Proceedings in MANUFACTURING SYSTEMS

Proceedings in Manufacturing Systems, Vol. 5 (2010), No. 3 ISSN 2067-9238

FINITE ELEMENT ANALYSIS OF BOLTED JOINT

Iuliana PISCAN[1], Nicolae PREDINCEA[2], Nicolae POP[3]

Abstract: *This paper presents a theoretical model and a simulation analysis of bolted joint deformations. The bolt pretension force, friction coefficient and contact stiffness factor are considered as parameters which are influencing the joint deformation. The bolted joint is modeled using CATIA software and imported in ANSYS WORKBENCH. The finite element analysis procedure required in ANSYS WORKBENCH simulation is presented as a predefined process to obtain accurate results.*

Key words: *joints, bolts, machine tools, FEM, analysis, stiffness.*

1. INTRODUCTION

Machine tools are characterized by high precision, even at heavy-duty regimes (high magnitudes of cutting forces). This requires very high structural stiffness.

Static stiffness k_s is defined as the ratio of the static force P_o, applied between tool and workpiece, to the resulting static deflection A_s between the points of force application.

$$k_s = \frac{P_0}{A_s}. \tag{1}$$

The stiffness of a structure is determined mainly by the stiffness of the most flexible component in the path of the force. To enhance the stiffness, the flexible component must be reinforced. An analytical and experimental (on the machine tool) evaluation of component deformations and compliance in the cutting edge, should be undertaken in order to evaluate the influence of various structural elements of the overall stiffness.

Local deformations in joints, for example bolted connections between rigid elements such as column and bed, column and table, and movable joints such as guideways are usually found to be the most flexible components of the breakdown (Fig. 1).

The rigidity of a structure having a bolted joint is low as compared with that of an equivalent solid structure.

Because the bolted joint is a widely used assembly method, a very precise analysis of the joint's parts deformation is required in order to provide for a very high accuracy analysis of the global mechanical system.

The design theory for bolted joint is based on linear stiffness analysis and a nonlinear one due to its external load after pretension [8].

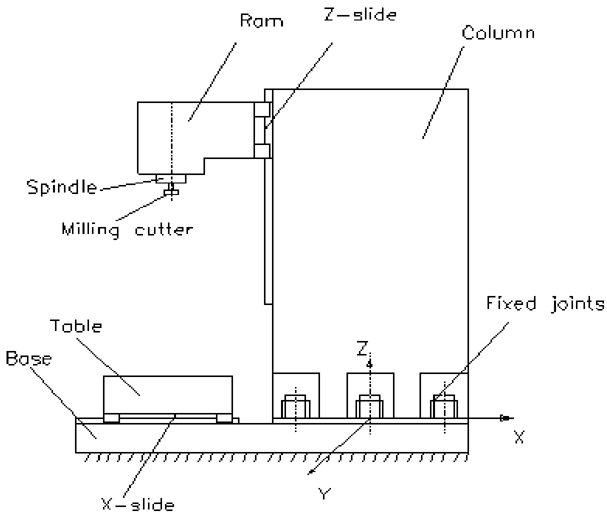


Fig. 1. Components of a milling machine.

Both the position and external load size have an influence on bolted joint deformation, and also the greatest influence has the size of the contact area between components [8].

Contact deformations between contacting surfaces in fixed and movable joints has played a significant role (frequently up to 50 percent) in the breakdown of deformations between various components of the machine tool structures. So far, it has been known that contact deformations are highly nonlinear and are due to surface imperfections on contacting surfaces [1].

A method that has been widely used in analyzing bolted joint structures and other mechanical structures is the finite element method (FEM) due to its convenience and reliable results [2].

2. LITERATURE APPROACH INFORMATION

Pederson [3] made a contact analysis using finite elements to calculate the elastic energy of both bolt and flanges parts. From the energies the author directly calculated the stiffness.

In some papers, bolted joints are treated in more general terms as complex contact problems.

[1] PhD Student, University "Politehnica" of Bucharest, Machines and Production Systems, , 313 Spl. Independentei, Bucharest, Romania, *E-mail: iuliana.piscan@yahoo.com.*
[2] PhD, Professor, University "Politehnica" of Bucharest, Machines and Production Systems, *E-mail:nicolaepredincea@yahoo.com.*
[3] PhD, Professor, North University of Baia Mare, Dr. Victor Babeş Str., no. 62 / A, Baia Mare, Romania,
E-mail: nic_pop2002@yahoo.com.

The deformable bodies systems solution lies in defining the contact areas of the main elements, the interaction forces and the tension field of elements.

Norbert Diekhans [4] calculated the bolted joint model implemented by Plock, showed in Fig. 2. In this model the contact area, the bolt and the flanges are represented by a pretensioned individual spring sistem.

The diagram from Fig. 3 shows the bolt elongation under the action of the pretension force F_v, the flange and in the contact area compression. The contact area presents a nonlinear load-deformation behavior. This non-linearity is reflected on the overall stiffness in the case of the working load, and becomes significant only at high work loads, where the bolt pretension is almost suppressed.

The clasical theory is based on the idea that the bolt and the sandwiched members of a given joint configuration can each be modeled as linear springs characterized by the stiffness k_b and k_f respectively, acting in parallel.

The spring stiffness of an elastic element such as bolt, nut, flange is the ratio between the acting force and the deformation caused by that force.

$$k = \frac{F}{\delta} . \tag{2}$$

The bolt stiffness, k_b, can be estimated in terms of the cross sectional area of the bolt, A_b, Young's modulus for the bolt, E_b, and the length of the bolt, L_b, as

$$k_b = \frac{A_b E_b}{L_b} , \tag{3}$$

Member stiffness can usually be obtained accurately only through FEM or experiments. If the material has the adequate thickness, then a pressure cone of compressed material in the shape of a frustum of a cone will be present and the average area should be considered.

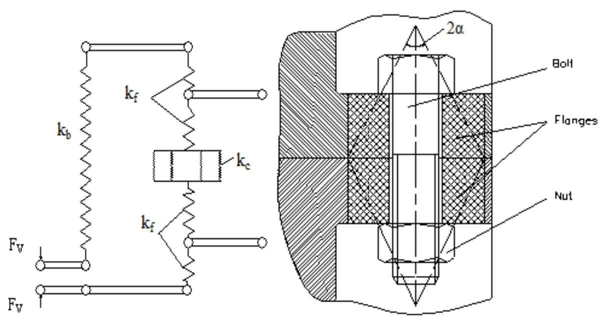


Fig. 2. The model elaborated by Plock [4].

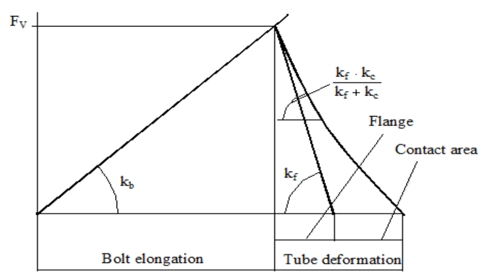


Fig. 3. Bolted joint stiffness diagram [4].

$$k_f = \frac{\pi E d \tan(\alpha)}{\ln\left[\frac{(2t\tan(\alpha) + d_b - d_h)(d_b + d_h)}{(2t\tan(\alpha) + d_b + d_h)(d_b - d_h)}\right]} , \tag{4}$$

where d_b is the bolt diameter, d_h is diameter of bolt head in contact with the clamped flange, t is thickness of the flange.

The value of the cone angle α assumed in the above relation has been the subject of investigation. For example, Osgood [5], Wileman et al. [6] have studied the load diffusion path at the pre-load stage, using the finite element approach and have demonstrated that the cone angle depends on the particular joint configuration.

Otherwise, if the material thickness is small, the compressed area can be assumed constant, in which case the compressive zone in the member forms a cylinder.

Therefore, the flanges stiffness is calculated with the formula:

$$k_f = \frac{A_f E_f}{L_f} , \tag{5}$$

where A_f is the area of flange, E_f − Young's modulus of the flange, and L_f − length of the flanges.

The total stiffness of the joint, k_j, can be computed assuming two springs in parallel as:

$$k_j = k_b + k_f . \tag{6}$$

For the contact stiffness computation the contact area between bodies and the force which acts on the surface can be calculated. Contact surface can be approximated within a cone bearing surface pressure, which is formed of thin flanges [7], represented by the dashed lines in Fig. 2.

The joint deformation is determined by applying an external load to the bolted joint. This deformation can be related with the bolt or flange stiffness by using the formula:

$$\delta = \frac{P_b}{k_b} = \frac{P_f}{k_f} , \tag{7}$$

where P_b is the portion of the external load carried by the bolt, P_f is the portion of the external load taken by the flanges.

Since the external tensile load is:

$$P = P_b + P_f , \tag{8}$$

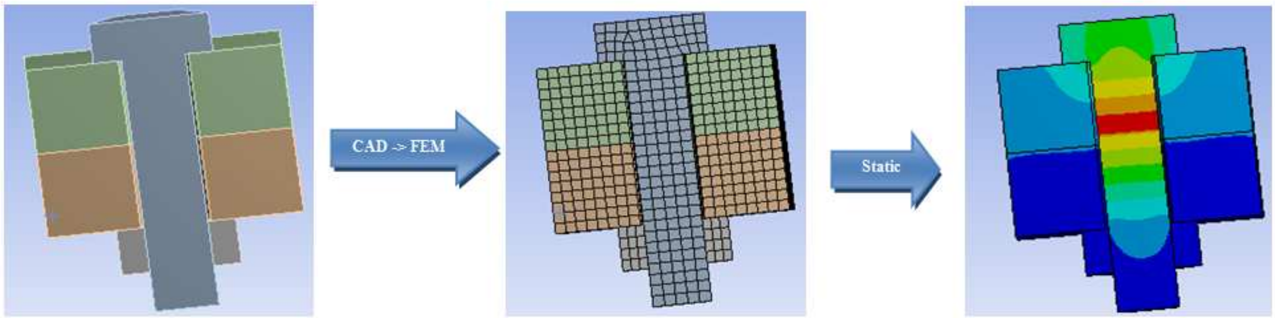
The resultant bolt load is:

$$F_b = P_b + F_v \tag{9}$$

and the resultant load on the flanges is:

$$F_f = P_f - F_v , \tag{10}$$

where F_v is the preload.



a b c

Fig. 4. The FEA analysis steps of a bolted joint: CAD model; b) meshed model; c) deformed model.

3. THE FEA ANALYSIS OF THE BOLTED JOINT

3.1. The FEA analysis procedure

The detailed finite element analysis for a bolted joint presented in Fig. 4 is exemplified in the following phases:

- The first phase is modeling the joint using CAD software. The model geometry was generated using CATIA software and then imported as a neutral file in ANSYS WORKBENCH. Due to symmetry conditions the model is sectioned. Geometric details, such as chamfers, radii of connection have only a local influence on behavior of the structure therefore those are neglected. In this analysis we neglect the bolt thread and surface roughness.
- Next, the prepared geometric structure is reproduced by finite elements. The finite elements are connected by nodes that make up the complete finite element mesh. Each element type contains information on its degree-of-freedom set (e.g. translational, rotational, thermal), its material properties and its spatial orientation (1D-, 2D-, 3D-element types). The mesh was controlled in order to obtain a fine and good quality mapped mesh. The assembly had 19 117 nodes and 5 420 elements.
- In order to solve the resulting system equation, boundary and loaded conditions are specified to make the equation solvable. In our model, the lower side of the bottom flange was fixed and five different bolt axial load were applied.

  The pretension in the bolt was generated at the midplane of the bolt using the pretension element PRETS179, which is contained in the ANSYS v11 element library. These elements allow direct specification of the pretension in bolt. For specifying the bolt pretension a local coordinate system was defined, with the Z axes along the bolt length.

  After the bolt pretension, an external load was applied to the bolted joint.
- The last phase is interpreting the results.

For contact analysis ANSYS supports three contact models: node to node, node to surface and surface to surface. In this case a surface to surface model was created and contacts elements were used.

ANSYS provides several element types to include surface-to-surface contact and frictional sliding. One of these elements is the 3D 8-node surface-to-surface contact element CONTAC174.

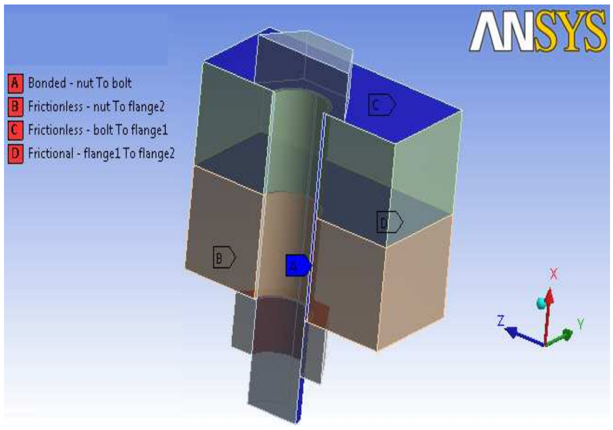


Fig. 5. Contact between surfaces.

Contacts elements use a target surface and a contact surface to form a contact pair. According to stiffness behavior the parts can be rigid or flexible. Our model is defined as flexible one.

The connection types between parts are defined in Fig. 5.

3.2. Stiffness analysis of the bolt part

The bolt stiffness k_b is often determined by simple calculations, approximating the deformation of the bolt head and of the nut. Assuming uniform tensile stress σ and uniform strain $\varepsilon = \sigma/E$ for a total contact pressure force P with a uniform bolt cross-sectional area A_b, the authors obtained for the total elastic energy U_b (sum of the elastic strain energy U_ε and the elastic stress energy U_σ in the bolt) [2].

$$U_b = U_\varepsilon + U_\sigma = \sigma\varepsilon\overline{V} = \frac{\sigma^2\overline{V}}{E} = \frac{P^2 A_b \overline{L}}{A_b^2 E} = \frac{P^2 \overline{L}}{EA_b}, \quad (11)$$

where $\overline{V}$ is the estimated volume obtained with the estimated length $\overline{L}$ that also account for the stiffness of bolt head and nut. In terms of a reference displacement υ_P corresponding to the total force P, the bolt stiffness is

$$k_b = \frac{P}{\upsilon_P} = \frac{P^2}{P\upsilon_P} = \frac{P^2}{U_b} = \frac{EA_b}{\overline{L}}, \quad (12)$$

using (11) to obtain the final expression.

3.3. Stiffness analysis of flange members

The stiffness of the flanges members is defined similar to (11)

$$k_f = \frac{P}{\upsilon_m} = \frac{P^2}{P\upsilon_m} = \frac{P^2}{U_m}, \tag{13}$$

but without the simple approximation of the total elastic energy [2].

The contact problem of two flanges must be solved iteratively, but this is a relatively simple contact problem. The contact diameter d_c must be determined such that only compressive contact forces act between the flanges. In [2] is given an initial estimate of the contact diameter d_c:

$$d_c \cong d_h + L\tan(\alpha). \tag{14}$$

In traditional finite element analysis, detailed information on displacements, strains, stresses, energy densities and general stiffness are obtainable if the contact pressure distribution $p = p(r)$ is available. Yet, it may be rather complicated iteratively to determine this, but is possible with most FE programs.

3.4. Contact pressure distribution with pretension only

The distribution of the contact pressure [8] can be obtained in a direct procedure, due to the influence on the global stiffness quantities, especially for components. The bolt analysis is carried out as a super element FE procedure

$$[S_{bp}]\{D_{bp}\} = \{F_{cp}\} \rightarrow \{D_{bp}\} = [S_{bp}]^{-1}\{F_{cp}\}, \tag{15}$$

where $[S_{bp}]$ is the bolt super element stiffness matrix of order equal to the number of nodes at the bolt-member contact surface, with degrees of freedom in the axial z-direction. The resulting corresponding displacements are $\{D_{bp}\}$ and the corresponding nodal contact pressure forces are $\{F_{cp}\}$. The sum of these forces is the total contact pressure force P

$$P = \|\{F_c\}\|. \tag{16}$$

The flange analysis may also be carried out as a super finite element procedure

$$[S_{fp}]\{D_{fp}\} = \{F_{cp}\} \rightarrow \{D_{fp}\} = -[S_{fp}]^{-1}\{F_{cp}\}, \tag{17}$$

where $[S_{fp}]$ is the flange super element stiffness matrix also of order equal to the number of nodes at the bolt-member contact surface, with degrees of freedom in the axial z-direction. The resulting corresponding displacements are $\{D_{fp}\}$ and the corresponding nodal contact pressure forces are $-\{F_{cp}\}$, measured in the z-direction. The pretension displacement is described by the mutual displacements $\{D_0\}$, given as a translational constant vector when no initial gap exists

$$\{D_0\} = e\{1\}, \tag{18}$$

where e is a positive displacement and the one vector $\{1\}$ has the value 1 in all components.

Contact condition after pretension is then

$$e\{1\} = \{D_{bp}\} - \{D_{fp}\} = \left([S_{bp}]^{-1} + [S_{fp}]^{-1}\right)\{F_{cp}\}, \tag{19}$$

or solved in terms of the contact forces

$$\{F_{cp}\} = \left([S_{bp}]^{-1} + [S_{fp}]^{-1}\right)^{-1} e\{1\}. \tag{20}$$

The contact conditions imply non-negative contact forces $\{F_{cp}\} \geq \{0\}$, and this can in fact be proved to be the case. The matrix $[K] = \left([S_{bp}]^{-1} + [S_{fp}]^{-1}\right)^{-1}$ is a stiffness matrix, which is positive definite and strictly diagonal dominant. From this follows for each row (or column) in $[K]$ that $\sum_{j=1}^{n} K_{ij}$ for all i, and therefore $\{F_{cp}\} > \{0\}$. Thus the contact problem for the pure pretension state has a solution with only positive contact pressure forces.

3.5. FEA contact formulation

There are several types of mathematical formulation of the contact problem [9], which exists in the normal and tangential direction of the surface contact.

Penalty function. Penalty function is a displacement-based solution:

$$[K]\{x\} = \{F\}. \tag{21}$$

Penalty function manages contact by adding springs to model at each element Gauss points.

It deals with contact stiffness and penetration:

$$K_{contact}\Delta x_{penetration} = F_{contact}. \tag{22}$$

Since surface-to-surface contact transmits contact pressure between Gauss points, and not forces between nodes, contact stiffness is in [*Force/Length*^{3}] units. Penetration is a mathematical formulation, since it never occurs in reality between two contacting bodies. Penetration exists to assure that the contact force is not 0. Even if very small (e.g. 0.1 µm), penetration influences the solution. Thereby, to obtain a converged contact stress, penetration must be as small as possible. It can be obtained by increasing contact stiffness as much as possible. But too high contact stiffness will produce ill-conditioned system matrix, with very high ratio in rigidity terms of the system matrix. For direct solvers it does not cause problem.

Contact stiffness (normal or tangential) can be input as an absolute value (KN or KT) or a factor (FKN or FKT) to the default Hertz contact stiffness (KH). KH depends of material rigidity and mesh size.

Lagrange multipliers. With Lagrange multipliers, contact forces are treated as a separate DOF:

$$[K^*]\begin{pmatrix} x \\ F_{contact} \end{pmatrix} = \{F\}, \tag{23}$$

Software solves directly for contact forces (or pressure). Therefore, Lagrange multipliers add equations to model and increase computational cost. It also introduces zero diagonal terms in the system matrix, resulting in a limited solver selection (direct solvers only). Direct solvers are more efficient with small models, but sometimes need important computer resources to run large models.

Lagrange multipliers have the advantage of not dealing with contact stiffness. Penetration still exists, but it essentially depends on mesh size since a finer mesh corresponds to more contact detection points.

Augmented Lagrangian. Among contact formulations available, Augmented Lagrangian is a Penalty function with penetration tolerance (FTOLN) and allowable elastic slip (SLTO – or tangential penetration). The sliding result is the sum of physical sliding and mathematical sliding (allowable elastic slip). When converged, the mathematical sliding is negligible compare to physical sliding. Augmented Lagrangian contact works as following: after each iteration, when penetration or sliding exceeds a limit, contact stiffness is automatically increased with Lagrange multipliers.

3.6. FEA results

The reduced model bolted joint simulations were carried out in order to study its behavior under the action of external loads taking into account as parameters the normal stiffness, the pretension force and friction coefficient. The influence of these parameters on the deformation of the bolted joint is shown in Figs. 6, 7, and 8. To study the influence of pretension force to the bolted joint we pretension the bolt ranging from a minimum force of 1000 N and a maximum force of 5000 N.

Since we considered a contact with friction between the flanges, was taken into account the friction coefficient as design variable contributing to the increase of deformations in bolted joint.

Due to the importance of contact stiffness parameter we varied the contact stiffness value to find out the maximum value of stiffness in which bolted joint deformation remains constant as shown in Fig. 8.

Although the charge of the structure is linear the response of the structure is non linear. These results were verified by numerical calculations and compared with theoretical results.

The influence of pressure on bolted joint deformations, presented in Fig. 9 shows a linear behavior of bolted joint but it hasn't a negligible influence on deformations.

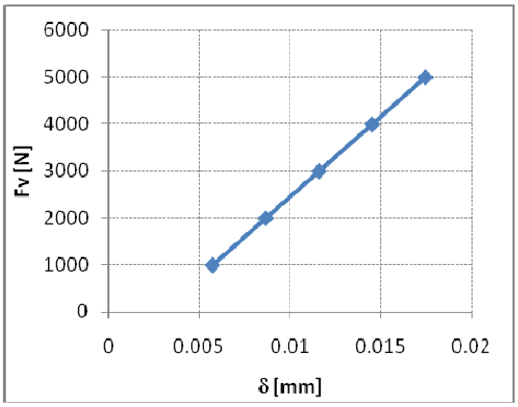


Fig. 6. Dependence pretension force-deformation.

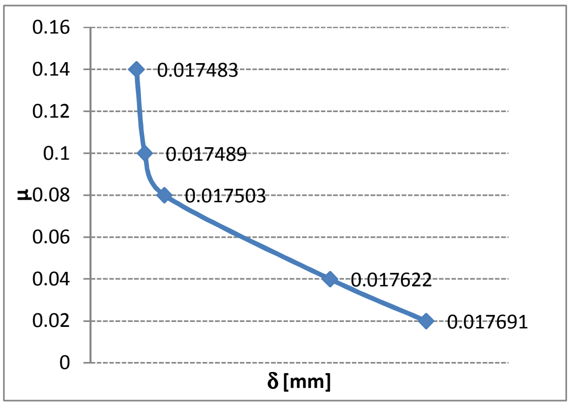


Fig. 7. Dependence friction coefficient-deformation.

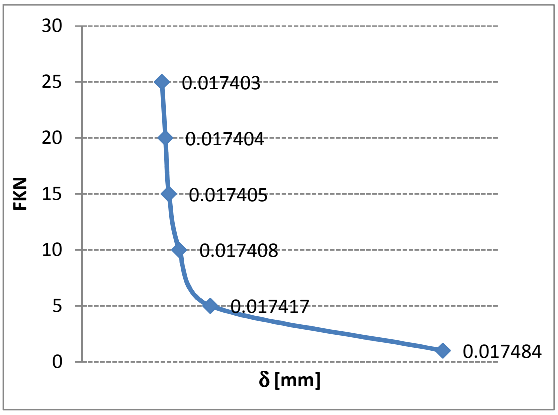


Fig. 8. Variation of contact stiffness factor.

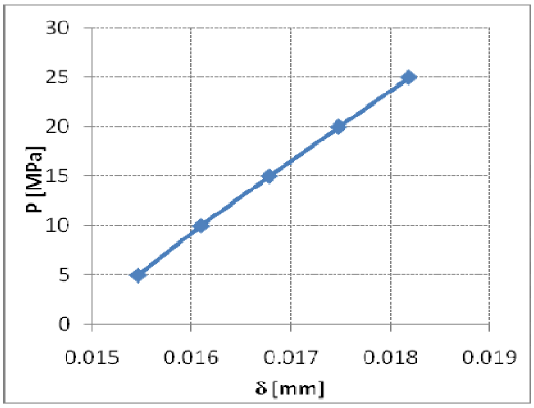


Fig. 9. Dependence pressure-deformation.

Contact deformations between joint components are represented in Fig. 10. The contacts between bolt and nut and between bolt head and flange have the same deformations and, also have the most significant deformations.

The contact status is shown in Fig. 11. Far means the elements from contact and target do not touch each other. Near means the elements are almost touching each other. Sliding means the contact elements slide on the surface of target surface. Sticking means the contact elements can not move and penetration happens.

Penetration represented in Fig. 12, exists in the contact between the bolt head and flange and in the contact between bolt and nut. The value of penetration is small enough.

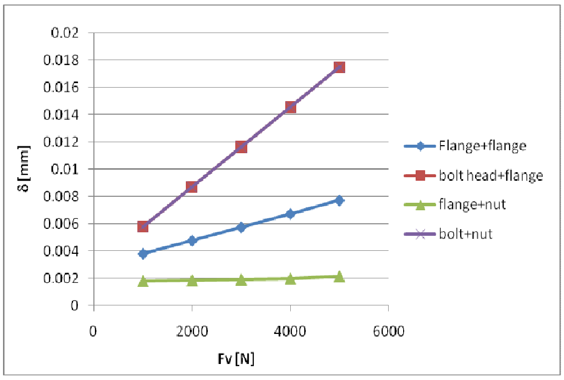


Fig. 10. Dependence pretensioning force and contact deformations between components.

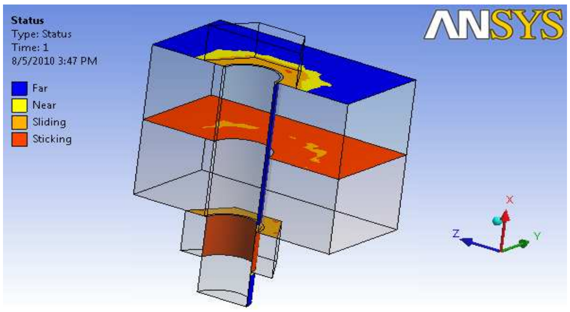


Fig. 11. Contact status.

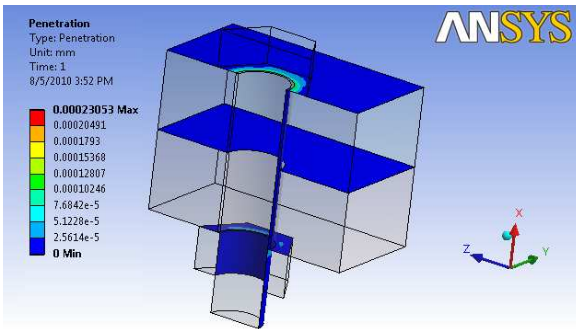


Fig. 12. Contact penetration.

4. CONCLUSIONS

Defining joints is one of the most difficult aspects when simulating the behavior of machine-tools, because there are many variables that can affect the joint's properties. By using finite element analysis software we can optimize the design process of machine tool components by identifying the parameters that has a influence on the static behavior of machine tools. By analyzing the results obtained in the post-processing phase, the user can evaluate the properties of machine tools still in the design stage, without the need to make prototypes.

Based on these preliminary results of bolted joint contact deformation analysis further research will be carried on the model optimization.

By knowing the influence of the parameters on the contact deformation we can optimize the stiffness of the structural components of machine tools.

In this paper we considered as parameters:

- Bolt pretension force supports a maximum value of 5000 N due to the maximum stress that the bolt can be subjected to;
- We started from the default stiffness factor as generated by the ANSYS Workbench software and increased to the point that the value of the deformation remains constant. We found that the contact stiffness has a non-linear variation;
- We considered a friction coefficient between the flanges of 0.02 minimum and a maximum of 0.14, which is a non-linear variation because increasing the friction coefficient will cause a decrease of the bolted joint deformation.

5. ACKNOWLEDGEMENTS

The work has been funded by the Sectoral Operational Programme Human Resources Development 2007-2013 of the Romanian Ministry of Labour, Family and Social Protection through the Financial Agreement POSDRU/88/1.5/S/60203.

REFERENCES

[1] E.I. Rivin, *Machine tool vibration*, Marcel Dekker, Inc., New York, 1999.

[2] J. Yoo, S.-J. Hong, J.S. Choi, Y.J. Kang, *Design guide of bolt locations for bolted-joint plates considering dynamic characteristics*, Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, Volume 223, Number 2 / 2009, pp. 363−375.

[3] N.L. Pederson, P. Pederson, *On prestress stiffness analysis of bolt-plate contact assemblies*. Applied Mechanics, Vol.78, (2008), pp. 75−88.

[4] N. Diekhans, *Kleinrechnereinsatz für Berechnungsaufgaben im Konstruktionsbereich: Berechnung von Werkzeugmaschinen* (Small computer application for calculation tasks in the construction field: Calculation of machine tools), Edit. City-Print Verlag Offsetdruck GmbH, Aachen, (1980), Aachen.

[5] C.C. Osgood, *Saving weight on bolted joints, machine design*; 1979 October.

[6] J. Wileman, M. Choudhury, I. Green, *Computation of member stiffness in bolted connections*. ASME Journal of Mechanical Design 1991;113:432–7.

[7] Z.M. Levina, *Research on the Static Stiffness of Joints in Machine Tools* Proceedings of the MTDR Conference, Manchester, (1967).

[8] N. L. Pederson, P. Pederson, *Bolt–plate contact assemblies with prestress and external loads: Solved with super element technique*. Computers and Structure, Vol.87, (2009), pp. 1374−1383.

[9] *** *ANSYS Theory Manual*, Version 11.0, 2007.