International Journal of Advancements in Mechanical and Aeronautical Engineering
Volume 3 : Issue 1 *[ISSN 2372-4153]*
Publication Date : 18 April, 2016

Load Simulation of Forging Press Upgraded by Tool Holder

[Jan Hlavac, Miroslav Bele]

***Abstract*— This article deals with a virtual simulation of forging press made with finite element method. The aim is to extend simulation model about a toll holders and tools. We analyzed differences between results of stress load of press' frame.**

***Keywords*— forging press, load simulation, FEM**

I. Forging press' load simulation

Forging presses are mechanical crank presses used for closed die forging. Those presses are designed to withstand enormous forging forces. Frame strength is the most important parameter but stiffness is important too.

Old one presses' frames were designed with excessed capacity castings. Modern trends are to design press frame capable to withstand negligible overloads and with emphasis to low costs. To do so, it is important to perform detailed design proposal. Todays' the most important design tool is virtual simulation with use of FEM (Finite Element Method). Virtual simulation is dependent on software and hardware capacity and on designer experience.

There is a research center of Forming Machines Design in University of West Bohemia where we support Czech producers. We deal with design of forging presses for more than ten years, therefore we can describe progress of virtual simulation.

A. *Development of press' simulation*

Press' frames were designed with knowledge of mechanic of solids in the past. In the late nineties time of virtual simulation begun. We used simulation of alone frame loaded by determined reactions at the beginning. It was sensitive to proper reactions' determination but it was only way with the then hardware and software capacity.

Doc. Ing. Jan Hlavac, Ph.D.
Department of Machine Design / Faculty of Mechanical Engineering /
University of West Bohemia
Czech Republic, Europe

Ing. Miroslav Bele
Department of Machine Design / Faculty of Mechanical Engineering /
University of West Bohemia
Czech Republic, Europe

Than a contacts task (contact between two or more bodies) occurred. Contact simulation was a significant improvement of virtual simulation actuality because exclusion of reactions' determination. Contact simulation demands higher computer capacity because of several iterations. Hardware and software capacity grows therefore simulation models are more and more complex.

This article represents influence of added tool holder and tool into a simulation. Load forces were applied directly on press table and slide before this change.

II. Virtual simulation

Model of the press was extended by tool holders and tools. Model consists of press' frame, crank shaft, connection rod, slide shaft, slide, table desk, tools holders and tools. Tools holders and tools are united into two bodies (bottom and lower). The crank shaft position is 15° before bottom dead center.

The simulation of original model without tool holders and tools was performed too. Simulation conditions are the same.

A. *Finite element mesh*

The finite element mesh is made by quadratic tetrahedrons. Element size is 64 mm for all bodies. Contact surfaces are enhanced to 32 mm size.

Material of all bodies is a steel with:

- Modulus of elasticity $E=2.0694 \cdot 10^{11}$ Pa
- Poisson ratio $\nu=0.288$
- Material is simply elastic

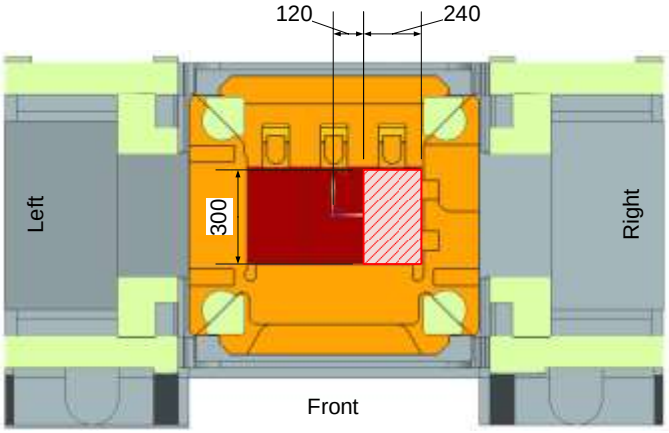

Figure 1. Top view of bottom tool and its holder (force acts on hatched area).

B. Boundary conditions

Load force acts on a right tool (upper and bottom) with 240 mm eccentricity (see Fig. 1). Load force acted directly on table and on slide in prior simulation.

The press model is constrained by part of bottom plates' surface (all DOFs are banned).

The crank shaft is constrained at the rear end. Rotation around its axis is banned.

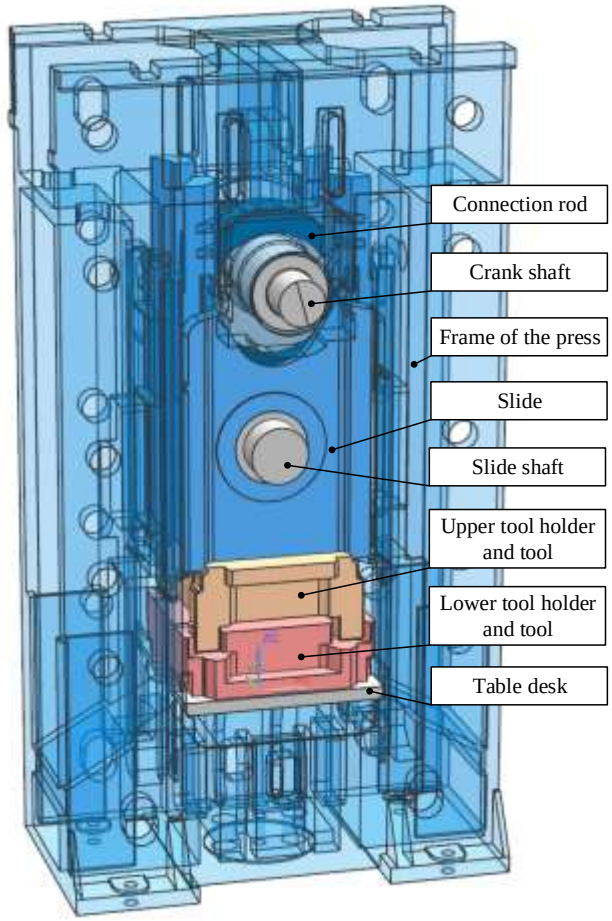

Figure 2. Virtual simulation model

C. Contact task

There are individual contact pair in the model:

1. Press frame – Crank shaft
2. Crank shaft – Connection rod
3. Connection rod – Slide shaft
4. Slide shaft – Slide
5. Slide – Press Frame
6. Slide – Upper tool holder
7. Press frame – Table desk
8. Table desk – Lower tool holder
9. Tool guide

Friction coefficient is set to f=0.08 or majority of contact pairs expect for pairs 6. – 8. where friction coefficient is set to f=0.15 and expect for pair 9. where friction coefficient is set to f=0.

There is a free play 0.2 mm in contact pair 5 and 0.1 mm in contact pair 9.

The tool holders and table desk are on real machine connected to its neighbor via nuts. In our simulation they are connected via gluing performed in the middle of contact area.

III. Results of virtual simulation

Results of virtual simulations (prior and current) are compared.

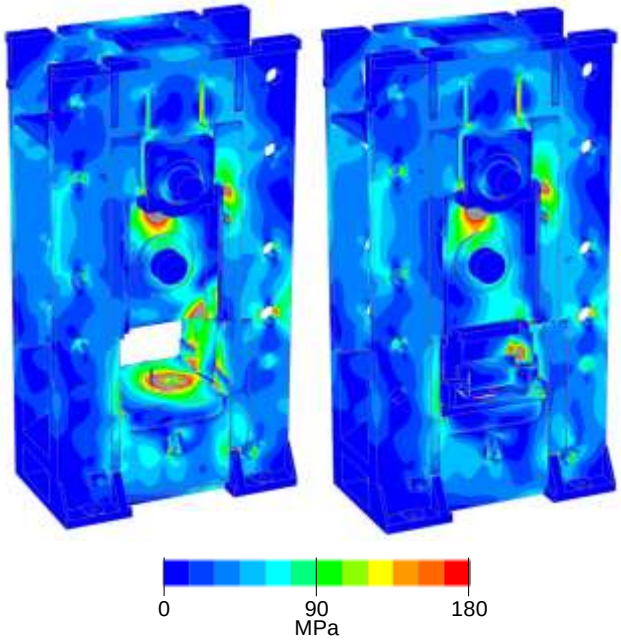

Figure 3. Overview of stress results of all model (left: prior model, right: current model; all bodies; von Mises)

There are stress results of both models on Fig. 3. It can be seen, that the stress distribution is similar around the frame expect for bottom part. Stress distribution on frame differs too.

The main goal of this article is to describe differences in the frame load between model without and with tool holders. To obtain correct results it is necessary to compare comparable data. Therefore important places are described on the frame (see Fig. 4 and Fig. 5). For those important places stress values were read (see Tab. 1).

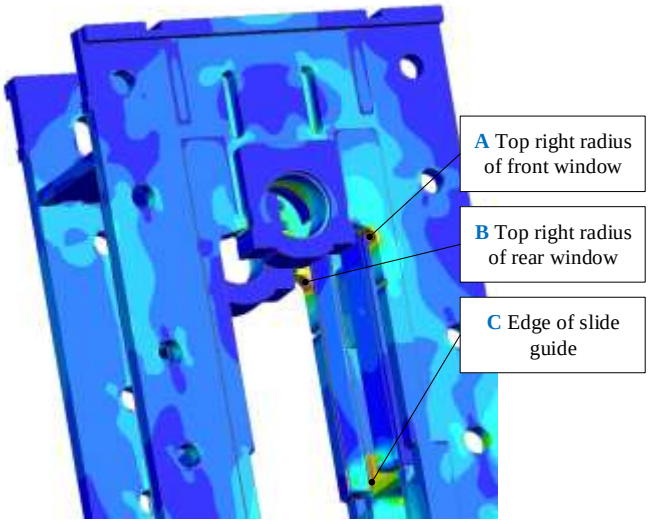

Figure 4. Important places – frame exterior

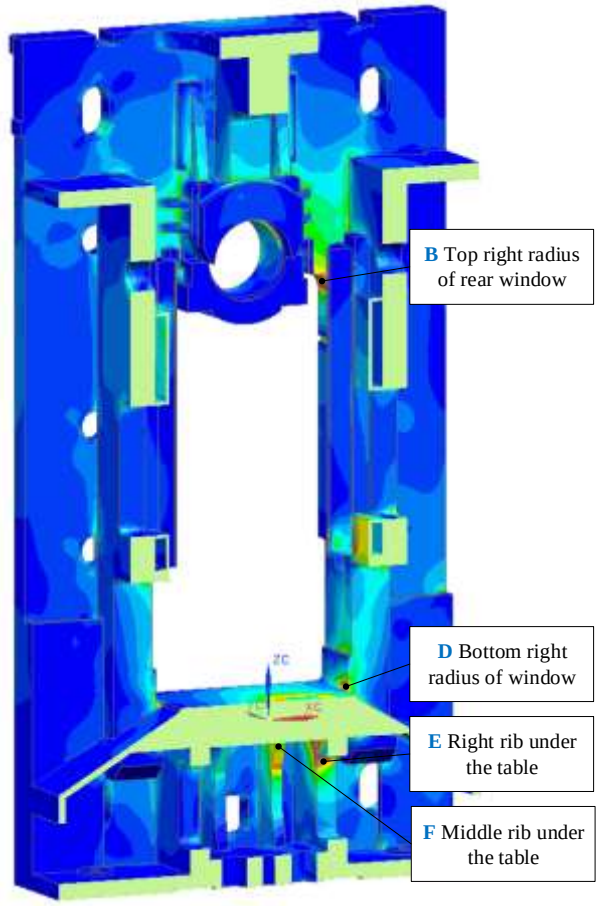

Figure 5. Important places – frame interior

It is obvious that upper part of press' load simulation was not influenced by adding of tool holders with tools, as was expected.

TABLE I. STRESS VALUES OF IMPORTANT PLACES

Place	Prior model	Current model
A	214	214
B	227	224
C	270	240
D	247	281
E	292	132
F	303	67

Considerably different situation is bellow press table. Stress values are lower for positions E and F in ribs under the table. Stress values decreased from 300 MPa to 132 MPa and even to 67 MPa for right rib. Explanation of stress decrease is in high stiffness of tool holder what distribute load on broader part of table. Unwanted side effect of tool holder can be seen in increasing of stress in place D where the table is connected to stand.

IV. Conclusion

The main aim to obtain more realistic results of stress distribution in press frame was fulfilled. Newly added tool holders with tools relevantly changed it in ribs under the table where error was expected.

Acknowledgment

This research was supported by SGS project SGS-2013-050.

References

[1] M. Zahálka, J. Hlaváč, J. Staněk, Možnosti pokročilých simulací zatěžování klikových lisů. Kovárenství, 2014, vol. 50, p. 41-44. ISSN: 1213-9289

[2] J. Hlaváč, M. Círek, M. Čechura, V. Kubec, Rozvoj virtuální simulace v konstrukci mechanických lisů. In Technológia 2011. Bratislava: Strojnická fakulta STUv Bratislave, 2011. p. 499-504. ISBN: 978-80-227-3545-2

About Author (s):

Doc. Ing. Jan Hlaváč, Ph.D.
- Associate professor at Faculty of Mechanical engineering / University of West Bohemia
- Deals with forming machines' design
- Cooperate with producers of machines