

PARAMETRIC DESIGN OF DIES FOR CUTTING AND DRAWING USING CATIA ENVIRONMENT

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Abstract: *Products which are going to be launched on the market are designed using different CAD systems. The design stage is a time consuming phase, because of geometrical changes which are often implemented to improve the products according to the customer needs. For each design change, not only the reshaping of the parts, but especially the redesigning of the tools needed for their manufacturing will increase the time of development phase. This paper presents a method of parametric designing, using Catia V5, of a compound die for a rotational part made from sheet metal using cold pressing processes. For each design change of the part, the die design will be automatically updated according to part dimensions.*

Key words: *rotational parts, die, parametric modelling, Catia.*

1. Introduction

In the last few decades the trends on the products' market are continuously changing. The customers' requests are at high levels, thus due to the worldwide competition on market, the companies have to design and manufacture new products in short time at high quality and low costs. In the context of product life cycle, the product development phase plays an essential role. In this phase a time consuming operation is the design stage which supposes more or less time according to the products complexity and the level of CAD/CAE/CAPP/CAM integration.

Nowadays the Computer Aided Design (CAD) tools are worldwide used. Computer Aided Design is defined to be the activity of using a computer system for designing, analysing and optimising of the products according to the principles of design of manufacturability [7]. The first developed CAD software was for 2D drawings and sketches (1960 appears SKETCHPAD - first 2D drawing software), and later it was developed software at 3D modelling level. In 1969, Syntha Vision was launched, the first CAD commercial software which had a solid modeller [1]. In the next years are released new 3D CAD software systems like AutoCAD, Unigraphics, Catia, and so on [1]. In 1989 was introduced by Parametric Pro (T-Flex) and later by Pro/Engineer, the concepts of parametric design, which means that the 3D models are defined by parameters [1]. A change of dimension value in one place also changes other dimensions to keep relations between all elements in the design. The parametric design is an important tool that helps

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manufacturers to reduce the time of development phase (design phase) especially when it's about parts families and their associated manufacturing sequences and necessary tools used for production. The designed products have been often changing, and for every changed design stage, the redesign time has to be taken into consideration. Parametric modelling represents change [9] and using it the time for redesigning will be substantially reduced. Nowadays, a lot of CAD/CAM software systems like: CATIA, SolidWorks, Pro/ENGINEER, Siemens NX, Delcam Power Shape, Autodesk Inventor allows parametric modelling and they sustain the companies to be competitive on the market. Zheng-hao et al. used a parametric modelling method in Pro/Engineer for a Globoidal Indexing Cam [11], Du Yao et al. used SolidWorks and KBE for a parametric design and calculation of metal profiles [10], Ghionea used Catia for a parametric design of the modular devices element [4], and there are many others applications for parametric modelling in the literature. In different CAD environments the parametrical drawings can be obtained using programming languages like C++, Visual Basic, AutoLISP/VisualLISP, OjectARX etc. For example in the literature it was identified in AutoCAD/VisualLISP software tools used to simulate the metal spinning process by multi-pass path [3] and to design cold stamping [2] or in combination with OpenDCL to design the manufacturing technologies for air-cooled single-cylinder engine block [6].

Catia (*Computer Aided Tree dimensional Interactive Applications*) is a Dassault Systemes product, being one of the most used integrated systems CAD/CAM/CAE worldwide, with applications in automotive, aircraft industry, machine and equipment manufacturing industry etc. [5]. It contains many workbenches for various engineering domains and it provides a variety of solutions including a high level of parameterization of the 3D models, being possible utilization of external data bases or files for parameters management.

2. Parametric Design Method of Dies for Cutting and Deep-Drawing

2.1. Problem Formulation

The research aims to design in a parametric mode a die used for manufacturing of a rotational part with various dimensions and configurations. The parametric die will be designed in Catia environment using external files to define the parameters for part geometry.

It was chosen the representative cylindrical flanged part manufactured from DC04 sheet drawing steel. The part parameters as can be seen in Figure 1 are: sheet thickness, inner diameter, flange diameter, part height, punch radius and plate radius.

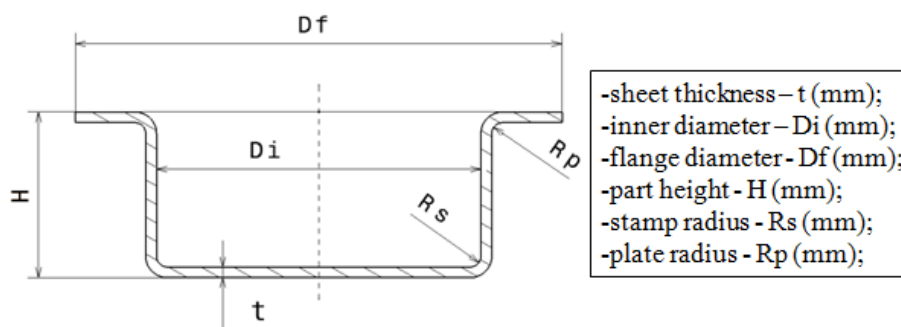


Fig. 1. Part parameters

To manufacture this part, are needed several basic cold forming processes as follows:

- trimming / cutting the blank from the entire metal sheet;
- one or more drawing operations;
- a calibration process;
- trimming extra material to get the flange diameter dimension.

It was chosen the design of a parametrical compound tool for the first operations: trimming the sheet metal blank and the first drawing operation. Thus, this tool can be used for the parts which are manufactured with one single drawing operation, or for the parts which need more than one operation for the first drawing operation.

2.2. Parametric Design Method

The idea of the used method is to design a parametric 3D die for a basic part, in this case a cylindrical flanged part (see Figure 1), according to the part dimensions. When the part dimensions are changed, the die dimensions will be automatically updated in a short time. If in a normal way, the design of a new die can take more than one week, using this application the die design will be done in less than one hour. This method can be very useful for parts families, when it is needed to keep the geometry, changing the part dimensions and also for necessary tools for its manufacturing.

Figure 2 illustrates the block scheme for the parametric design of the compound tool for trimming the blank and drawing operation, according to part dimensions and tool standardised components configurations. It can be seen which are the input data and which are the final outputs data.

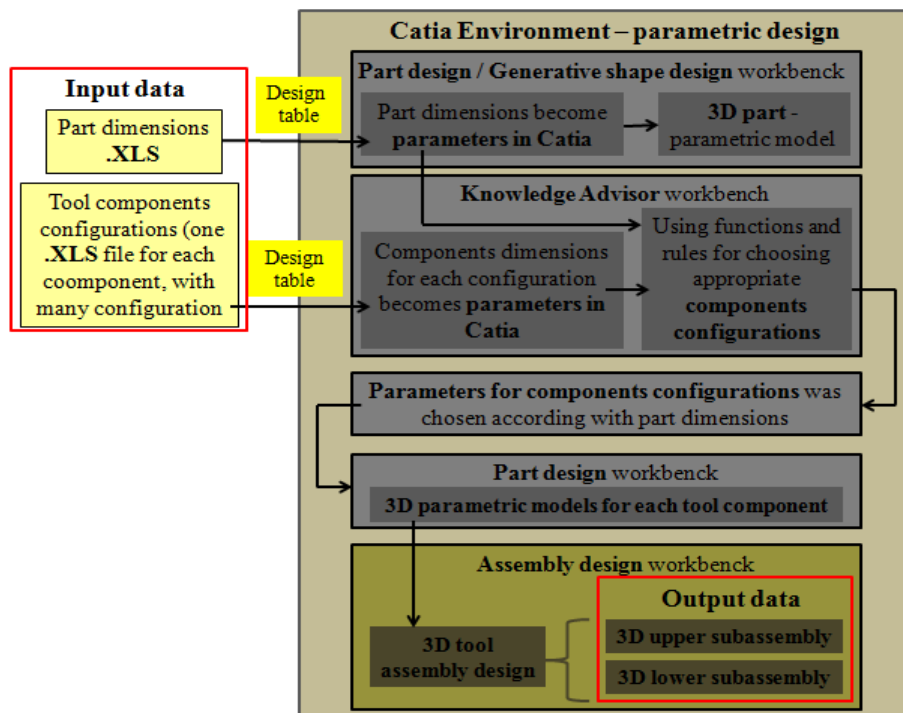


Fig. 2. Working principle block scheme

Below is presented the design method, step by step, starting from input files up to final output tool assembly as 3D model.

Step 1 - Excel files creation for part and components parameters.

Based on an Excel template which contains a part drawing (Figure 3), it was customized the file where the part parameters are introduced. Row number 1 contains the parameters names, and row nr. 2 contains their values. Certainly, the technical conditions for the drawing process have to be accomplished [8]. This conditions are also checked in this excel file, using mathematical functions and formulas, according to the parameters introduced by the user. If the parameters introduced did not respect the technical drawing conditions, in the bellow cell the "Conditions" title are written an warning text (Figure 3). In this case, the user has to change the specified parameter until the drawing conditions will be accomplished. Note that if all drawing conditions will not be respected, the tool will be designed, but the drawing process will not work in right conditions.

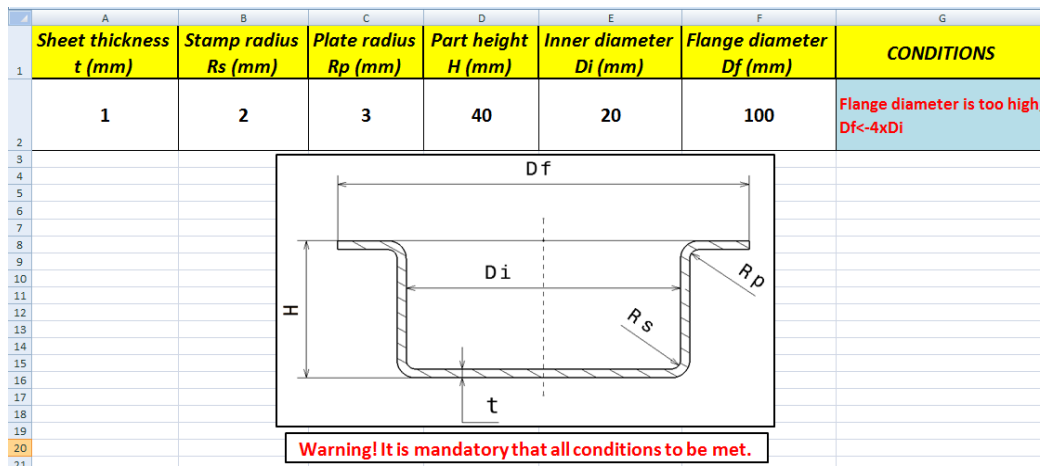


Fig. 3. Excel file for part dimension

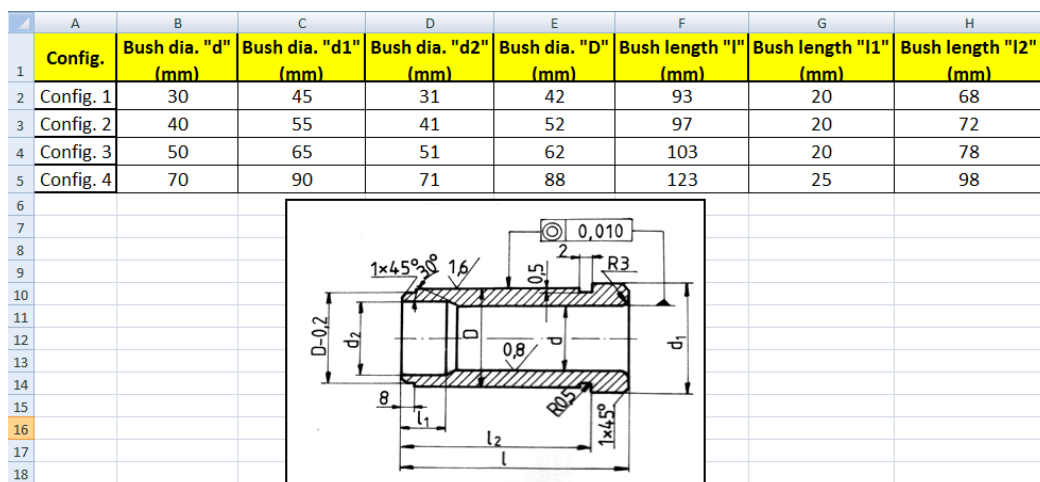


Fig. 4. Excel file for a tool component configuration

The tool assembly is composed from many different components, usual with some standardised dimensions [8] and with some customized dimensions according to the blank and part gauge. For each standardized component was developed one excel file containing many component configurations (Figure 4). A row corresponds to a configuration, namely a dimensional variant. In Figure 4 it can be seen an example of a tool component (guide bush for tool column), with four rows, thus four component configuration. According to the representative part dimensions, automatically will be chosen the right configuration.

Step 2 - *Parameters import from Excel to Catia environment using “design table” function.*

First, the parameters that have been defined in Excel files were created also in Catia using “formula” function available in Part Design workbench. Second, using “design table” function available in Knowledge Advisor workbench [12], to Catia parameters will be associate a parameter value from Excel files. For the part and each tool component, was implemented their own “design table” association. In Figure 5 is presented an example of a design table association between six part parameters defined in Excel file and Catia parameters. When the parameters will be modified in Excel files, they will be automatically updated in Catia environment.

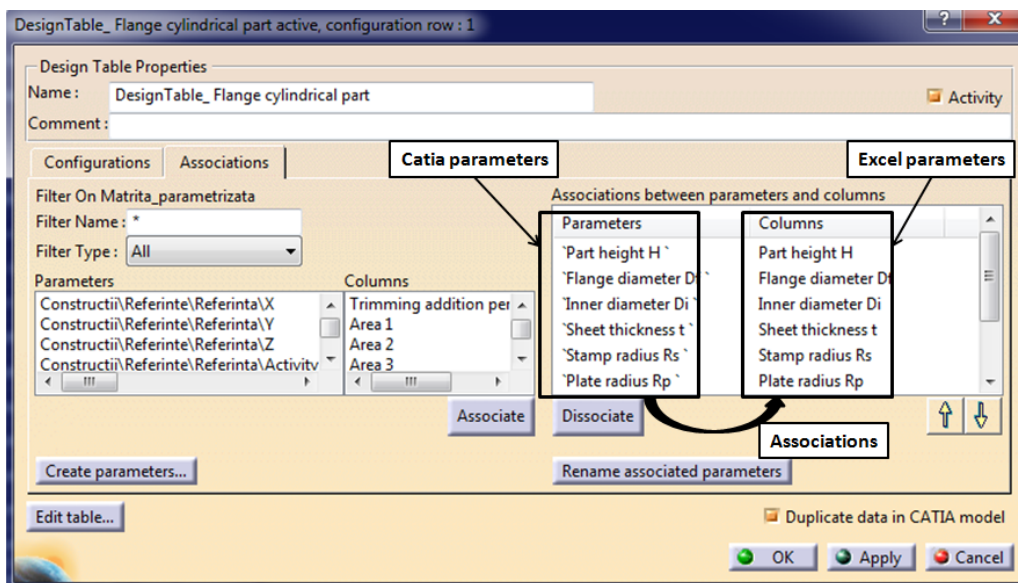


Fig. 5. Association between Excel and Catia parameters

Step 4 - *Defining rules for choosing the right components configuration.*

As mentioned above, for each tool component are defined many standardised configurations. According to the part dimensions, the right components configurations have to be selected. For that it was implemented a series of rules and formulas using commands available in Knowledge Advisor workbench. It was used the “if” function to create relation between part parameters and components configurations. In Figure 6 is shown the choosing rule for the fixed pawl according to blank diameter value. For

example, for blank diameter $Db \leq 60$ mm, will be used the first pawl configuration, thus the first row from its Excel file, which contain pawl standardised dimensions.

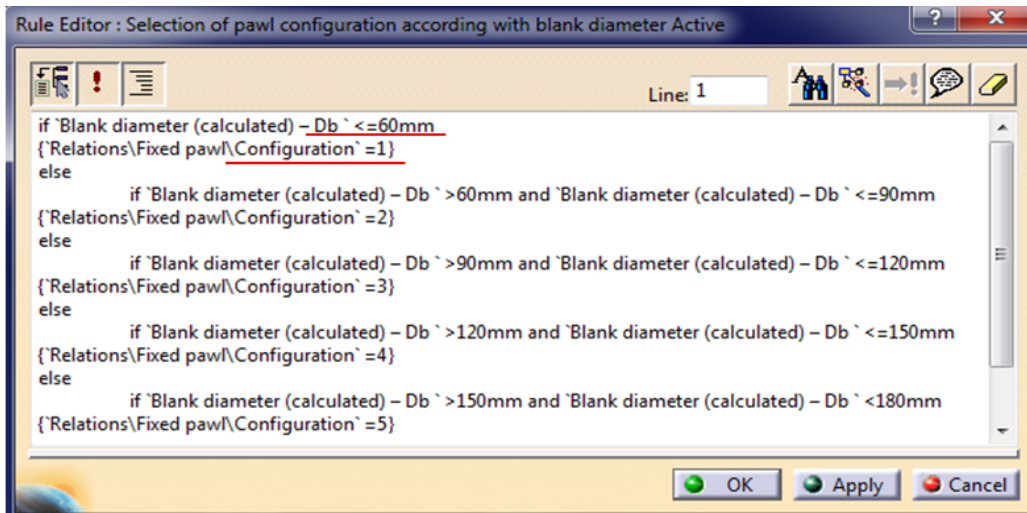


Fig. 6. Choosing rule for fixed pawl

Step 5 - Part and tool component parametric modelling.

Using the parameters chosen in step 3, the tool components were modelled with usual commands available in Part Design and Generative Shape Design workbenches. All the components were designed in an “Input parametric model” CATPart (Catia specific file for single parts) and copied using publication and links, each tool component in its own catpart document. In final stage, all catparts are assembled in two subassemblies: lower subassembly and upper subassembly (Figure 7).

3. Case Study - Parametric Die Designing for a Cylindrical Flanged Part

As above was mentioned, for a cylindrical flanged part, was chosen to design a parametric tool for sheet metal blank trimming and for the first drawing operation. The tool components were modelled according to the part dimensions which are the input of the parametric model. They are introduced in an Excel file and associated to Catia environment. If a new tool for a new part dimensions have to be developed, should be follow next steps:

- open the Excel file with part parameters and the parametric model in Catia;
- input the new parameters in Excel file and save the modification;
- switch to Catia environment and wait until the modifications are noticed; a dialog window appears to inform that something was modified in Excel parameters and ask to synchronize them with Catia parameters; for synchronize, “Yes” button has to be pressed (Figure 7);
- update the parametric model; when the parameters are synchronised, all tool components will be in red, asking for update. Thus press the “update” button;
- final step is to update the final outputs, the tool lower and upper subassemblies.

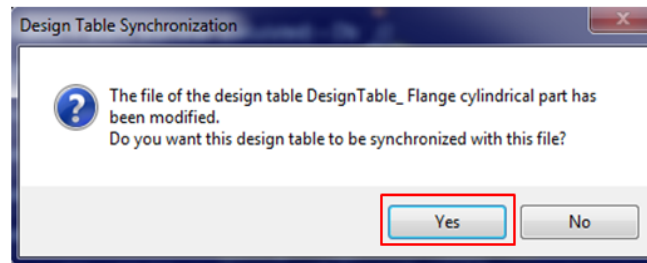


Fig. 7. Dialog window for parameters synchronization

To keep the links between the parametric model (input) and the final subassemblies outputs, it was integrated in a working structure (Figure 8). This structure was named “input-output structure” and it is a CATProduct specific Catia file. It allows to copy with link the tool components from the parametric tool to their own part in each subassembly. Figure 8 shows the input-output working structure and the final tool assembly.

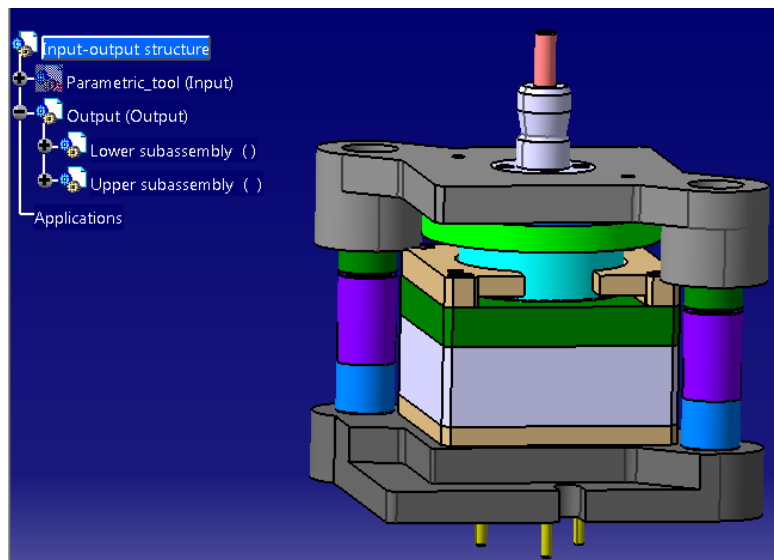


Fig. 8. Input-output working structure with final tool assembly

4. Conclusion

It was developed a parametric trimming and drawing tool, used to manufacture a cylindrical flanged part. Together with the Catia software system were integrated several Excel files to input the part parameters and standardized tool components configuration. At any time the tool components configuration files can be updated with new rows when new configurations are needed. The parametric model is easy to use even if the user has no knowledge in design domain, only the part parameters from Excel file has to be modified and update the 3D models. This parametric method can be used to develop 3D models of parts and products for different domains, using any CAD software which allows parametric design. It works very well for parts family and has a big advantage - to reduce the products development time.

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