D. LETIĆ, B. DAVIDOVIĆ, I. BERKOVIĆ, B. RADULOVIĆ

ISSN 0543-5846
METABK 51(4) 489-493 (2012)
UDC – UDK 621-75:539.371.374:518.1:621.72:620:16=111

DEVELOPMENT AND IMPLEMENTATION OF COMPUTER METHODS AT THE ANALYSIS OF THE DEFORMATION OF THE BEAM BODY WITH THE FINITE ELEMENTS METHOD (FEM)

Received – Prispjelo: 2011-12-09
Accepted – Prihvaćeno: 2012-03-30
Preliminary Note – Prethodno priopćenje

This is a very substantial activity in which designer finds out possible deformations, rigidities (stifnesses) and stress states of the machine parts and fits after forming its virtual geometry, and later the real one as well. They are of the essential importance at the fast and accurate analysis of the possible elastic and plastic body deformation. In this case the point is the solid object of the device beam of an – agro – machine of relatvivelly more complex configuration.

Key words: Strain and Sfifness, FEM modeling deformation, stress states, CAD/CAE

Razvoj i implementacija kompjuterskih metoda pri analizi deformacije tijela nosača metodom konačnih elemenata (MKE). Ovo je vrlo bitna aktivnost u kojoj konstruktor utvrđuje moguće deformacije, krutosti i stanja naprezanja strojnih dijelova i sklopova poslje obrazovanja njegove virtuelne, a kasnije i stvarne geometrije. One su od esencijalne važnosti kod brze i točne analize moguće elastične i platične deformacije tijela. U ovom slučaju radi se o solid objektu nosača uređaja agro-stroja, relativno složenije konfiguracije.

Ključne riječi: Deformacija i krutost, MKE modeliranje, stanja naprezanja, CAD/CAE,

INTRODUCTION

The finite elements method (FEM) can be applied at the finding of deformations or stifnesses of the machine and other parts exposed to forces, pressures, thermal or some other strains (stresses) [1, 2]. It is applied with very complex forms of machine, building and other parts and fits with great static and dynamic vagueness (ambiguity), exposed to complex loads. (loading) [3]. Meanwhile, nowadays this method is also applied for studying many processes, among the others for studying organic tissues, so that it (substantially) significantly contributes to the development of medicine [1]. FEM is the part of the discrete analysis and surely that with it realizes the approximation of the real continuum of the analyzed solid [4].

The classification of the machine part into final elements, the numbering of the lump (key, nude) points and fixing their coordinates, is a very ample task. The parts in mechanical engineering can be of these configurations: linear, flat, space (spatial) or combined ones. Their regulated cluster is a system or network of final elements. Flat final elements are most often of the triangular form. The spatial elements are in the form of the eight – lump geometry objects. They are connected in lump points that present virtual compact connections [5, 6]. Therefore, the application of FEM has become large scale not earlier than when the positioning and solving of these systems of equations have been automated, and computers have become more powerful [7]. As an example can be selected (chosen) the model positioned on (the) two supports, loaded (pressed) by forces in the chosen points. More detailed insighths in this method can be seen in the papers [8, 9] or [10] and they will not be here explained with more details.

D. Letić, I. Berković, B. Radulović, Technical Faculty "Mihajlo Pupin'', Zrenjanin, University of Novi Sad, Novi Sad.
B. Davidović, Technical High School, Kragujevac, Serbia.

The algorithm of the application of FEM method for the calculation of the deformation of the beam with static supports

For the curvilinear beam of the devices (plants) of the agricultural machine is calculated the distribution of deformation and the intensity of stress, applying FEM. Here is used one of the most efficient program packages, Auto CAD Mechanical that contains the necessary tools for the selection and simulation of the values of the needed parameters at FEM calculations. The speed and the efficiency of the algorithm depend on the user, that is the team which uses this method, and certainly it is the engineering team that is in charge of the products development. Meanwhile, as the experiences reveal, this algorithm approach is very efficient as well with the users as students, whose education is directed to the develooping engineering[6]. In the matter of the mentioned software and the arranged solid object designed in CAD/CAE form (Figure 1), the arranged collection / set of procedures should be as follows[10]:

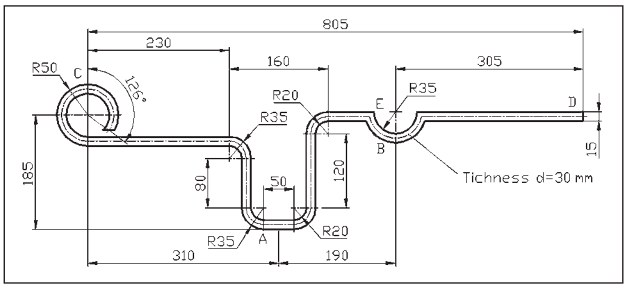

Figure 1 The essential contour and overall dimensions of the cross section of the agriculture machine beam

Preparing the object for the deformation analysis with FEM method

- From the status line of the program include the auxiliary work regimes as the models of the object oriented sights (marks), (of) so – called Osnap type: Endpoint, Midpoint, Center, Quadrant and the others.
- From the running dialogue activate the control Content ▸ Calculations ▸ FEA.
- At the request of the command line: Specify one point within the drawing contour of the model (Figure 1) and confirm (acknowledge) by Enter. The contour is doubled by the new layer.
- In the new – open dialogue FEA 2D - Calculation (Figure 2) fix the next parameters:
- In the field Dafault, the constant thickness of the beam model is of the thickness, d = 30 mm.
- In the field Mesh, adjust the length of the side of the triangle element on the value of 6 mm.
- The material for the object select by this procedure: Table ... , and so open the dialogue Select Standard for Material.
- In the field of the card Buttons open the material catalogue DIN Material.
- In the Dialogue Select Material Type select the material 50CrMo4 with the elastic modulus E = 210 000 N/mm^2, the flowing limit Re = 900 N/mm^2 and Puason coefficient = 0,3. The material properties are automatically, reversibly (recurrently) entered into the dialogue FEA 2D, what can be seen in the Figure 2.
- Close the dialogue Select Material Type and in the main dialogue FEA 2D – Calculation, and then activate the screen key - Config...
- In the dialogue FEA Configuration (Figure 3) adjust the size value of the survery (review) diagram Scale Factor for Symbols on 0,7.
- Close it by the screen key OK.

The defining of the resistance of the supports in the points A and B.

- In dialogue FEA 2D - Calculation select the symbol of the solid support (the third symbol in the field Loads and Supports, Figure 2) which will be positioned in the point A(370, 60) .
- Define the point Aby the sight (mark) Midpoint: Specify insertion point <Enter= Dialog box>: _mid of.

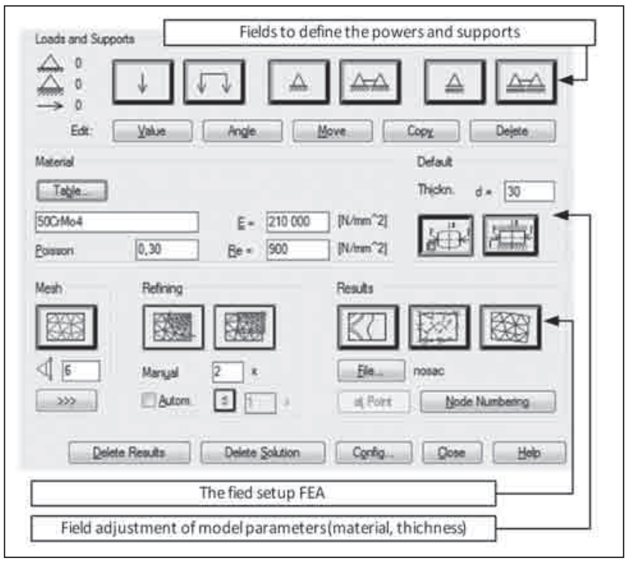

Figure 2 The dialogue FEA 2D - Calculation for 2D with selected thickness d= 30 mm for FEM simulation

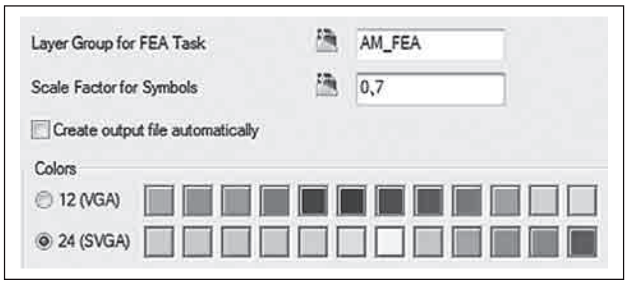

Figure 3 The selection of the secondary parameters FEA

- Define the angle of the resistance of the first beam support: Specify an rotation angle: 270 (Enter).
- In the dialogue FEA 2D - Calculation select the symbol of the loose support (the fifth icon) that (which) will be positioned in the point B(560, 200).
- On the request: Specify insertion point <Enter =Dialogbox> : define precisely the point B by the marker Snap to Quadrant.
- Select the resistance angle of the second support of the object: Specify an rotation angle: 270 (Enter).
- With the previous proceedings are defined both supports in the points A and B.

Defining of the radial forces that effect in the points C and D.

- In the central dialogue FEA 2D - Calculation select the symbol of the spot force.
- Define the effect (action) in the point C: Specify insertion point <Enter=Dialogbox> : _qua of.
- Definisati intenzitet sile u Njutnima: Enter a new value <1 000 N> 1 800 (Enter).
- Specify the angle of the force effect (in degrees): Specify an rotation angle: 90 (Enter).
- In the current dialogue select again the force symbol acting in one point.
- Define the effect in the final D: Specify insertion point <Enter=Dialogbox> : _endp of.

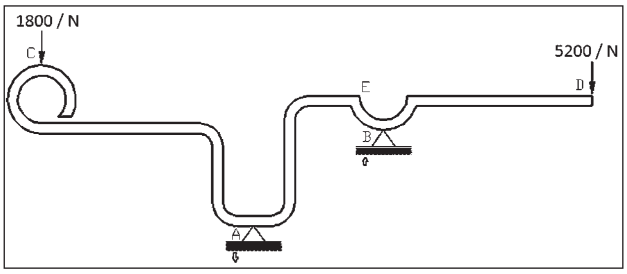

Figure 4 The basic contour of 2D model with the generated forces and the support resistances

- Define the force intensity: Enter a new value <1 000 N> 5 200 (Enter).
- Specify the angle of the force effect: Specify an rotation angle: 90 (Enter).
- The graph of the external (outer) ferces with marked supports is given in the Figure 4.

The calculation of the beam deformations by (with) the method of final elements

- In the field Results of the dialogue FEA 2D - Calculation activate the screen key of the deformation symbol.
- In the dialogue FEA 2D - Displacements activate the screen key (taster) OK.
- The calculation will be done in the command line of the program.
- Here can be followed the next proceedings and the results of the calculation:

Delete - Working...
Generating Mesh - Working...
Delete - Working...
Load Mesh
Calculating:

Generate Nodes in middle of edges of triangles
Search loads and supports
Number of elements 1 689, Nodes 3 924
Renumbering of Nodes
Allocation of memory for Equation System 4 151 kB
Preparation of Equation System
Calculation of Equation System
Calculation of Stresses
Calculation of Inner Loads
Write Support Loads
Write calc.values in Mesh
Delete - Working...
Displacement - Working...

- Forming of the basic point of the graph (diagram) dislocation of the deformation of the beam model can be, e.g. in: Specify base point <Return = in boundary>: 500, 100 (Enter).
- Here define also:
- The point of the entering of the deformation graph, e.g.: Specify insertion point: <Ortho on> @0,250 (Enter).
- The point of the entering of the table with the deformation values:
  Specify insertion point: 1 000; 370 (Enter).
- Continue the proceeding with: <Return> (Enter).

The calculation of the intensity of stress in the beam by the mathod of final elements

- In the field Results of the dialogue FEA 2D - Calculation activate the screen key of the symbol of the stress schedule.
- In the dialogue FEA 2D - Isolines (Isoareas) accept (on default) the first screen key of the isoline contours (Figure 2), and then initiate the screen key OK.
- In the command line can be followed the next proceedings and results of the calculation:
  Delete - Working... Von Mises Isolines – Working...
- Form the basic point of the dislocation of the stress graph in the model: Specify base point <Return = in boundary>: 500, 100 (Enter).
- Here define as well:
- The point of entering the stress graph, e.g.: Specify insertion point: <Ortho on> @0,-270 (Enter).
- The point of entering the table with the value, stress, e.g.: Specify insertion point: 1 000, -220 (Enter).
- Check the stress value at the point E. The point is precisely defined by Endpoint on the lower contour stress graph .
- The stress value is automatically fixed and it is given in N/mm^2: Select isoline for description <Return> 96,468 (Figure 5).
- The proceeding will be finished by activating the screen key Close.
- Zumming of the all parts of the drawing can be performed by the proceeding Zoom Extents:
- Here can be followed the next proceedings and the calculation results:

Command: '_zoom
Specify corner of window, enter a scale factor (nX or nXP), or [All/Center/Dynamic/Extents/Previous/Scale/ Window] <real time>: _e
Command: *Cancel*.

The analysis of the beam deformation

- Through with the calculation of the deformation and the stress by FEA method is comprised (covered, included) 1 689 final elements with all in all (totally) 3 924 lumps (nodes).
- The necessary memory for the calculation was 4 151 kB.
- The calculated resistance force of the support at the point A is F_A = 3 610,53 N and it is of the opposite

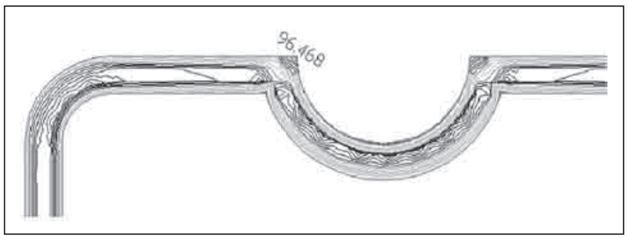

Figure 5 The stress value at the selected (chosen) point E in the regime of the isoline stress performance

(contrary) direction of the previously fixed one (direction).

- The calculated resistance force of the support at the point B is F_B = 10 610,53 N and is of the same direction in relation to the previously fixed one.
- The greatest deformation of the machine part is on , the right end. The feed in the direction of Y axis is Δy = 10,54 mm.
- This and other extreme values on X axis can be directly/read from (out of) the working surface. So is, e.g. maximal deformation value on FEM basis, also, equal: MaxY x Coeff. = - 60,22 x 0,1817 = 10,94 mm (Figure 6). Similarly it can be obtained as well for the longitudinal deformation.
- By picking on some isolines, after the phase of the forming of the graphic performance of the stress, are being obtained the numerical stress values on the very graph. One of such values, as it is given in the Figure 6, is generated from the point E.
- The stresses in the material are complex, beginning from the pressure to the extension (stretching) and primary bending.
- All visualized deformations are on the graph of deformation and structure of the final elements (Figure 6 and 7) or in the corresponding table (Figure 8). One of these stresses is extreme and it is σ_{max} = 1 543,4 N/mm^2.
- This stress ought to be compared with the allowed, that is critical stress and the stress of flowing for the selected material. It is very important criterion at the analysis and decision forming in FAM methodology.
- The highest (greatest) stress concentration is in the location of the moving support B (Figure 8). Because of that, there ought to be positioned the corresponding radius of rounding (bulging, crowning) in order to increase the design reliability. Meanwhile, it is only one of the proceedings to the integral correction with the aim to satisfy the criterion functions at (during) this analysis.

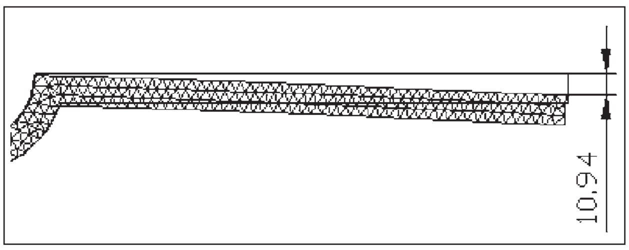

Figure 6 The value of the maximal radial deformation in the field of C point

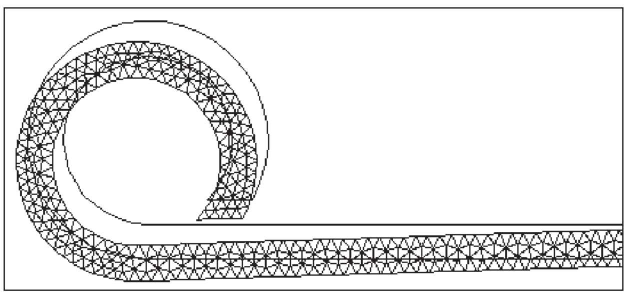

Figure 7 The value of the maximal radial deformation in the field of C point

CONCLUSION

At the repeated calculation by FEM method of the intensity of stress and / or deformation, there can appear certain deviations in the obtained results. They are usually of the greatness rank of 1 % to 5 %. By the computer simulation applying FEM, can be obtained as well the data for the complex 3D forms of the machine parts, substantially faster than by the other methods. After the finished calculation, the results will be presented on the same drawing, so that within the defined limits the form and the greatness of the intensity of stress are coloured in the defined (automatic) way so that one can get right insight in its distribution. FEM enables obtaining the stress values and deformations in the whole machine part, through the program way and developed and arranged set of proceedings as it is here recommended. It is similarly to the experimental proceedings, meanwhile with considerably more data indispensable to the engineers of development. Here can be inmediately seen are there any zones where the stresses are greater than the critical ones, and if they exist, one performs certain corrections. It is done through: material changing, location of the source of concentration of deformation and stress, change of dimension or load, or the place of its acting. From the previous calculations it is clear that deformations, distributions and intensities of stress exceed the

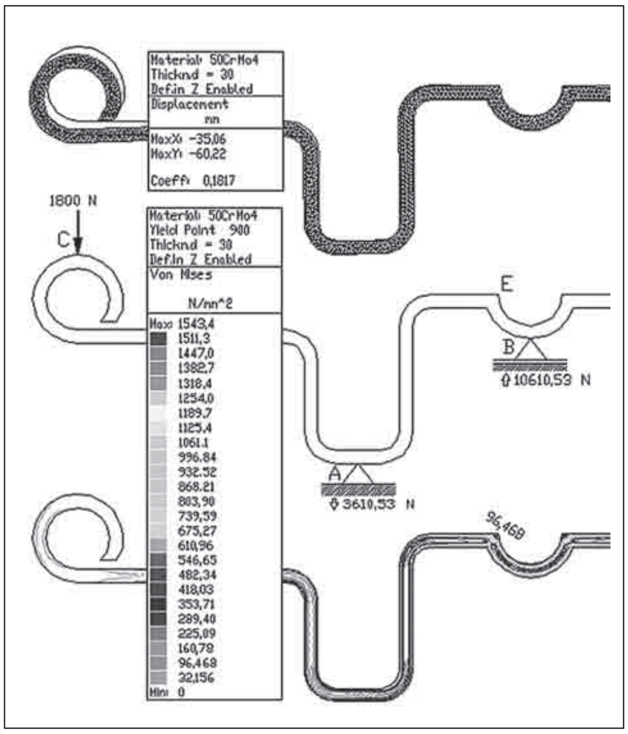

Figure 8 The basic contour of the 2D (3D) model with the forces and resistances lattice structures of final elements: the graph of forces and resistances of the supports and the graph of stress distribution with tabular summary (outline) of the results of FEM calculating

value of the limit of elasticity, i.e. the limit of plastic deformation (flowing).

The flowing limit for this material is 900 N/mm^2, while many stresses exceed this value on the basis of the insight in the tables in the Figure 8. In that case one ought, as it is omphasized, to change the beam material, or decrease (reduce) the intensity of forces. If it is impossible, the third possibility, which now gets (has) the primary role, comprises the changed geometry and repeated dimensioning of this solid object. With this, already in the phase of designing would be avoided the risk of deformation such a beam, comprising as well indispensable degree of safety, so that the allowed resulting intensity of stress (here it is analizyed as a bending one) would be thrice lower (smaller) than the limit of the plastic deformations.

High (great) stresses can cause (incite) permanent deformations of the machine part at those positions, and it gives a clear sign to the designer that something in the draft (form) must be changed. The results of the previous FEM calculation confirms it clearly (Figure 5). It means that the significant changes must be accomplished (fulfiled, realized) in the designing and projecting. Applying sich a way of designing, can be avoided the production (operation, make) of a classical prototype what brings multiple benefit (favour). Because of the complexity and extent (scope) of the procedures, this method only recently gets wider application especially owing to the higher degree of the development of the computer technologies and perfection of the cheaper programs for its execution. It is perhaps the most important point for designers to become interested in the using of the new calculation methods as FEM or e.g. the methods of the final elements – MOE, the method of the mixed elements - MME and the like [5]. In this way will be enabled the research work which would comprise more complex models, and using simulation and animation and point to the concentration of the intensity of stress and elastic and inadmissible (prohibited) plastic deformations on the models formed in the virtual space and exposed to the dynamic load.

REFERENCES

[1] Cook D. R., Finite Element Modelling for Stress Analysis, John Wiley & Sons, Inc, (1995), 26

[2] Sekulović, M. The Final Elements Method, IRO "Building Book", Belgrade, (1988), 6

[3] Rao S. S., The Finite Element Method in Engineering, 2-nd Ed., Pergamon Press, (1989), 641.

[4] Solin P., Segeth K., Doleze I., Higher-Order Finite Element Methods, Chapman & Hall/CRC Press, (2003), 11

[5] Poceski, A. The Mixed Method of the Final Elements, DIP "Building Book", Belgrade, (1990), 16-18

[6] Staniek R., Gessner A., Zielnica J., Ptaszyński W., Myszkowski A., Ciszak O., Stoić A., Stress and displacement analysis of a modern design lathe body by the fi nite element method (FEM), Metalurgija, 51 (2012), 1, 51-54

[7] Lee, K. Principles of CAD/CAM/CAE systems, Addison – Wesley, USA, (1999), 7

[8] Stein E., Zienkiewicz C. O., A pioneer in the development of the finite element method in engineering science. Steel Construction, 2, (2009), 4, 264-272.

[9] Kovačević D., Soković M., Budak, I., Antić A., Kosec, B., Optimal finite elements method (FEM) model for the jib structure of a waterway dredger, Metalurgija, 51 (2012), 1, 113-116

[10] Letić D., Desnica E., Davidović B., AutoCAD Mechanical 2011, Computer Library, Belgrade, (2011), 2-17

Note: The responsible translator of the English language is Srđan Šerer, Technical Faculty "M. Pupin", Zrenjanin, University of Novi Sad, Serbia