STRESS ANALYSIS OF LABORATORY DRAWING TOOL USING CA SYSTEMS

Peter ZEMKO, Mária KAPUSTOVÁ, Jana ŠUGÁROVÁ, Jozef BÍLIK

Authors: ***Peter Zemko, MSc. Eng., Mária Kapustová, Assoc. Prof., PhD., Jana Šugárová, PhD., Jozef Bílik, Assoc. Prof., PhD.***
Work place: ***Institute of Production Technologies, Department of Forming, Faculty of Materials Science and Technology***
Address: ***ul. J. Bottu 25, 917 24 Trnava, Slovak Republic***
Phone: ***+421 944 134 468***
E-mail: ***peter.zemko@stuba.sk, maria.kapustova@stuba.sk, jana.sugarova@stuba.sk, jozef.bilik@stuba.sk***

Abstract

The contribution suggests meaning of software extensions of CAD systems for verification of forming tools and by that accelerating preproduction phase at workshops use. New and advanced FEA features of current 3D CA systems simplify verification even very complex forms of products. Tribute submits processed design for tool model for specific component with applying CAD software Inventor from AUTODESK Company.

Key words

forming tool, construction elements, modeling, verification, stress analysis

Introduction

Forming tools as active elements of machine industry enforce high educated deposit into production process and sophisticated solutions, which positively affects production process. Current market conditions claim on designers and manufactures of forming tools high demands, relevant to reduction of time necessary needed on project and cut-downs of input investments. Without application of computing techniques in the process of engineering forming tools, it is not possible adequate respond to customer's requests. Performing an analysis of a mechanical part or assembly in the design phase of forming tool can help bring a better product to market in less time. Using method of final elements (FEM) with CA application, allows to fully manage problems of stress condition analysis and deformations of construction materials.

Today's market with the 3D CAD systems has wide representation. To most widespread belong CATIA, UGS, SOLIDWORKS, PRO/ENGINEER, INVENTOR.

Understanding values of stress analysis

Utility of Autodesk Inventor Stress Analysis helps us at designing of forming tools:

- Verify if the tool part or tool assembly is strong enough to withstand expected loads without breaking or deforming inappropriately.
- Gain valuable insight at an early stage when the cost of redesign is small.

- Determine if the tool part can be redesigned in a more cost-effective manner and still perform satisfactorily under expected use.

Stress analysis tool features

Autodesk Inventor provides tools to determine structural design (forming tool) performance directly on Autodesk Inventor Simulation model. Stress Analysis includes tools to place loads and constraints on a part or assembly and calculate the resulting stress, deformation, safety factor, and resonant frequency modes.

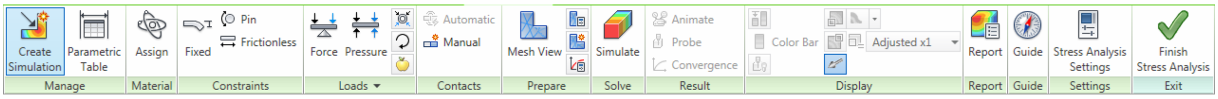

Fig. 1 *Stress analysis tool panel*

With the stress analysis tools, we can:

- Perform a structural static or modal analysis of a tool part or tool assembly.
- Apply a force, pressure, bearing load, moment, or body load to vertices, faces, or edges of the tool model, or import a motion load from dynamic simulation.
- Apply fixed or non-zero displacement constraints to the tool model.
- Model various mechanical contact conditions between adjacent tool parts.
- Evaluate the impact of multiple parametric design changes.
- View the analysis results in terms of equivalent stress, minimum and maximum principal stresses, deformation, safety factor, or modal frequency.
- Add or suppress features such as construction parts, re-evaluate the design, and update the solution.
- Animate the model through various stages of deformation, stress, safety factor, and frequencies.
- Generate a complete and automatic engineering design report of forming tool in HTML format.

The stress analysis environment allows analyzing assembly or part design and evaluating different options quickly under different conditions, for example using various materials, loads and constraints (boundary conditions), etc.

Stress Analysis workflow:

1. Creating simulations and specify their properties.
2. Excluding components not required for simulation.
3. Assigning materials.
4. Adding constraints.
5. Adding loads.
6. Specifying contact conditions, an optional step.
7. Specifying and preview the mesh, an optional step.
8. Running the simulation.
9. Viewing and interpreting the results.

Forming tool model analysis

For verifying the accurate drawing tool construction with blankholder was selected simple type of forming tool without guide (fig. 2), which was designated for experimental production of angular drawpieces with dimensions of 50x60x15mm and thickness of the wall 1mm. For the functional elements was chosen steel STN 19 191 (DIN C 105 W 1).

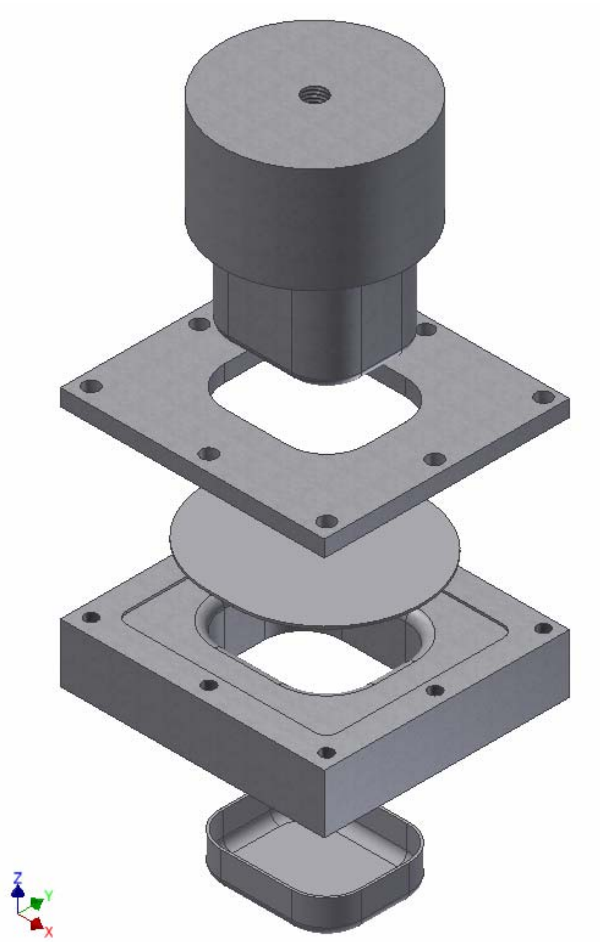

Fig. 2 *Drawing tool model*

For realization of stress analysis of the drawing toll model it was necessary to define loads. Drawing tool loading force was 200kN = drawing force of given drawpiece made out of material with Rm=1000MPa.

After defining the materials, loads, and constraints for the condition we wanted to test, and established contact conditions and mesh preferences we could run the stress analysis.

Results interpretation of stress analysis

The output of a mathematical solver is generally a substantial quantity of raw data. This quantity of raw data would normally be difficult and tedious to interpret without the data sorting and graphical representation traditionally referred to as post-processing. Post-processing is used to create graphical displays that show the distribution of stresses, deformations, and other characteristics of the tool model.

Interpretation of these post-processed results is the key to identifying:

- Areas of potential concern as in weak areas in a model.
- Areas of material waste as in areas of the model bearing little or no load.
- Valuable information about other model performance characteristics, such as vibration, that otherwise would not be known until a physical model is built and tested (prototype).

The contour colors display in the results corresponds to the value ranges shown in the legend. In most cases, results displayed in red are of most interest, either because of their representation of high stress or high deformation, or a low factor of safety.

Von Mises stress

Three-dimensional stresses and strains build up in many directions. A common way to express these multidirectional stresses is to summarize them into an equivalent stress (1), also known as the von Mises stress σ_v.

$$\sigma_v = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}} \tag{1}$$

where σ_v is von Mieses stress in MPa, σ_1, σ_2, σ_3 are principal stresses in MPa.

Von Mises stress results use color contours to show us the stresses calculated during the solution for our drawing tool model (Fig. 3 and Fig. 4).

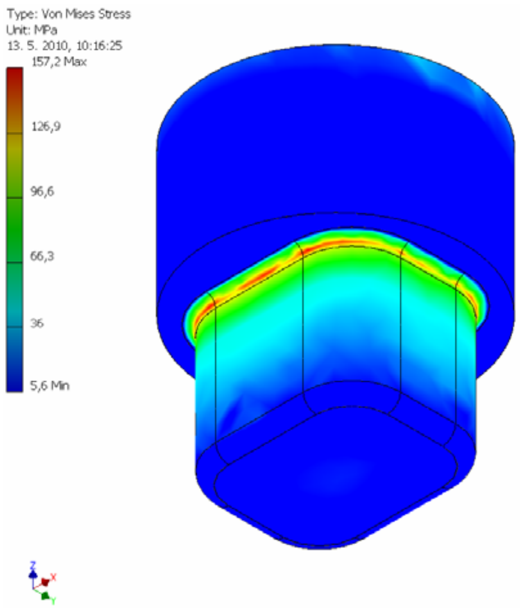

***Fig. 3** Von Mises stress for upper drawing tool part*

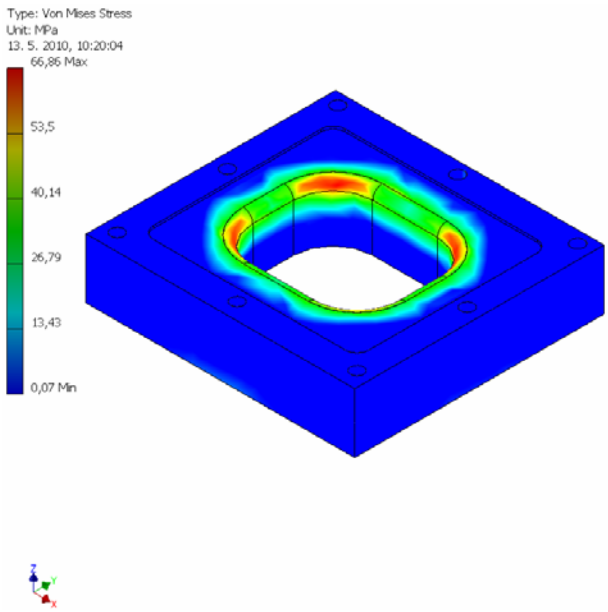

***Fig. 4** Von Mises stress for bottom drawing tool part*

After performing stress analysis of drawing tool model in defined technological conditions, we can state, that maximum stress values for upper part of drawing tool reaches 157.2 MPa (Fig. 3) and bottom part of tool 66.86 MPa (Fig. 4).

Deformation

The displacement results show us the deformed shape of our drawing tool model after the solution (Fig. 5 and Fig. 6). The color contours shows the magnitude of deformation from the original shape.

The deformation of upper part of drawing tool reaches maximum value of 0.02477 mm on the contact area of drawing punch and semiproduct.

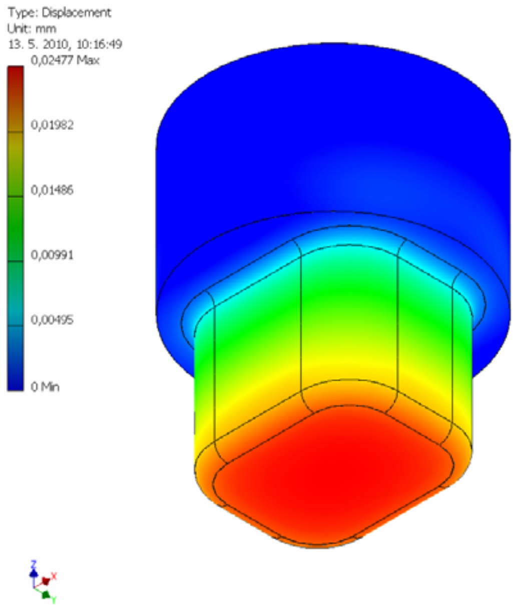

Fig. 5 *Deformation for upper drawing tool part*

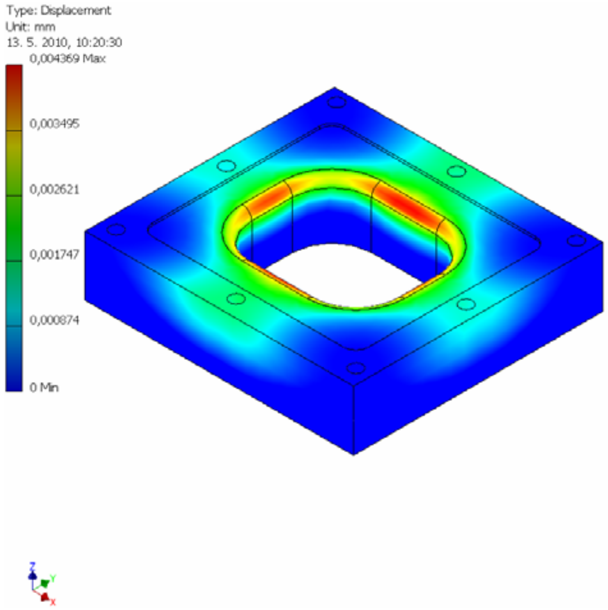

Fig. 6 *Deformation for bottom drawing tool part*

Deformation of bottom part reaches maximum value of 0.004369 mm. Maximal consequential values suggests that deformations are only in values of elastic deformations, therefore there is no negative impact on functional parts and we can assume that there won't be negative effect on quality of final product.

Safety factor

Safety factor (2) shows the critical areas of the analyzed model that are likely to fail under load (Fig.7 and Fig.8).

$$k = \frac{\sigma_k}{\sigma_v} \qquad (2)$$

where k – specified safety coefficient, σ_k – yield in MPa, σ_v is von Mises stress in MPa.

The color contours correspond to the values defined by the color bar. Minimum values should be in interval from 1.3 to 2 for given construction type.

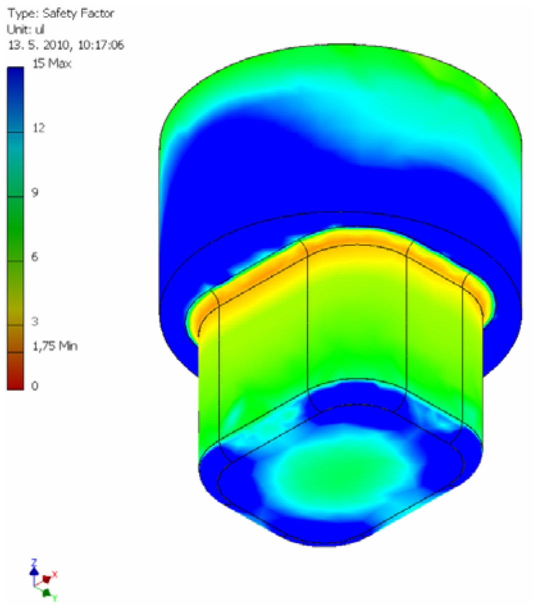

Fig. 7 *Safety factor for upper drawing tool part*

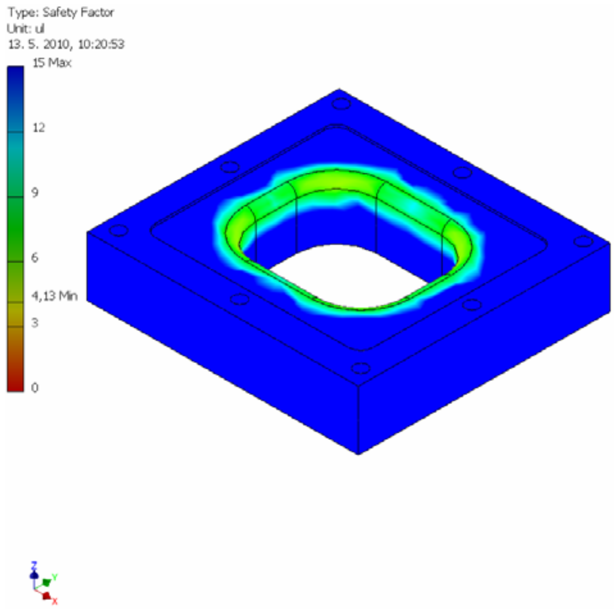

Fig. 8 *Safety factor for bottom drawing tool part*

The safety factor for upper part reaches minimum value of 1.75 and bottom part 4.13. Following the stress analysis results of the drawing tool model, it was evaluated as constructional suitable for current load.

Conclusion

In conclusion we can observe that stress analysis serves as a very important tool for behavior prediction of designed model in working conditions. Manual analysis thereinbefore mentioned parameters is not only time-consuming, however comes with risk of inaccurate results. Therefore in the light of current trends in the sphere of technical production preparations, in the ambit that major task represent also design of tool equipment in production process is important, among other things, even realization basic analyses, where in a substantive way is possible to affect whole process of design of forming tools, utilization CA technologies. Right application of stress analysis sharply decreases number of necessary physical tests, or later repairs.

Following presented evaluations we can realize changes on model and subsequently evaluate impacts implemented changes, i.e. wider area of design possibilities to improve the final product.

This article was supported by MANUNET project of ERA-NET scheme - Multivariate optimization of the metal spinning processes – research and development (projekt 7. RP EÚ MANUNET 2008-SK-001 and Vega 1/10060/08.

References:

[1] HRIVŇÁK, A., POLÁK, K. *Theory of forming and forming tools.* Bratislava: ALFA, 1989, p. 244-335.
[2] PLANIČKA, F., KULIŠ, Z. *Basics of the plasticity theory.* Praha: ČVUT, 2004, p. 163-166. ISBN 80-01-02876-3

[3] POLÁK, L. Forming tools – knowledge products of machine industry. In *Strojárstvo*, 2007, 2, p. 50-51.
[4] TOMÁŠ, M. Projection of shaping tools using CAD-methods. In *Strojárstvo*, 2006, 4, p. 16-17.
[5] AUTODESK. [online], [cit. 10.2.2010]. <*http://images.autodesk.com/adsk/files/inventor_sim_2010_getstart.pdf*>