MODELING OF TANDEM TOOLS USING COMPUTING TECHNIQUE

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Abstract

The contribution suggests meaning of software extensions of CAD systems for creation production documentation of working tools and by that accelerating preproduction phase at workshops use. New and advanced features of current 3D CA systems simplify modeling 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

tandem tool, construction elements, modeling, CAD systems, system upgrade

Introduction

Compression molding at the present time belongs between progressive production technologies, whereby progressiveness of compression molding rests in cut - downs of consumption stock on part or construction, in improving property of starting material, low production times, high productivity, in posse automation and mechanization of helping operation and act. Forming tools as products of machine industry enforce high educated deposit into production process. Current market conditions, which claim fast response on customer requisites, also lay on designers field of forming tools and theirs producers high demands. Without application of computing techniques in the process of engineering forming tool, it is not possible adequate respond to customer's requests. By using various CAD application, allowing, among other things, reduction of time necessary needed on project, user's extension and databases of normalizes have their major task too [4].

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

Computer support of engineering forming tools

Construction of working tool influence many factors e.g.: shape and size of workpiece, production sort, type of machine, a o. Tool for mechanical working are typical for their individuality, which results from need of individual realization of tool, separate for every part under specific consideration of production conditions (series, available machine features, parts quality etc.).

Process of computer supported construction of forming tools consisted from these stages:
- processing incoming data,
- own computer-aided design,
- output of information in the asking form.

In the first one stage is realizes calculation of assembly- technological parameters for specific forming tool, which are foundation for the next stage. Primarily are used independent programs as e.g.: the semiproduct size calculation, working force, functional parts dimensions etc., but also complex programs e.g.: for calculation assembly- technological parameters at drawing cylindered extract with flange [4].

In the second phase through the use of CAD systems its realize proposal conception of forming tool for specific production conditions and its constructive solution and consecutive approach to creation of drawing documentation for individual parts production of tool. These activities are possible to carry out with use databases of assembly units and parts, and from those are later set specific tools, q.v. but it dependents on existence of this databases. As well at drawing design of working tools for production of shapes and dimensions similar part is possible put to use drawing built - up infobases from solution tools, which are adjust to required proportions according to shape of new parts.

The third stage introduces substantiation of computer data to real form of drawing documentation, which serves as basis for individual parts production of forming tools.

User's system upgrade - modernization features of CAD the systems

From listed stage characteristic is evident that the largest area of computing support in designing of working tools is in the second stage, when are used different CAD systems and their software extension - user's system upgrades.

User's system upgrades are possible to classify by three subgroups:
- computing (MEP, toothing calculation , screw join, etc.),
- universal (for creation of drawing documentation – marks, symbols, tolerance charts, etc.),
- specialistic (containing close specialistic databases part, e.g.: guiding stand - up, chevron chaplet and cabinet, guiding elements, etc.).
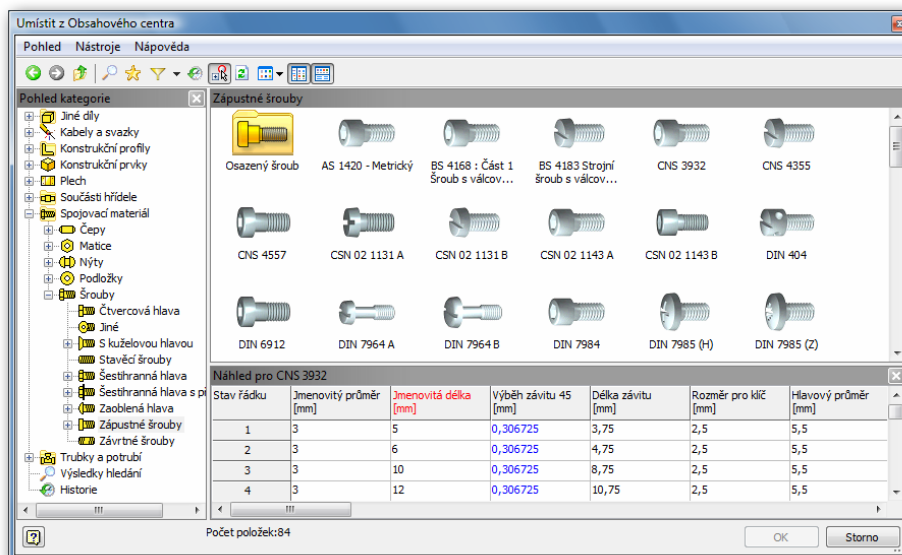
Fig. 1. *Content center window*

Gradually improving CAD system are these single features, typical for these user's system upgrade, becoming standard in basic programmatic packages. In the event of standardized parts it is possible in installation of custom system upgrade choose the appurtenant standards, which user works most frequently with and from those consecutive realizes selection dimensions of concrete depositing part. Obviosity is also insertion of parts, or e.g.: joints in the necessary look (front elevation, ground plan, side projection), eventually in the cut. After insertion of parts into already designed drawing realizes automatic adaptation of others line which can be in collision. Along with insertion standardized part into design are inscribed even data about these part in form of Info-dot, which consecutive simplifies creation of bill of material in assembly drawing. Info-dot is depositing toward each normalizing part and contains items, which this part exactly specify (figure positions, title, norm, amount pcs., etc.). Data contained in the Info-dot individual standardized also un-normalized parts are automatic inscribe in appurtenant table item of bill of material in order of position figures [4].

Specialized user's system upgrades are oriented above all on automation routine activities as creating drawing documentation with translation focus specific area, which are e.g. working tools. These system upgrades are created by manufacturers of single parts for forming tools and contain libraries with individual elements and alone tool assembly is possible to set up with their assembling and modification of operational sections. Besides traditional user's superstructures, which are integrated after installation into environment of graphic editor, it is possible to use also internet's norm libraries, where desired component can be consecutive downloaded by user into his own computer in form of file.

Processing design of tool model for chosen part

For modeling specific parts was he applied CAD software Inventor from company AUTODESK. Function of suggested part is clamping console. For the component was selected steel S235J2G3. Through the use of modulus for design sheet part was created part and later also production sequence of required components (Fig. 2). Following hereof progress was created model of forming tool.
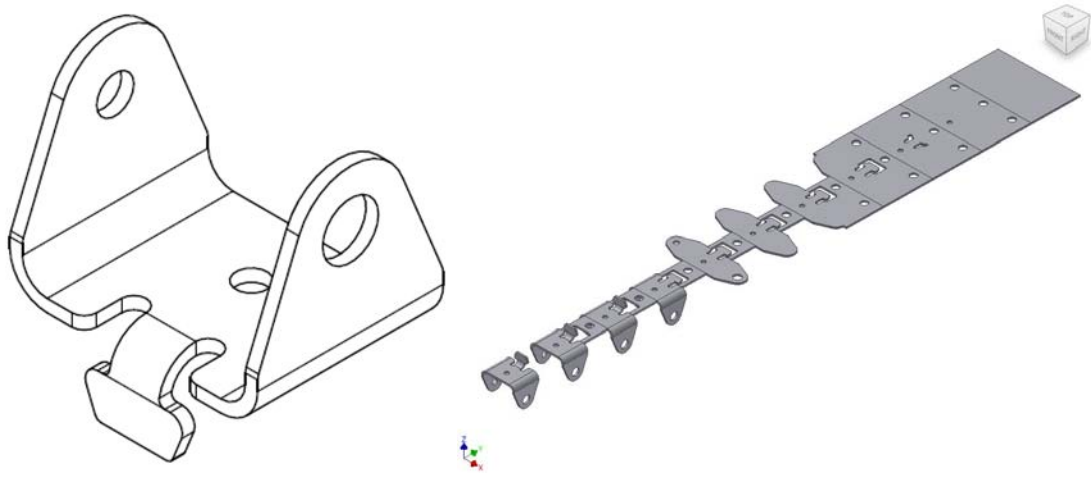
Fig. 2. *Component and production sequence*

For modeling individual parts of tool was used module for 3D modeling.

The most frequently used functions for design of assembly parts were:
- sketch: Line
  - Circle
  - Curve
  - Rectangle
  - Round

- 3D models: Extrude   Fillet
  - Rotate   Chamfer
  - Hole   Move plane

After launching Inventor an open window displayed for new file option. The first thing was necessary to model single parts of forming tool. For design of components was selected file ***Norm.ipt*** .

First created component of tool was die set according to FIBRO company catalog.

Die set consists of 4 elements:
- base plate ,
- retainer plate,
- pillars (4x),
- head guides (4x).

Modeling of the base plate
Through the use of sketch tools as line, circle, dimension and bond was created top view of base plate for die set.
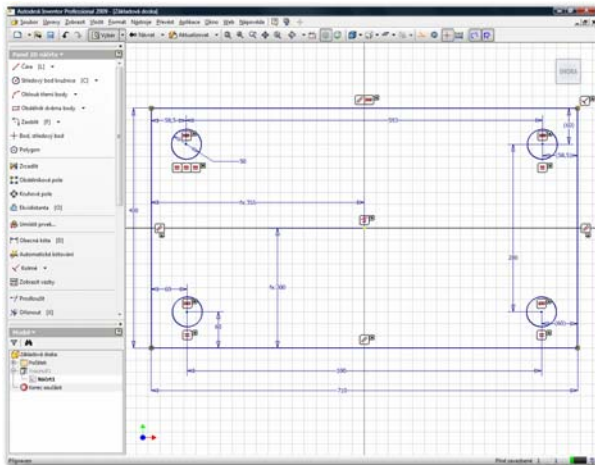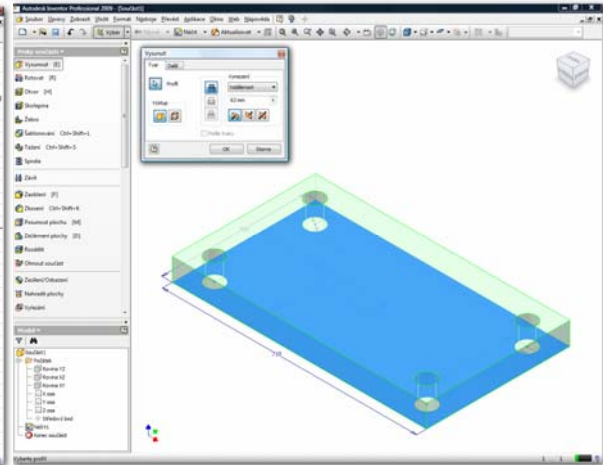
Fig. 3. *Top view of base plate* **Fig. 4.** *Extrusion of base plate*

From sketch into 3D design environment was done by clicking on the **Return** icon.

With the **Extrude** function was made a 3D model according to entered parameters.

Model was saved under the name Bases.

The same way was designed retainer plate of die set.

Other components, i.e. pillars and head guides were downloaded from internet content library. The content library offers also catalogs of parts for forming tools for a better orientation in the offer.

Modeling of die set assembly
For the design of die set it was necessary to create a new file
File for the assemblies is **Norm.iam**.

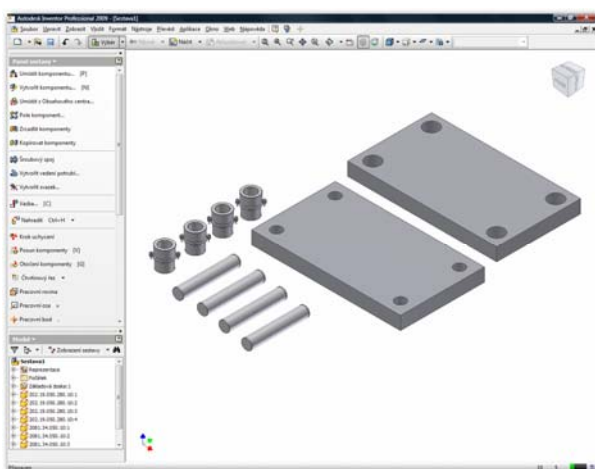With the function **Place component** were inserted parts of die set.
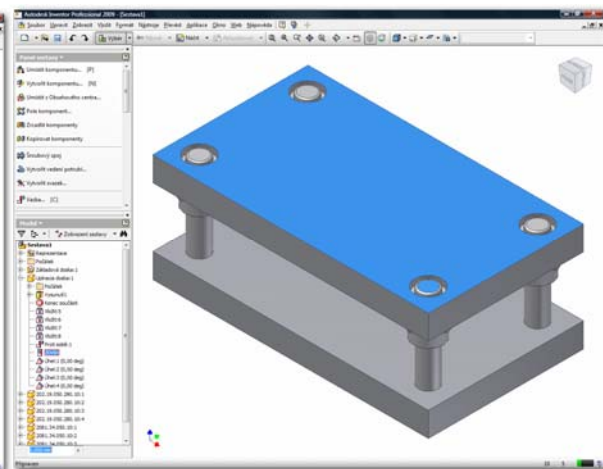

Fig. 5. *Inserted components of die set* **Fig. 6.** *Die set assembly*

The next step was to make constrains between components with the function **Place constrain**.
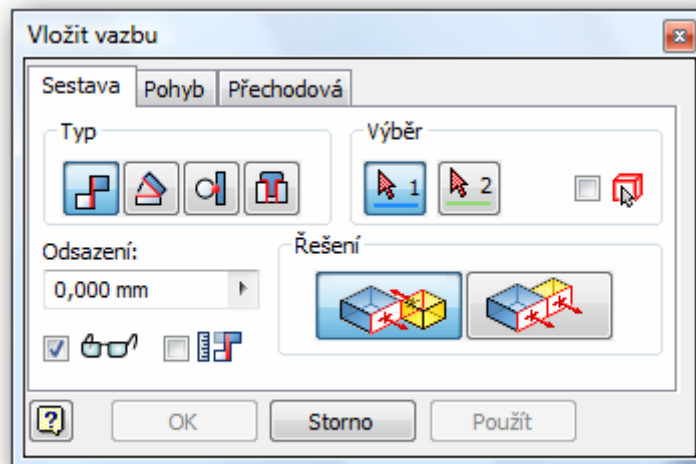

Fig. 7. *Constrain window*

Using the constrain function **Insert**, constrains were made between the holes on plates, pillars and head guides. The **Mate** constrain made a parallelism between the plates and constrain to regulate stroke.

Similar way were created further tool components and their sub-assemblies e.g.: punches, guide elements, pilots, etc.

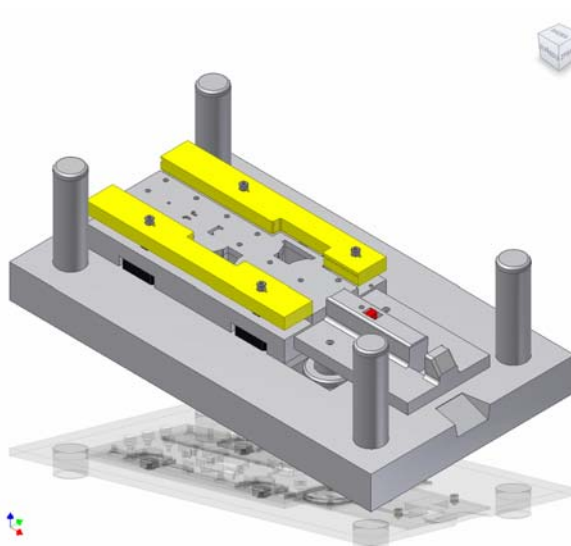Placing several components and sub-assemblies into design a complete forming tool was made.
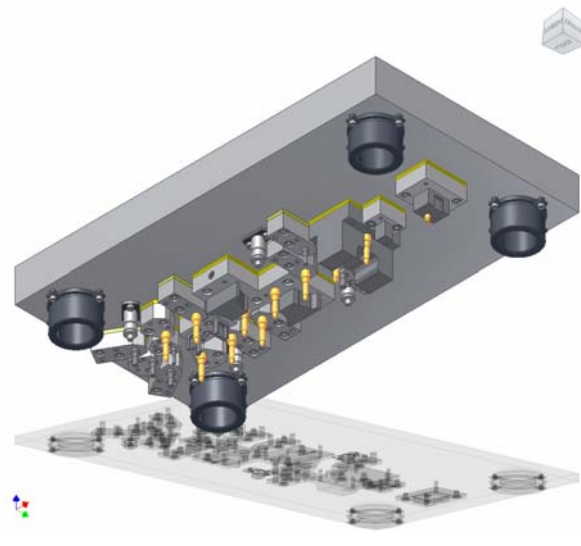

Fig. 8. *Lower assembly of forming tool*

Fig. 9. *Upper assembly of forming tool*

After the design was done it was necessary to make a detection of collisions.
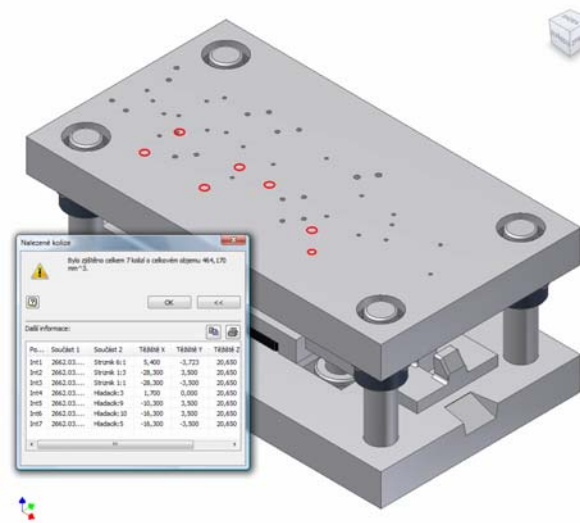

Fig. 10. *Collisions detection*

The test showed some collisions which were needed to fix. Projected report describes components in collision, location and centroid of collision solid. Collision detection was the last step of modeling. Next step was to make a drawing of forming tool.

Conclusion

The objective of contribution was design of forming tool by computing techniques. Modeling parts of tools in 3D system as program Inventor, allows us in the present wide possibilities in development, production, process and individual modeling of working tools and theirs parts. Along with development of technique it comes to progress of used system and by that to increase in effectiveness exploitations used devices and to expansion of output and even to enhancement of individual production.

This form of documentation creation through the use of CA software is very advantageous, because it relieves modeling even of very complex forms of products, thanks to development and designing new features, which these systems offer.

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