RESEARCH PAPERS
FACULTY OF MATERIALS SCIENCE AND TECHNOLOGY IN TRNAVA
SLOVAK UNIVERSITY OF TECHNOLOGY IN BRATISLAVA

2016 Volume 24, Number 39

APPLICATION OF NUMERICAL SIMULATION FOR THE VERIFICATION OF DESIGN OF CLAMPING A WORKPIECE IN JAWS

Jarmila ORAVCOVÁ, Eva RIEČIČIAROVÁ

SLOVAK UNIVERSITY OF TECHNOLOGY IN BRATISLAVA,
FACULTY OF MATERIALS SCIENCE AND TECHNOLOGY IN TRNAVA,
INSTITUTE OF APPLIED INFORMATICS, AUTOMATION AND MECHATRONICS,
ULICA JÁNA BOTTU 2781/25, 917 24 TRNAVA, SLOVAK REPUBLIC
e-mail: jarmila.oravcova@stuba.sk, eva.rieciciarova@stuba.sk

Abstract

This paper deals with the use of the numerical computer oriented method for the analysis and verification of the design of workpiece clamping. The inaccuracies of the positioning in clamping locating points were simulated using the FEM software ANSYS. The influence of inaccuracies in the positions of contact locating points on the displacements of the workpiece characteristic points and their position in clamped removable jaws was examined. The reaction forces in the contact locating points were also computed and analyzed.

Key words

contact locating points, FEM method, simulation, displacement, reaction force

INTRODUCTION

The technological operations of workpiece clamping are a part of workpiece manufacturing. Clamping stiffness and speed have a significant impact on the workpiece quality, duration of the manufacturing cycle and also on the manufacturing cost (1). Correctly designed clamping elements represent one of the notable influences on the productivity and on the quality of the machined area (2).

The design of a clamping element requires the correct construction and positioning of contact elements and selection of the clamping jaw type (3, 4). The direction and size of reaction forces result from the design. The analysis and verification design of clamping components using a numerical computer focused method allows for the elimination of errors in the design before putting the clamping component into production. In this paper, the influence of the positioning of contact locating points on the displacements of characteristic points and the magnitude of reaction forces was investigated.

THEORETICAL BACKGROUND

The simulation program code ANSYS can be applied for the solution of a wide variety of technical problems using the finite element method (FEM). In contact tasks, energetic formulation is used. For each element, the functional Π^e defining the total potential energy of an element is given by the relationship (5)

$$\Pi^e = \Pi^e(u) = A^e - W^e \,, \qquad [1]$$

where u is the vector of nodal displacements, A^e represents the total strain energy given by the formula

$$A^e = \frac{1}{2}\int_{V^e} \{\varepsilon\}^T \{\sigma\} dV \qquad [2]$$

and W^e is the work of external forces acting on the element which can be computed from the relationship

$$W^e = \int_{V^e} \{u\}^T \{\gamma\} dV + \int_{A_p^e} \{u_p\}^T \{p\} dA + \sum \{u_i\}^T \{F_i\} \,, \qquad [3]$$

where u_p - is the vector of nodal displacements on the areas with applied superficial pressure,
u_i - the vector of nodal displacements in which the concentrated forces are imposed,
p - the vector of applied pressure,
γ - the vector of volumetric forces,
ε - the vector of nodal strains,
σ - the vector of nodal stresses,
F_i - the vector of concentrated nodal forces.

For each element, a functional Π^e expressing the total potential energy of an element is defined as

$$\Pi^e = \frac{1}{2}\int_{V^e} \{\varepsilon\}^T \{\sigma\} dV - \int_{V^e} \{u\}^T \{\gamma\} dV + \int_{A_p^e} \{u_p\}^T \{p\} dA + \sum \{u_i\}^T \{F_i\} \,. \qquad [4]$$

Including the internal nodal forces $\{q^e\}$, the total potential energy of an element can be formulated in matrix form [5]

$$\Pi^e = \frac{1}{2}\{u^e\}^T [K^e] \{u^e\} - \{u^e\}^T \{f^e\} - \{u^e\}^T \{F^e\} \,, \qquad [5]$$

where $[\boldsymbol{K}^e]$ is the stiffness matrix of the element e,
$\{f^e\}$ is the vector of external nodal forces.

Assembling the resulting system of equations for the whole body leads to the expression for the total potential energy of the system in the form

$$\Pi = \sum_{i=1}^{n_0^e} \Pi^{ei} = \sum_{i=1}^{n_0^e} \left(\frac{1}{2} \{u^{ei}\}^T [K^{ei}] \{u^{ei}\} - \{u^{ei}\}^T \{f^{ei}\} \right). \qquad [6]$$

The solution for vector field of nodal displacements corresponds to the minimum of the total potential energy. Then the task is to find the field of nodal displacements u, fulfilling the given boundary conditions and loading that minimizes the functional [6]

$$\frac{\partial \Pi^e}{\partial u_i} = 0. \qquad [7]$$

Minimizing the potential energy according to the nodal displacements leads to the system of algebraic equations

$$\frac{\partial \Pi^e}{\partial u^e} = [K^e]\{u^e\} - \{f^e\} - \{F^e\} = 0 \qquad [8]$$

from which the unknown values of nodal displacements can be computed.

PROBLEM DESCRIPTION AND SIMULATION MODEL

The finite element program code ANSYS was used for the development of the simulation model. Geometrical shape and dimensions of the investigated prismatic workpiece are shown in Fig. 1a. It was considered to clamp the workpiece with removable jaws. The real configuration of locating elements was simplified to the contact locating points 1, 2 and 3 in which the spherical locating elements toughen the workpiece (Fig. 1a, b). The finite element mesh was created by the 3D element SOLID 185 (Fig. 1b). The elements CONTA173 and TARGE170 were applied to take into account the contact between locating elements and the workpiece (Fig. 2). For the workpiece, the elastic-plastic material model with bilinear stiffening was applied with the following values: the Young modulus E = 200 GPa, the Poisson ratio ν = 0.3, the yield stress R_e = 210 MPa and the tangent modulus E_1 = 10000 MPa. Clamping jaws were in the simulation replaced by clamping forces F_1 = 1000 N and F_2 = 1000 N acting in the nodes according to Figs. 1a, b.

Three different cases of locating element positioning with distinct inaccuracies (Table 1) were simulated. In the first simulation (CASE 1), the positions of the contact local points were considered to be in prescribed exact locations without any deviations towards the workpiece in the y-axis direction $u_{1y} = u_{2y} = u_{3y} = 0$.

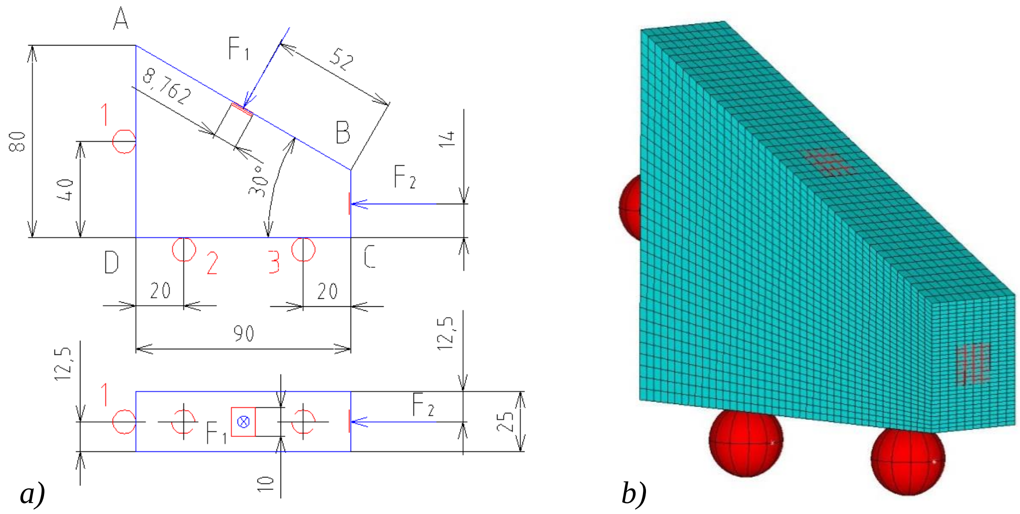


***Fig. 1** Geometrical model (a) and finite element model with defined clamping forces (b)*

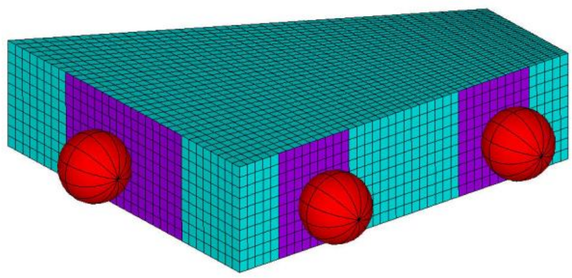


***Fig. 2** Definition of the contact elements*

Table 1 Position of the contact locating points

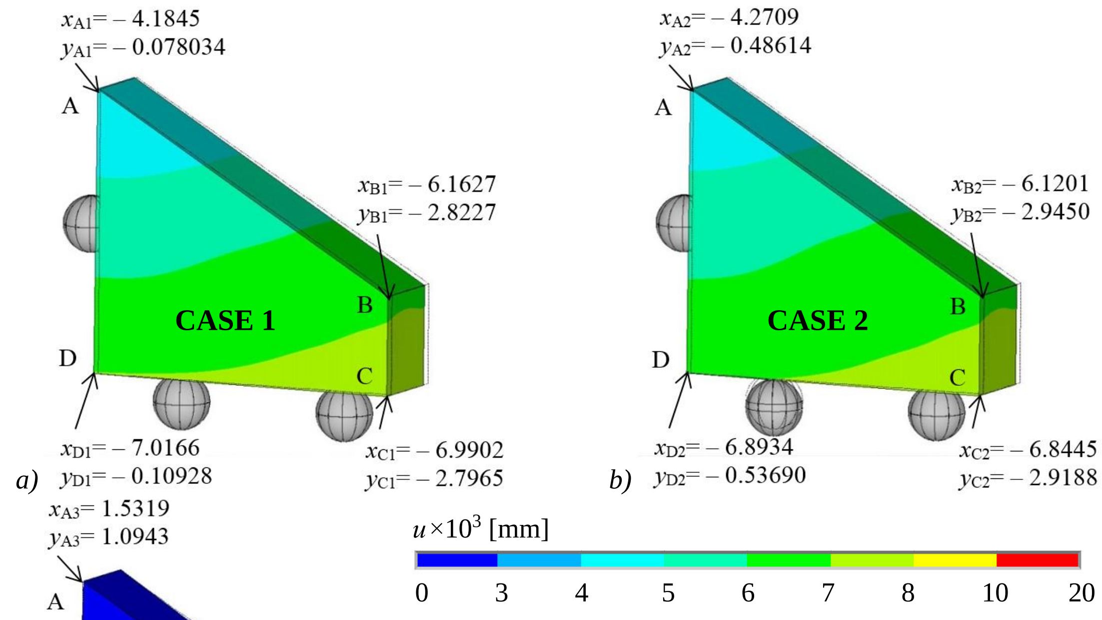
Locating point number	Deviation in y-axes direction [mm]		
	CASE 1	CASE 2	CASE 3
1	0	0	0
2	0	– 0.01	0
3	0	0	– 0.01

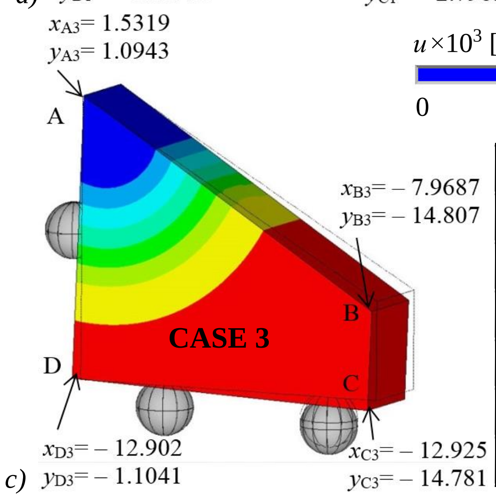
In the second simulation (CASE 2), the contact locating point 2 was shifted from its prescribed position to the position with the deviation of $u_{2y} = -0.01$ mm in the y-axis direction away from the workpiece. In the third simulation (CASE 3), the inaccuracy in the positioning of the contact locating point 3 was investigated taking into account the movement of this contact locating point down in the y-axis direction with the value of $u_{3y} = -0.01$ mm.

RESULTS

Within the ANSYS environment, the displacements of characteristic points of the investigated workpiece were analyzed using the described simulation model. Displacements of the characteristic points of the workpiece marked with letters A, B, C, D (Fig. 1a) are shown in Fig. 3 for the cases from 1 to 3. The total displacements are plotted using color contours. The computed displacements of the characteristic points A, B, C, D in the x-axis and y-axis directions are shown by their values for the single cases in Figs. 3a, b, c.

The results for CASE 2 indicate only small displacements which are comparable with the displacements computed for CASE 1, representing the prescribed positioning of the contact locating elements. The slightly higher values of the displacements were found for the characteristic points A and D in the y-axis direction. In CASE 3, the unfavorable state occurred. The displacements of the characteristic points B, C, D were of one order higher in comparison with CASE 1. Contact location point inaccuracies had an influence also on the change of the size of reaction forces. The values of reaction forces are shown in the Fig. 3 table.





CASE	REACTION	POINT 1	POINT 2	POINT 3
1	R_x [N]	1327.0	7.3	166.3
	R_y [N]	55.1	123.0	687.8
2	R_x [N]	1326.0	0	174.3
	R_y [N]	145.2	0	720.9
3	R_x [N]	1329.0	22.0	149.4
	R_y [N]	-131.2	377.2	620.0

***Fig. 3** Displacements of characteristic points $u\times10^3$ [mm] and reactions for investigated cases*

DISCUSSION

It was supposed that in CASE 2 and CASE 3, the workpiece is under the imposed clamping forces in contact with each contact locating point. As it follows from the numerical simulation in CASE 2, the reactions at the contact locating point 2 are zero, which means that the workpiece is not in contact with the clamping system at the contact locating point 2. On the other hand, the displacements of the workpiece were relatively small and comparable with the displacements computed for the "ideal" (designed) positioning of clamping system corresponding to CASE 1.

The most unfavorable conditions of workpiece clamping occurred in CASE 3 when the contact locating point 3 was shifted by 0.01 mm down the *y*-axis direction. In this case, the maximum total displacements of the workpiece reached the value of 0.02 mm.

For comparison and verification of the total reaction forces, the reactions at the contact locating points 1, 2 and 3 were calculated analytically taking into account the workpiece as the 2D solid body. Using this approximation, the calculated reaction R_x in the *x*-axis direction was 1500 N. The sum of reactions R_y in the *y*-axis direction at the locating points 2 and 3 was 866.03 N. The results obtained by numerical simulation and presented in the Fig. 3 table, are in very good agreement with the values of reaction forces calculated analytically.

CONCLUSION

The application of numerical simulation for the analysis and verification of workpiece clamping design, enabled the use of different forms of clamping. The influence of inaccuracy in the positioning of contact locating points on the displacements of characteristic points of the workpiece was investigated. It was found that shifting the contact locating point 2 in the *y*-axis direction exhibited only a small deviation from the positioning prescribed in the clamping design and corresponding to the investigated CASE 1. Inaccuracy in the positioning of the contact locating point 3 resulted in quite large displacements of the workpiece which could lead to a decrease in the quality of the machined areas. As it follows from the obtained results, the accuracy of the clamping system positioning affected also the reaction forces in the contact locating points.

Finally, it can be concluded that numerical simulation of the workpiece behaviours under different conditions of its clamping can provide very useful information and it can be applied for verification of the position of workpiece machined areas clamped in various types of clamping devices.

Acknowledgement

The authors wish to thank for financial support to the VEGA project No. 1/0876/15 and the MTF institutional project No. 2016/006.

References:

1. MATÚŠOVÁ, M., JAVOROVÁ, A., 2008. Modular clamping fixtures design for unrotary workpieces. *Annals of Faculty of Engineering Hunedoara - Journal of Engineering,* Tom VI, Fasc 3, pp. 128-130. ISSN 1584-2673.
2. KALOCAY, R., BERGS, T., KLOCKE, F., 2014. Impact of Clamping Technology on Horizontal and Vertical Process Chain Performance. In: *New Production Technologies in Aerospace Industry,* 10.1007/978-3-319-01964-2-2.

3. Yanchao Deng., 2011. A Theoretic Study on Cam-Clamping Device. *Manufacturing Science and Technology*, PTS 1-3, Vol. 295-297, pp. 1536-1539.
4. Jiao, Zhiwei, 2015. Clamping Characteristics Study on Different Types of Clamping Unit. In: *Proceedings of PPS-30*, Vol. 1664.
5. BENČA, Š., 2009. *Riešenie nelineárnych pevnostných úloh pomocou MKP. (FEM solution of nonlinear strength problems.)* Bratislava: STU. ISBN 978-80-227-3077-8.

ORCID:
Jarmila Oravcová 0000-0002-2414-8064