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

ISSN 0543-5846
METABK 51(4) 513-517 (2012)
UDC – UDK 620.16:621-59:518.5=111

THE HIGH – PERFORMANCE ALGORITHM OF THE COMPUTER METHODS AT THE ESTABLISHING OF THE STATES OF STRESS OF THE BRAKE MECHANISM BY THE FINITE ELEMENT METHOD (FEM)

Received – Prispjelo: 2011-12-09
Accepted – Prihvaćeno: 2012-02-25
Preliminary Note – Prethodno priopćenje

Designing of the high – performance algorithms by the computer methods at the establishing of the states of stress of the brake mechanisms by the methods of the final elements is very substantial with fast and precise analysis of the state of stress and rigidity of the machine parts and the fits of machine parts affter forming its virtual, and later as well as real geometry. There are multiple reasons for it, and they include: economy, interchangeability and primarily its operating certainty, whose function is unavoidable especially with the parts as the brake mechanisms. To that effect are the results in the designing obtained by final elements analysis (FEA) or similar methods, very useful.

Key words: stress, brake mechanism, FEM modelling, CAD/CAE

Efikasni algoritmi kompjuterskih metoda pri utvrđivanju stanja naprezanja kočionih mehanizama metodom konačnih elemenata (MKE). Algoritmi su vrlo bitni kod brze i točne analize naprezanja i krutosti strojnih djelova i sklopova poslije obrazovanja njegove virtuelne, a kasnije i stvarne geometrije. Višestruki su razlozi za to, kao što su: ekonomičnost, izmjenljivost djelova a prvenstveno njegova eksploataciona pouzdanost čija je funkcija nezaobilazna naročito kod djelova kao što su kočioni mehanizmi.

Ključne riječi: naprezanja, kočioni mehanizmi, MKE modeliranje, CAD/CAE

INTRODUCTION

The Final Elements Method (FEM) is a part of the group of modern numerical metods [1, 2]. It can be applied also with very complex forms of the machine, building (constructional and other parts and sets with high statical indefiniteness, exposed to the complex loading [3]. The applying of the analytical methods for these calculations would be inefficient. The essential idea of FEM is in the division of the continuous mass (volume) of 2D or 3D models onto the final elements of highly great precision [4]. The Final Elements Method belongs to the method of discreet analysis, by which the geometry continuum of some material is divided on the series of adequately little elements which are final and present the approximation of the real continuum. They can be linear, flat or space, and their arranged assembly is a system or a net of final elements. Flat final elements are most often of the triangular form, and flat ones are in the form of the eight – lump geometrical body. They are connected in lump points that present imagined lump connections [5]. For every lump point where are connected final elements, are positioned the balance equations of the forces of elasticity that define the connection of the intensity of stress and deformation [6].

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

The balance of the final elements is analogous to the balance of the members in the lattice girder. External loadings and limitations are included in the equations for the final elements on the form outline (contouring conditions). By solving the system of the adequate equations, one can obtain the values of motion, that is the deformation and the intensity of the stress in all the lump points. The very division of the machine part to the final elements and the determination of their coordinates, is very comprehensive task. In addition, the systems of the equations that should be set and solve, are in principle enormous. Therefore, the application of FEM has become large scale not earlier that the positioning and solving of these systems of equations have been automated, and computers have become powerful [7]. The modern FEM – s enable high – speed calculations of the intensity of stress and deformations provoked by the external forces through the heating that is being hampered by thermal and other dilatations. In addition, one can vary form and model dimensions, in order to achieve high degree of mass rationality, high reliability at work, and especially high accuracy of the calculation results. The rapid development of the last Final Elements Method during the last 60-s years (Hrenikoff, 1941.) was going parallel with the development and application of the computer technologies [8]. For the first time the Final Elements Method was applied with the stress – strain analysis of a fuselage, and wider application has begun only when the manual operations have been automated. So it has been enabled to the engineer to perform corrections and simulations on a model in order to forsee its behaviour after the making, the using respectively. The FME enables to its designer nowadays to envisage intensities of stresses and deformations, the vibration regimes, to form the optimal forms of the machine parts, the fluids motion [9] and else. It can be applied in the cases when is neccessary precise analysis and when the product cost has an important role. Especially are by calculations covered the machine and other objects with high – risk performances. The spatial 3D final elements have bee applied as well with exceptionally complex forms in space. The surface 2D final elementts enable higher degree of the calculation efficiency. They can be applied with flat (stab - like) forms of the same thickness (Figure 1). In addition the impact of the third dimension (the length of the object or circumference) surely takes into consideration at the rigidity determing and / or the mass of the final element at forming the adequate matrixes.

THE ALGORITHM OF THE APPLICATION OF FEA FOR THE CALCULATION OF THE INTENSITY OF STRESS OF THE MACHINE PART

For the brake lever is calculated the division of the intensity of stress applying FEM. In addition is used one of the most efficient program packages AutoCAD Mechanical 2012 that contains the necessary tools for the selection of the majority of the reguired parameters at the FEM calculation. The speed and efficiency of the algorithm depend on the user, that is the team that use this method, and certainly it is the engineering team that is in charge of the development of products. Meanwhile, as the experiences reveal, this algorithm approach is very efficient with the students whose education is directed to the developing engineering and the computer methods of designing [6]. In the matter of the mentioned software, the arranged collection of procedures should be as follows [10]:

- To prepare the object designed in CAD/CAE form [11]. See Figure 1.
- From the status line include the auxiliary work regime as the modes of the object oriented sights, so-called Osnap type: Endpoint, Midpoint, Center, Quadrant and the others.
- Activate the control Calculations ▸ FEA.
- At request: Specify interior point: to specify a point within the drawing contour of the brake lever.
- In the open dialogue FEA 2D - Calculation (Figure 2) fix the next parameters:
- In the field Thickness enter the thickness of the part as *d*=15 mm.
- In the field Mesh, adjust the length of the side of the final element in the form of traingle on the value of 2,5 mm.
- By the procedure Table... open the dialogue Select Standard for material (Figure 3). In the adequate field of the mode Buttons select the material catalogue according to DIN standard. The concrete selected material is AlMn1F10.
- In the basic dialogue FEA 2D - Calculation (Figure 2) activate ek. key Config... and in the dialogue

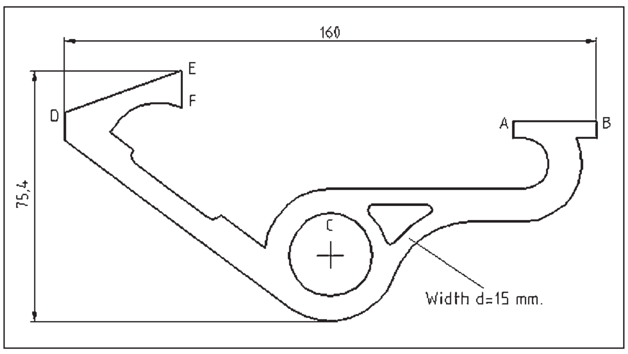

Figure 1 The essential contour and overall dimensions of the cross section of the brake lever

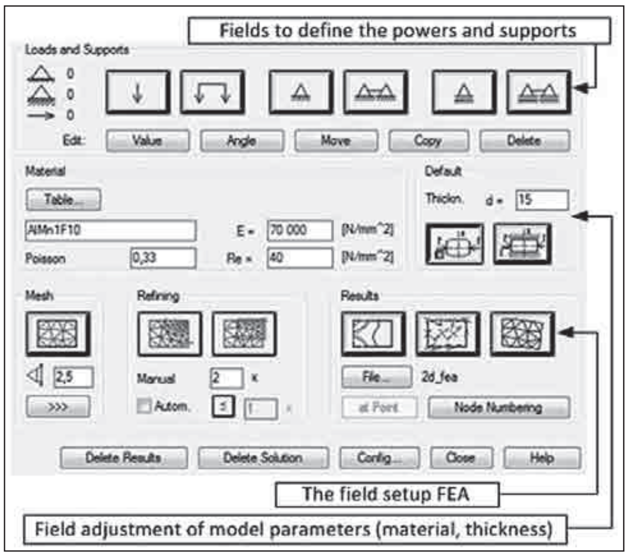

Figure 2 The dialogue FEA 2D - Calculation for 2D analysis by the method FEA

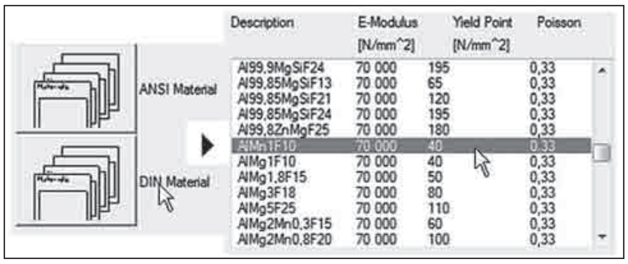

Figure 3 The selection of standard material of the object of the analysis

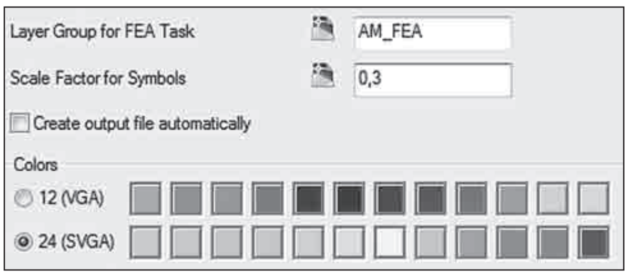

Figure 4 Selection of visual parameters FEA

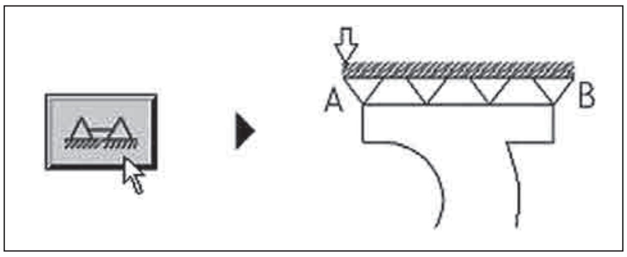

Figure 5 The phases of the solid support defining

FEA Configuration (Figure 4) adjust the scaling value of the Scale Factor for Symbols on 0,3.

- In order to define the resistances of the supports along the line A-B select the symbols of a solid continual support which will be put between the points A and B, precisely, on the Osnap basis (Figure 5).
- The defining of the loose resistance of the supports in the pivot C can be performed through the symbol of the loose support of the continual support whose support will be the axle fixed in the opening of the brake lever. To that effect should be precisely defined one point on the circular surface (Figure 6).
- Defining by Osnaps of the surface forces along the interval D-E.
- Enter the value of the specific force: Enter a new value <1 000 N/mm > 80 (Figure 7).
- Defining of the concentrated force in the contral point between the points E and F.
- In the dialogue FEA 2D – Calculation select the symbol of the forces that act in that central point.
- The direction of the forces closes the angle of 90^o to the vertical side.
- Define the central point F: Specify insertion point <Enter=Dialogbox> .
- Enter the force intensity: Enter a new value <1 000 N> 120.
- Enter the vector angle of the force: Specify a rotation angle: 0 (Figure 8).

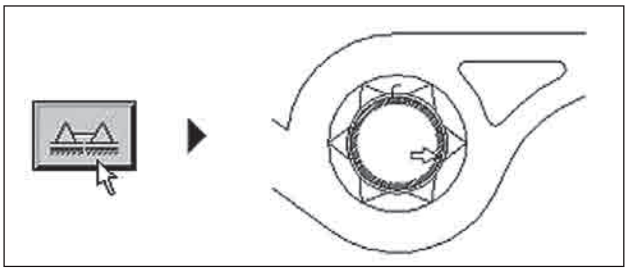

Figure 6 The defining phases of the loose support

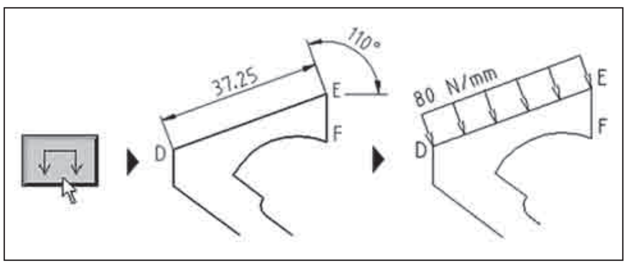

Figure 7 The defining phases of the width, angle and line of the actions of the forces

THE FORMATION OF THE FINAL ELEMENTS NET AND THEIR CALCULATION

- In the dialogue FEA 2D – Calculation click on the symbol Mesh, and after that are being formed the final elements (Figure 8).
- In the field results select the first option. Thus is being opened the dialogue FEA 2D - Isolines (Isoareas), and after that click on the ek. key of the full contours (Figure 9).
- The click on the ek. key is automatically initiated the formation of the graphic for the division of the stress intensity.
- The calculation will be displayed in the control line and can be followed on the basis of the next procedures and results:

  Generating Mesh - Working...
  Delete - Working...
  Load Mesh
  Calculating:
  Generate Nodes in middle of edges of triangles
  Search loads and supports
  Number of elements 952, Nodes 2 175
  Renumbering of Nodes
  Allocation of memory for Equation System 3 625 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...
  Von Mises Isoareas – Working
- The graphic presentation of the calculation results is being performed by formation of the basic point for the location of the grapic and the table with the numerical values of the stress intensity division of FEM. The procedure is ended with: <Return>
- The graphical display of the calculation results is presented in the Figures 9 and 10.
- By the analysis is established that the calculation is performed on 952 elements with 2 175 included lumps. The necessary memory for the calculation was 3 228 kB .

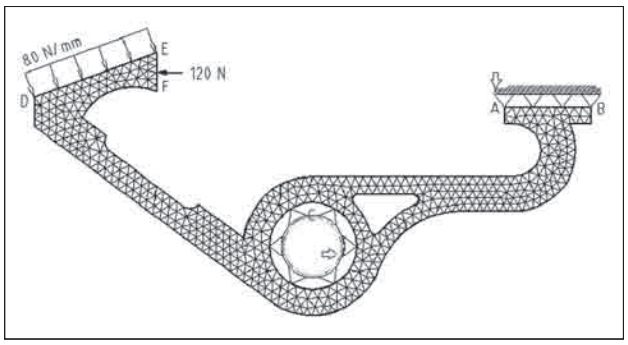

Figure 8 The net of final elements and the vectors of the external forces

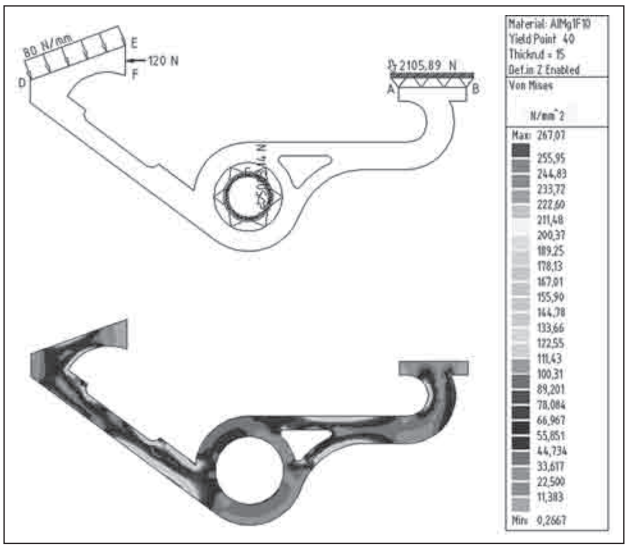

Figure 9 The graph and the table of the schedule of the intensity of stress on the object

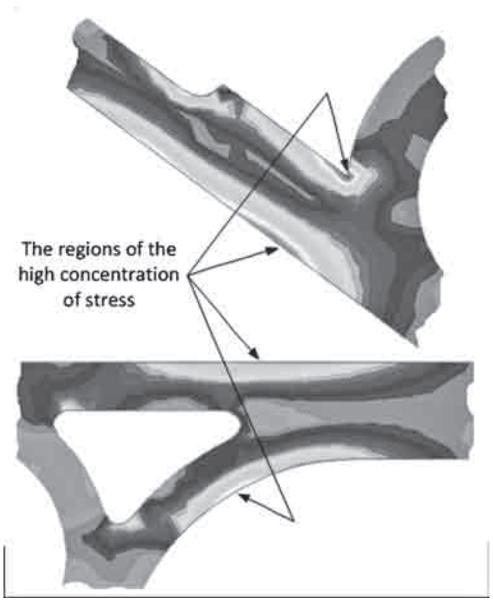

Figure 10 The regions of the high concentration of stress

- The intensities of stress are complex and they refer to the pressure, extension and bending.
- The highest intensity of stress can be seen in the graphic display and on the table and it is 277,35 N/mm^2. This intensity of stress should be compared with the allowed one, critical intensity of stress for the given material, respectively
- In addition to it, should be compared as well as the intensities of stress that are higher than the intensity of stress of flowing of the material, because they are in any case highly critical.
- The highest concentration of the intensity of stress is established in the angle between the slanting lever and the external circumference of the journal (Figure 10).
- Because of these reasons should be replaced the material with better performances, improve the brake lever geometry, e.g. set the adequate radius

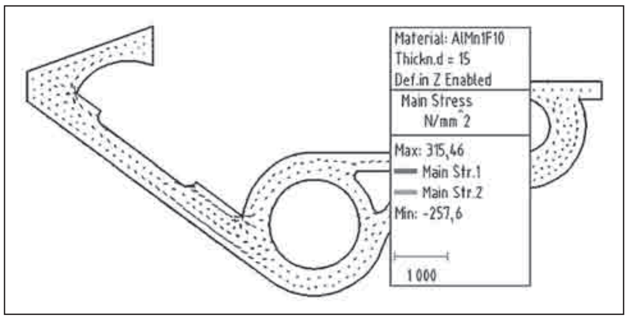

Figure 11 The chart and the table of the schedule of the vectors of stretching and pressing internal intensities of stress in the object

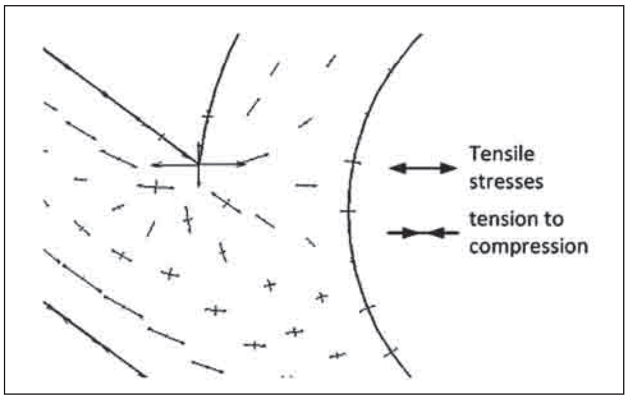

Figure 12 The one region of the concentrated stretching and pressing intensities of stress

of the rounding which with this form on that place lacks and / or expand the thickness of this machine part.

- The displays of the intensity of stress can be presented as well with the vector form as it is given in the Figure 11, Figure 12, respectively.
- In addition is applied another procedure from the dialogue FEA 2D - Calculation.

THE FEM METHOD AND CONCLUSION

By the computer simulation of the FEM application can be obtained as well as the data also for the complex 3D forms of the machine parts substantially faster than by the other methods (Figure 13). After the performed calculation the results are presented on the same drawing so that in the defined borders the form and size of the intensity of stress are painted in a specific way; so can be obtained the precise imsight in its distribution. FEM enables, by the use of computer, to obtain the extents of the intensity of stress and deformations in the whole machine part, similarly to the experimental procedures, but with considerably more data. With this immediately can be seen are there any zones where the intensities of stress are higher than critical ones and, if they exist, certain corrections are being performed changing material, the forms of the sources of the intensity of the stress concentration, changing dimension or loading, or the location of its activity. From the previous calculations is clear that many intensities of stress sur-

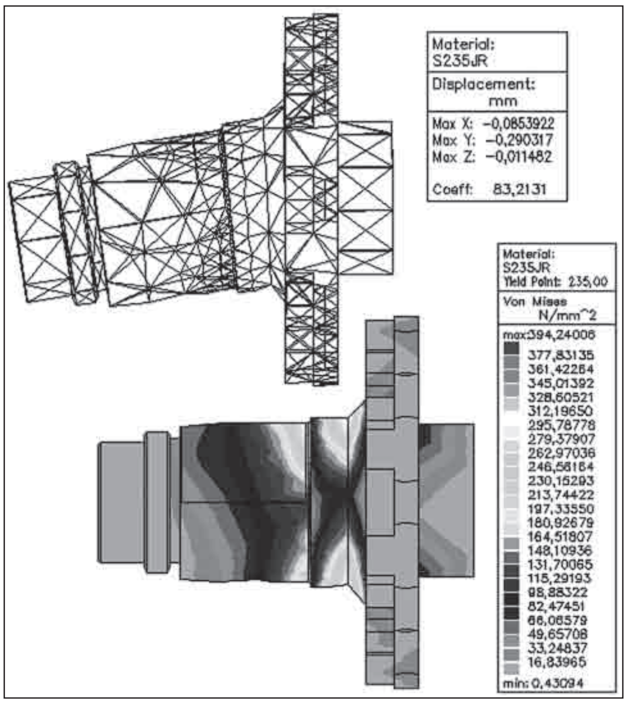

Figure 13 The graph and table of the schedule of the intensity of stress and deformations on 3D object

pass the value of the elasticity limit i.e.the limits of the plastic deformation (flow) in the both of considered cases. High intensities of stress can cause permanent deformations of the machine part at those places. It is the clear signal to the designer that something must be changed in the concept. The results of the previous FEM calculation of FEM distinctly confirm it (Figure 9), and so points that both in the geometry and in the material have to be performed substantial changes. Applying such manner of designing, one can avoid the making and testing of the classical prototype, and it gives multiple benefits.

As the consequence of complexity and extensivity of procedures, this method only recently begins to be widely used, owing to the higher degree of development of the computer technologies and detailed elaboration of the program for its performing.

Perhaps the most important fact is that designers become interested in the use of these calculation methods like FEM, or e.g. the method of the limiting or boundary elements (LEM or BEM), the method of the mixed elements (MME) and like [5]. In this way are opened the spaces for the research work which would comprise more complex models, and using simulation and animation, would point to the concentration of the intensity of stress and deformations on the models formed in the virtual space, and which are exposed to the dynamic loading .

REFERENCES

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

[2] Sekulović, M. Metod konačnih elemenata, IRO Građevinska knjiga, Beograd, (1988), p. 6

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

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

[5] Poceski, A. Mešoviti metod konačnih elemenata, DIP Građevinska knjiga, Beograd, (1990), p.

[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 finite 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., CAD mašinskih elemenata i konstrukcija, Ogledi u AutoCAD Mechanical 6, Kompjuter biblioteka, Čačak, (2004), p. 18

[11] Letić D., Desnica E., Davidović B., AutoCAD Mechanical 2011, Kompjuter biblioteka, Beograd, (2011), p. 2-17

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