simulation of the behavior of an orientation and clamping flexible system of an axis during the milling process, the experimental model using the CAD soft Solidworks was designed. The proposed system is a modular system design with modular part design by Halder (Fig. 1). At the base system has reinforcements that have the role of taking over the axial movement of the axis caused by the action of milling forces [2].

3.2. The design of the milling conditions

The groove key, having the dimensions 8×4×25 mm, is executed with a four teeth mill, with the cutting depth t = 2 mm, from two successive drilling, the end mill tool diameter being Ø8.

The feed per tooth f_z is 0.10 mm/rot, and feed f = 0.10×·4 = 0.40 mm/rot as shown in Fig. 2.

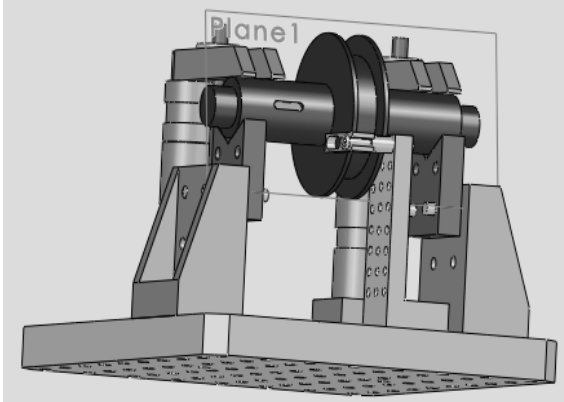

Fig. 1. 3D Model of the clamping system.

The cutting speed is being calculated with relation (1), were C_v represents the cutting constant, D mill diameter [mm], t_l – length of contact [mm], f_z – feed cutting edge [mm/tooth], a_p – cutting deep [mm], z_T – teeth number, K_v – correction coefficient of the speed with the work condition as in [3]:

$$v_{ct} = \frac{C_v \cdot D_c^{\,q}}{T^m \cdot t_l^{\,x} \cdot f_{zd}^{\,y} \cdot a_p^{\,n} \cdot z_T^{\,p}} \cdot k_v \,. \tag{1}$$

From calculation we obtain v_{ct} = 8.2 m/min.

The mill speed n can be calculated with:

$$n = \frac{1000 \cdot v_{ct}}{\pi \cdot D} \approx 326 \text{ rpm.} \tag{2}$$

From the machine book the value for a greater tool speed is chosen that is close to the results of speed calculation. Therefore, if the process is realized on a FUS 22 machine, we have chosen n = 315 rot/min.

The determination of the tangential cutting force [3] is done using the relation (3).

$$F = C_{Fx} \cdot a_p^{\,X_F} \cdot f_z^{\,Y_F} \cdot z_T \cdot D_c^{\,-qf} \,. \tag{3}$$

With the values of the coefficients indicated in [3], we obtain F = 234 *N*.

The horizontal and vertical forces respectively are calculated as percent of the tangential force: F_h = (0.8....0.9)F, F_v = (0.4...0.6)F.

For the horizontal and vertical forces respectively the following values are obtained: H = 200 N, V = 120 N.

4. THE GENERATION OF THE MATHEMATICAL MODEL WITH FINITE ELEMENTS

The first step in a finite element analysis is the model design in particular the structure meshing, the transition from the physical contents of the material from which the structure is executed, to the conventional model – geometric, discrete used in the FEM analysis (FEA).

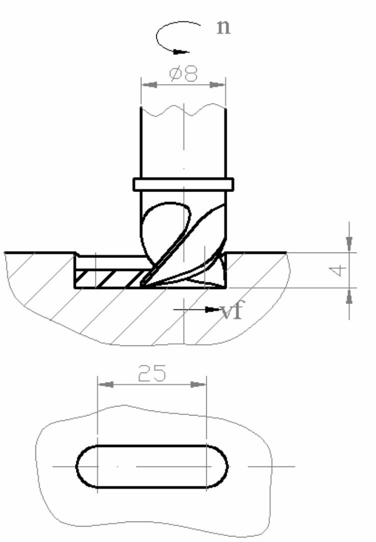

Fig. 2. Groove key processing.

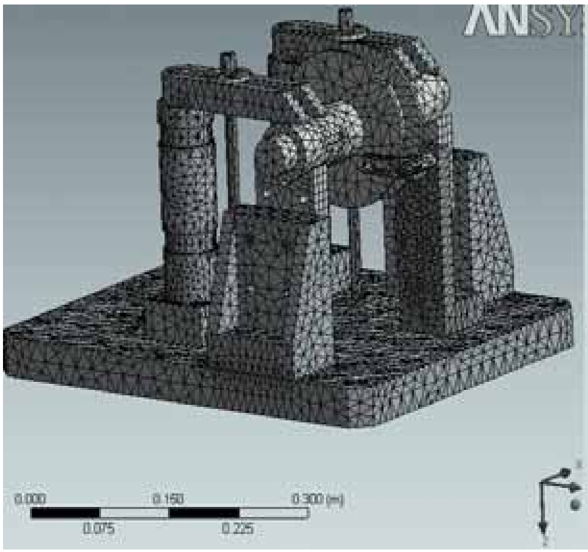

Fig. 3. Network meshing.

Very important issues to be taken into account when generating the mesh are: the range of number of nodes, range of elements, and range of element dimensions. They are required by the structure details for the purpose of analyzing.

The mashing process is complex. Its aim is to obtain a good approximation of the continuous real structure, from a geometrical point of view. Also, the application of the forces, restrictions in support points, rigidities of the masses, are very important for the model.

Therefore, the study of the infinite points of the continuous given structure is approximated by the study of o finite number of points (nodes) of the discrete system network of the calculation model.

The discrete structure approaches from mechanical and geometrical point of view the real structure.

The meshing should be performed on one hand, through a network as simple and as uniform in lines and (or) areas for model development, processing and interpretation of results should be as convenient.

On the other hand, is not always rational that the network to be uniform, because the structure may be areas where there are geometric or mechanical discontinuities (eg, points of application tasks to concentrate) or regions in which for the result, it is necessary a higher volume of information - so nodes and elements.

The structure mesh is uniform and is shown in Fig 3.

5. REZULTS OF THE CALCULTION OF THE STATIC BEHAVIOUR

The static analysis of the flexible processing system assumes the structure deformation evaluation of the workpiece under the action of the fixing forces and also the process forces.

The forces were applied as in Figs. 4, 5, 6, 7 and 8.

The maximum deformation in the fixing screw that supports the lateral buffer has the value of 3.8439 e^{-6} m.

Also, the deformation can be evaluated on the three axes, its values being for the X axis of 3.7948 e^{-6} m, Y axis of 5.17 e^{-7} m (screw fixing tree), and Z axis of 1.22 e^{-6} m (screw that fixed the ladder to catch tree) – Figs. 14 and 15.

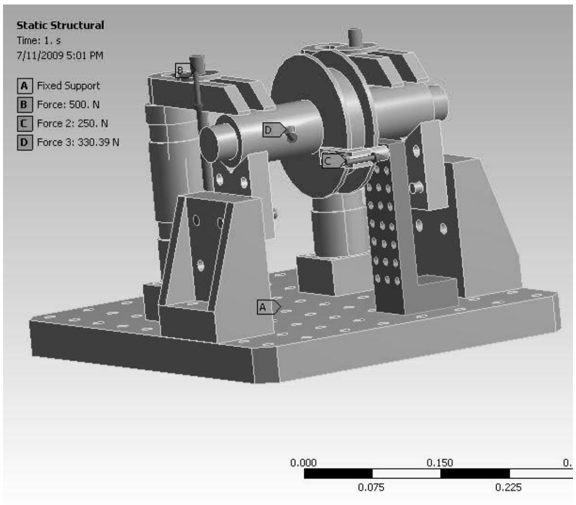

Fig. 4. Forces.

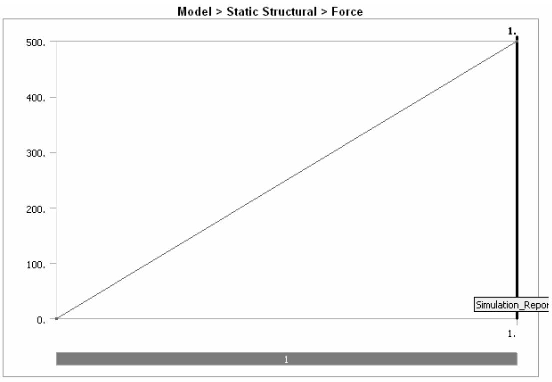

Fig. 5. Clamping force.

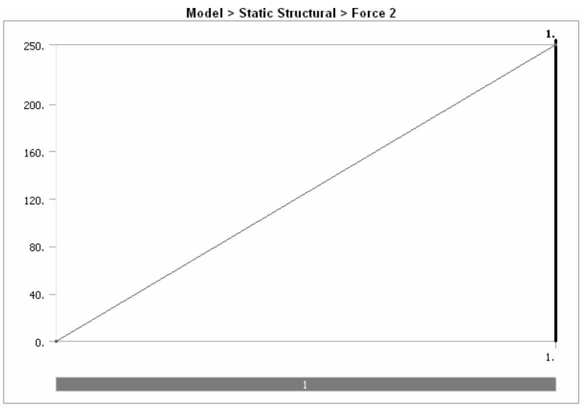

Fig. 6. Axial force.

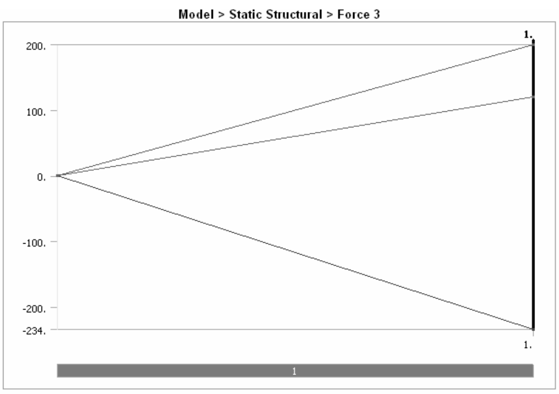

Fig. 7. Milling forces.

Model > Static Structural > Loads

Object Name	Fixed Support	Force	Force 2	Force 3
State	Fully Defined			
Scope				
Scoping Method	Geometry Selection			
Geometry	1 Face			
Definition				
Type	Fixed Support	Force		
Suppressed	No			
Define By		Vector		Components
Magnitude		500. N (ramped)	250. N (ramped)	
Direction		Defined		
X Component				200. N (ramped)
Y Component				-234. N (ramped)
Z Component				120. N (ramped)

Fig. 8. Structural loads

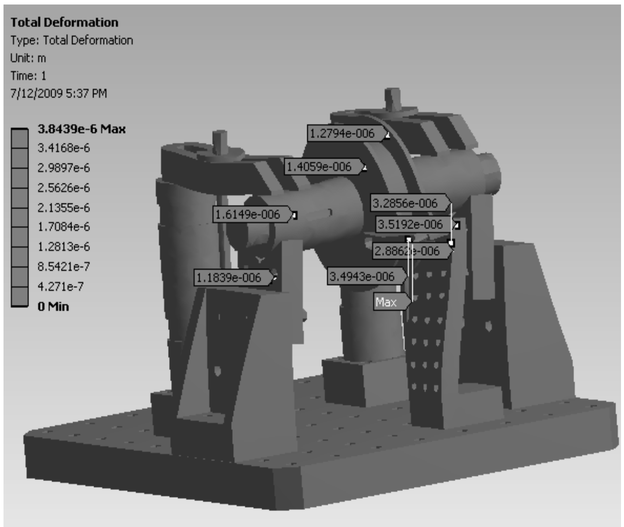

Fig. 9. Total deformation.

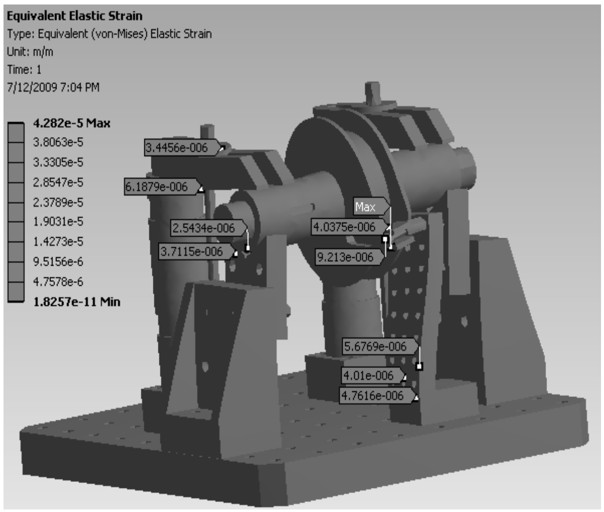

Fig. 10. Equivalent Strain Von Missis.

The post processing phase contains also the effective examination of the simulation results in table or chart form, allowing the evaluation of results.

The graphic representation of results is sometimes more useful for the user (Figs. 10, 11, and 12) .

Total deformation (Fig. 9) has a maximum value on the lateral support, which has the role to limit the axial movement of the axis.

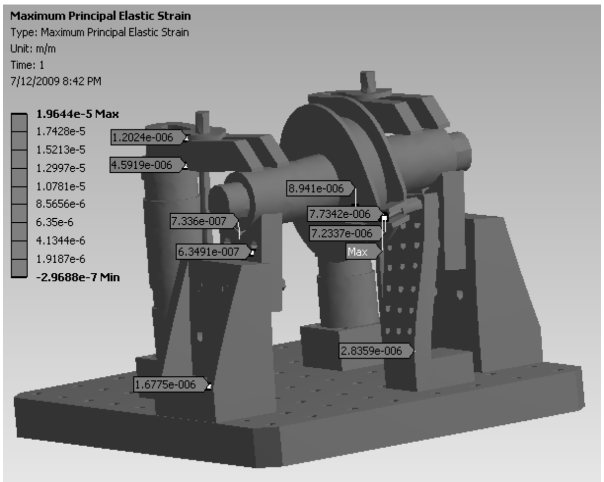

Fig. 11. Maximum principal elastic strain.

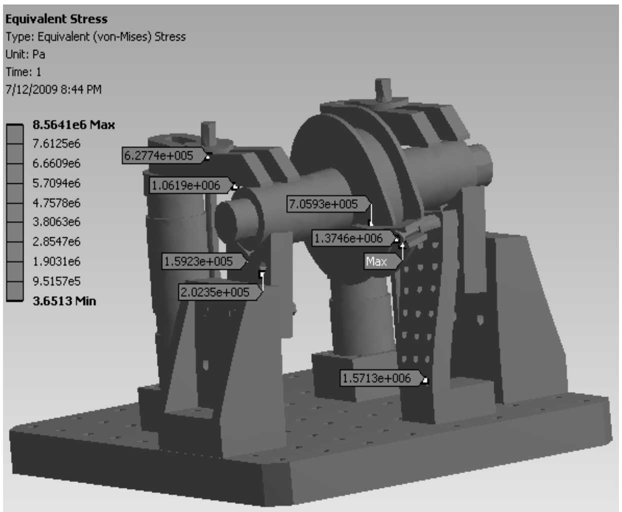

Fig. 12. Equivalent stress.

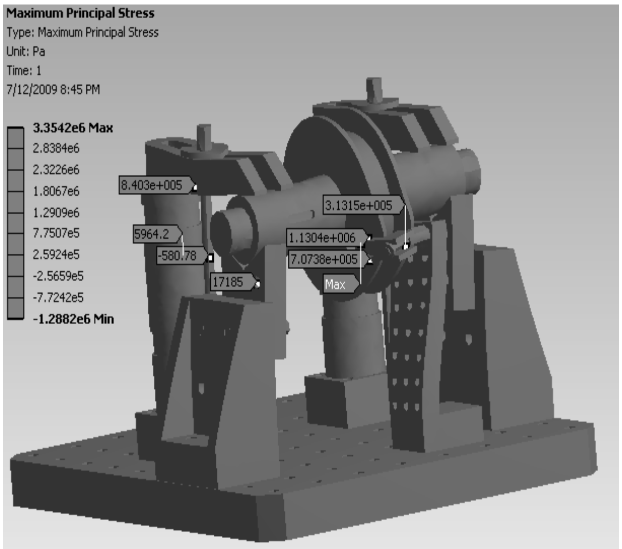

Fig. 13. Maximum principal stress.

6. CONCLUSIONS

The finite elements method is a numerical method for analysis problems related to mechanical deformable structures through which specific strains, displacements and stress were calculated. After result post processing, the simulations results were effectively examined and the zones which can present a risk during processing and also could have a bad influence on the processing precision were determined. For the considered application the deformations are in the admissible limits (0.03–0.05 mm) and the processing precision is not affected.

Object Name	*Total Deformation*	*Equivalent Elastic Strain*	*Maximum Principal Elastic Strain*	*Equivalent Stress*	*Maximum Principal Stress*
State	Solved				
Scope					
Geometry	All Bodies				
Definition					
Type	Total Deformation	Equivalent (von-Mises) Elastic Strain	Maximum Principal Elastic Strain	Equivalent (von-Mises) Stress	Maximum Principal Stress
Display Time	End Time				
Results					
Minimum	0. m	1.8257e-011 m/m	-2.9688e-007 m/m	3.6513 Pa	-1.2882e+006 Pa
Maximum	3.8439e-006 m	4.282e-005 m/m	1.9644e-005 m/m	8.5641e+006 Pa	3.3542e+006 Pa
Minimum Occurs On	EH 2315		Part1	EH 2315	EH 15481
Maximum Occurs On	Part3				EH 11620

Fig. 14. Structural stress.

Model > Static Structural > Solution > Results

Object Name	*Directional Deformation*	*Directional Deformation 2*	*Directional Deformation 3*
State	Solved		
Scope			
Geometry	All Bodies		
Definition			
Type	Directional Deformation		
Orientation	X Axis	Y Axis	Z Axis
Display Time	End Time		
Results			
Minimum	-8.342e-009 m	-1.2662e-006 m	-6.0116e-007 m
Maximum	3.7948e-006 m	5.175e-007 m	1.226e-006 m
Minimum Occurs On	EH 2315	Surub 8	Prezon
Maximum Occurs On	Part3	Surub 8	Part1

Fig. 15. Static displacement.

REFERENCES

[1] Mihailidis A, Pupaza C., NerantzusI, Karaoglnidis G., (2008). *Modeling and simulation of o spherical bearing mount*, Annals of DAAAM for 2008 & Proceedings of the 19th International DAAM Symposium, ISSN 1726–9679 ISBN 978–3–901 509–68–1.

[2]****Modular Elements Catalogue HALDER, (2008)* http://www.halder-inc.com/ .

[3] Picoş, C, et al. (1974). *Calculul adaosurilor de prelucrare şi al regimurilor de aşchiere* (Desing of the addition processing and of the cutting behavior), Edit. Tehnică, Bucharest.

[4] Sorohan S, Petre C. (2004). *Programe si aplicatii cu elemente finite* (Programms and applications with finites elements). Printech, Bucharest, ISBN 973–718–005–4.

[5] Ghionea, I. (2007). *Considerations about the methodology and results for the finite element analysis of a mechanical assembly*, Proceedings of the 16th International Conference on Manufacturing Systems – ICMaS, ISSN 1842–3183.

[6] Sorohan, S., Constantinescu, I. (2003). *Practica modelării şi analizei cu elemente finite.* (Practical modeling and finite element method analysis), Bucharest.

Author:

Eng, Raluca Magdalena NIŢĂ, Ph. Student, University "Politehnica" of Bucharest, Machine and Production Systems Department.

E-mail: ralu_magda@yahoo.com

PhD, Eng, Econ, Nicoleta RACHIERU, S.C. ALPLAST CONF S.R.L., Romania,

E-mail: nrachieru@yahoo.com