# GEOMETRIC MODELING OF PARTS USING SOLIDWORKS AND CATIA APLLICATIONS

TUDOR Bianca, TECUCEANU Diana Georgiana and POPESCU Marius Andrei
College: Industrial Engineering and Robotics, Specialization: Consultancy in Mechanical System Design, Year of Study: I, e-mail: biancaalbu90@gmail.com

Scientific leader: Conf. dr. ing. **Iulian TABARĂ** and Ş.l. dr. ing. **Alexandra ROTARU**

*ABSTRACT: In the presence of the scientific paper we want to highlight the design of two different parts crankshaft, respectively 3D support frame using two different design programs CATIA and SolidWorks.*
*The crankshaft transmits the rotational to the connecting rod, as part of an engine assembly. Crankshafts are dimensioned according to the power developed by the engine which define the loads to which it is subjected.*
*Regarding the support designed in SolidWorks we remind you that the three-dimensional sketches consist of successive lines and arcs as well as three-dimensional spline curves, and in the following presentation we will use an artifice to design the piece.*

*KEYWORDS: geometric modeling, CATIA, SolidWorks.*

## 1. Introduction

CATIA (Computer Aided Three-dimensional Interactive Application) is a CAD/CAM/CAE cross-platform commercial software suite developed by the French company Dassault Systems and marketed worldwide by IBM. CATIA supports several stages of product development, from conceptions, design (CAD), manufacturing (CAM) and analysis (CAE). It is widely used throughout the industry, especially in the automotive and aerospace sectors.

SolidWorks is a highly productive 3D CAD software tool with integrated analytical tools and design automation to help stimulate physical behavior such as kinematics, deviation, dynamics, vibration, temperature or fluid flow to suit all types of design. This 3D modeling CAD software is widely used in the mechanical engineering and design industry. It is the industry standard for product development.

The aim is to provide an image of the generation of different parts treated in two distinct design programs by various methods. The version of the program used is 2016 – for SolidWorks and for CATIA V5.

## 2. Modeling a crankshaft

The first step is to create a new “Part” file in the CATIA V5 software (Figure 1).

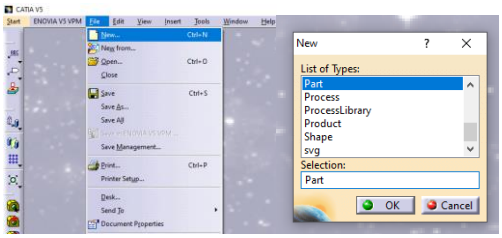


Fig. 1. Creating a new file

A sketch (Figure 2) was created in the ZX plan (Figure 3) to begin modeling the crankshaft.


Fig. 2. Creating a sketch

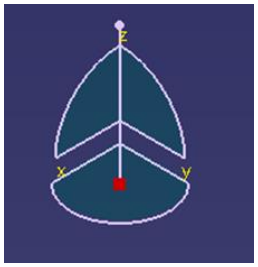


Fig. 3. Choosing the plane ZX

We made the crankpin journal using the “Circle” command from the command manager (Fig. 4).

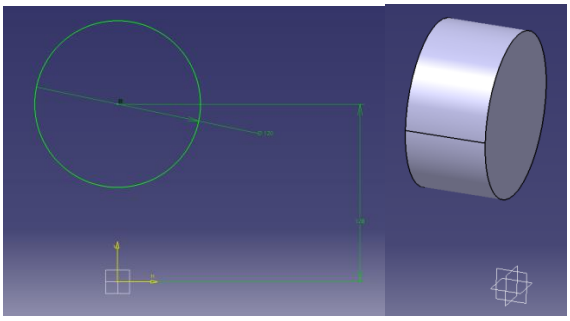

Fig. 4. Creating a crankpin journal

The next model was a shoulder for centering the crank in the crankshaft system using the bearing of the previously made bearing spindle as a support using the same command as the top “Circle“ (Fig. 5).

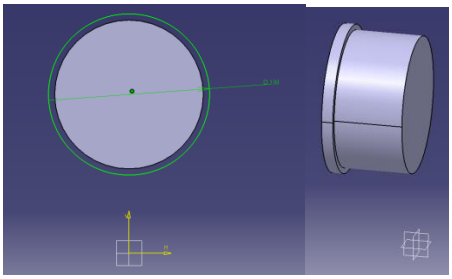

Fig. 5. Creating a shoulder

We made the “turns” profile using the face of the previously shaped shoulder as a sketch support (Figure 6).

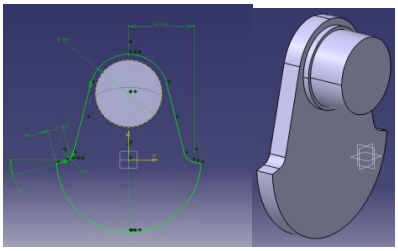

Fig. 6. Creating a turning profile

The next thing to do for the model is the shoulder for the crankpin journal using the front turn as a support (Figure 7). Next the spindle bearing was made (Figure 8).

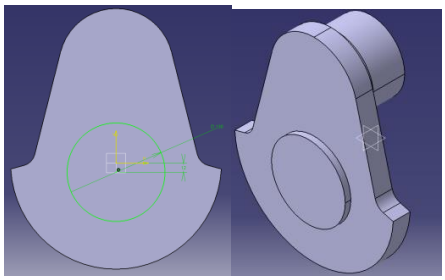

Fig. 7. Creating a shoulder

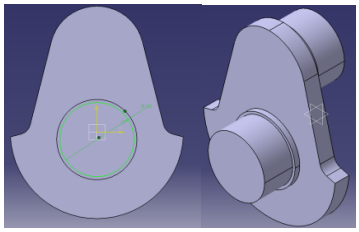

Fig. 8. Creating a spindle bearing as main journal

Next we made the other shoulder of the spindle bearing using the “Circle” command (Figure 9), following the other turn (Figure 10).

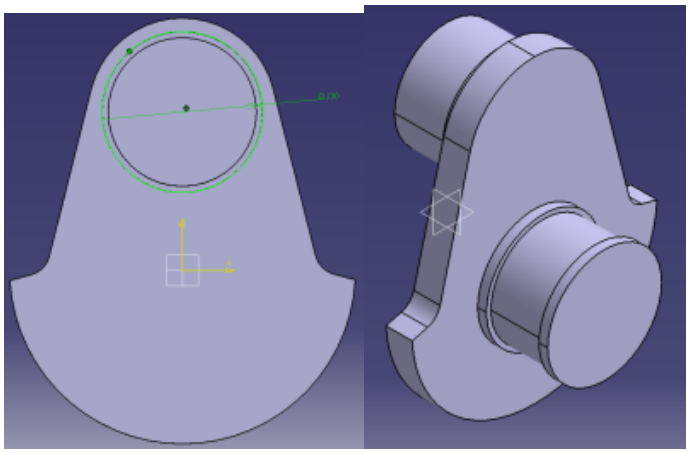

Fig. 9. Creating the other shoulder of the spindle bearing

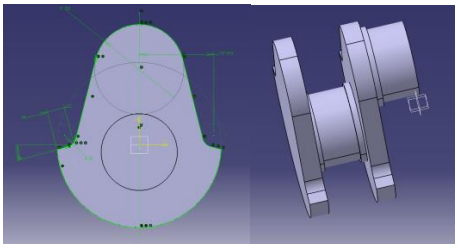

Fig. 10. Creating the other turning

The rest of the crankshaft was made in the same way as presented above. Finally, a crankshaft was made that can be used on 4 cylinder engines with 4 crankpins journal (Figure 11).

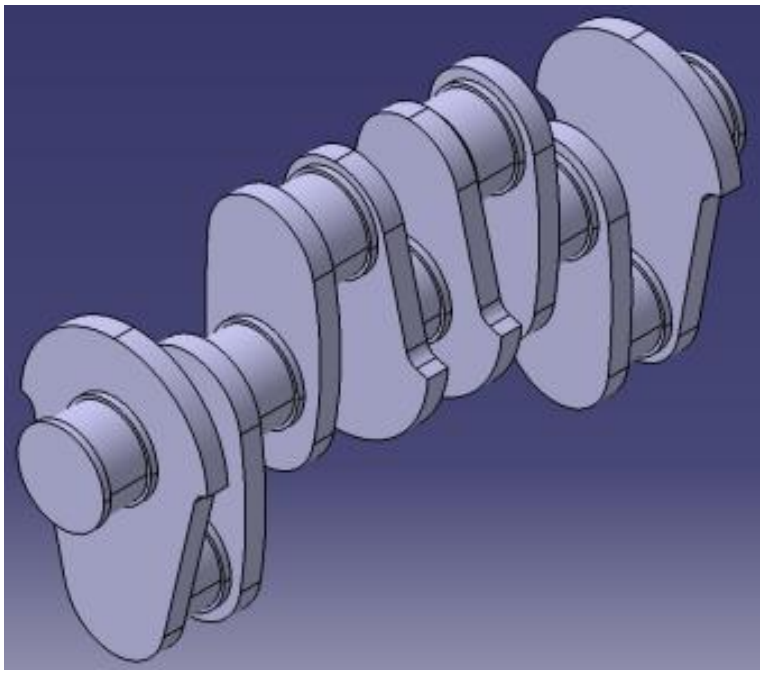

Fig. 11. The crankshaft

We have exemplified below what are the spindle bearings and crankpins, the spindle bearings hold the crankshaft and they are alined forming the spining axis, and the crankpins are the bearing supports on which the connecting rods are mounted (Figure 12).

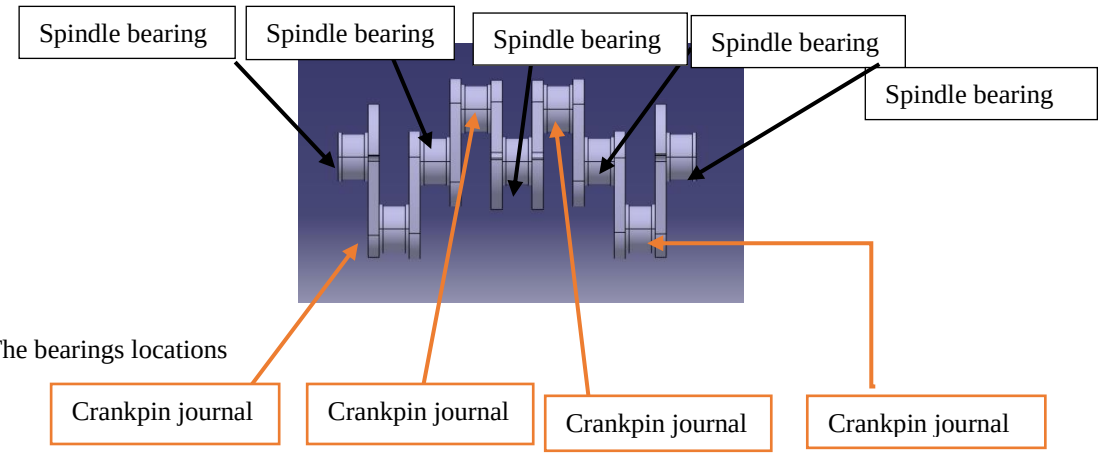


Fig. 12. The bearings locations

## 3. Modeling a 3D support frame

The first step is to create a new “Part” file in the SolidWorks software (Figure 13) in which we will select 3D Sketch (Figure 14).

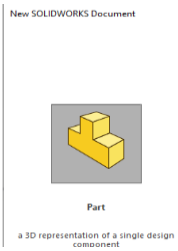


Fig. 13. Creating a new file

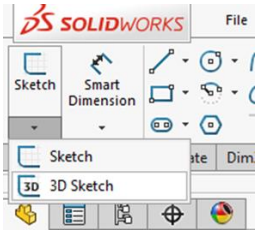


Fig. 14. Selecting a 3D Sketch type

We start by drawing first the right half of the piece drawing a line using the Line function on the OX axis starting from the beginning in the XY plane (Figure 15) a segment that we will quote in the properties window of this line in Parameters box.

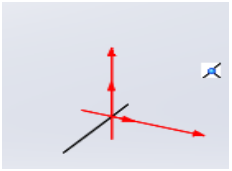

Fig. 15. Choosing the plane XY

We will continue sketching a segment in the YZ plane (Figure 16) moving the cursor over the right end and of the already drawn segment at which point we simultaneously press TAB + the left button to move to the YZ drawing plane. The newly appeared segment will be listed in the Line Properties -> Parameters box window.

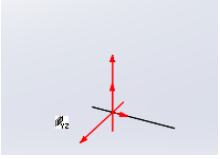

Fig. 16. Choosing the plane YZ

Once all the segments that make up the frame have been sketched we will move on to the next stage namely the 5 connections, successively selecting the lines that form each intersection in the sketch using the Sketch Fillet function from the Sketch Bar (Figure 17).

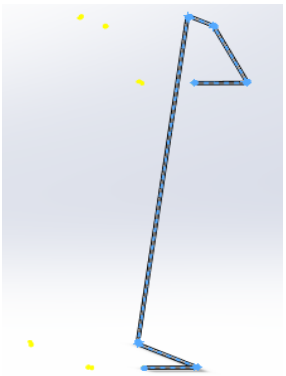

Fig. 17. The connections

The first half of the model will be made by drawing a circle along the 3D Sketch (Sweep) so the right plane and the Sketch button will be selected to open a new sketch. We will use the Zoom to Area function, enlarging the area that comprise the first segment the one at the top of the dragging and select Circle to draw a circle with the center at the left end of the segment (Figure 18). Exit sketch and return to the command manager to Features button frum where we will select Sweep thereby appearing a new Sketch under the name Sketch 1 which will be found in the text box corresponding to the profile.

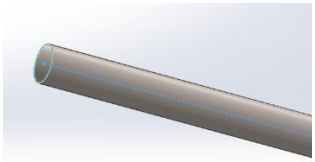

Fig. 18. The circle at the end of the segment

To finish the piece we mirror the pattern in relation to the circular face: Insert -> Pattern -> Mirror by clicking on the upper circular face, followed by clicking anywhere on the frame (Figure 19).

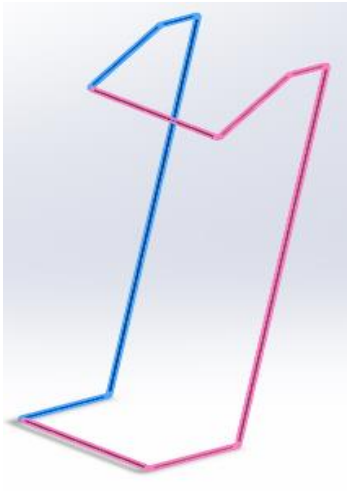

Fig. 19. The mirror pattern

The last step is to choose the material, in our case using Alloy Steel. The 3D support frame is completed and its shape can be found in Figure 20.

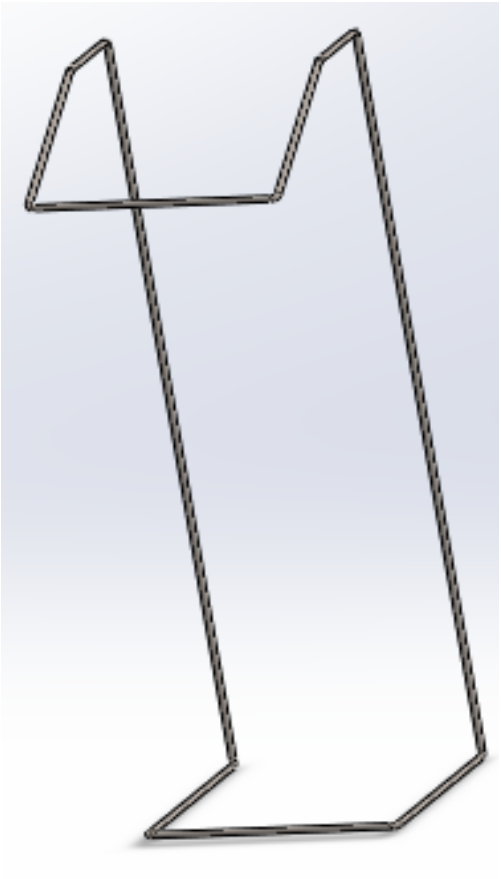

Fig. 20. The 3D support frame

## 4. Conclusions

The contribution of the team that lead to the elaboration of this scientific paper consists in the design of two pieces frequently used in daily life through different procedures and software suites approaching the calculation of quotas for their finalization in the most efficient and feasible way.

## 5. References

[1]. Moise, V., Simionescu, I., Elen, M., Neacsa, M. and Tabara, I. (2015), *Analiza mecanismelor aplicate*, Editura Printech, Bucuresti, pag. 282, ISBN 978-973-718-89 [Cod CNCSIS 54].
[2]. Maican, E. (2006), *SolidWorks - modelare 3D pentru ingineri*, Editura Printech, Bucuresti, pag. 425, ISBN 973-718-544-7 [Cod CNCSIS 54].