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3D MODELING METHOD USED FOR THE DESIGN OF A VEHICLE KNUCKLE

Abstract: *The paper presents the 3D virtual modelling process of a vehicle suspension part using CAD software. The knuckle is the subject of this design due to its geometric complexity. It is part of the rear double wishbone suspension of a rear wheel drive vehicle. There are presented the main steps for the design of the knuckle including the specific features for each operation. The further purpose of this design is the reverse engineering and manufacturing by using 3D printing technology.*

Key words: *design, geometry, knuckle, CAD, manufacturing*

1. INTRODUCTION

The paper describes the steps for designing a knuckle from the rear suspension of a vehicle, using the software Solid Works. It resumes a procedure that is part of a more complex study regarding the reverse engineering of a suspension mechanism using 3D printing technology.

Reverse engineering is used in this particular situation for obtaining an exact copy of an already existing product, created with the new 3D printing technology. The purpose is to compare the mechanical properties of the parts in both classic and 3D printed manufacturing technology cases.

From design and geometric point of view, this paper presents the succession of steps used to generate the 3D model. All mechanical parts are based nowadays on computer aided design that includes work with dedicated software like CATIA, Solid Works, AutoCAD, Inventor end many others [4, 6]. Although each of this software has strong and weak points, the general features have similar effect on generating a solid part.

The usual method for obtaining the geometrical data of a part is measurement. This is often used in computer aided design projects, as in papers [2], [3] and [5].

Another technology is using 3D scanning that was described in papers [1], [7] and [8]. It recreates the surface of the part using a point cloud, obtained with a scanner.

Although this second method is more precise, the first one was preferred, due to software accessibility for creating geometry.

2. PART STUDY AND DIMENSIONING

The knuckle is a mechanical part that connects the wheel to the suspension arms and also damper. The assembly from which the knuckle is part of is a rear double wishbone suspension.

Cylindrical joints that allow suspension vertical motion link the wishbone arms to the knuckle. The wheel is mounted on the knuckle through bearings.

The first step in creating the 3D model of the chosen part for this design is the analysis of the real part. During this step the shape and geometric characteristics are studied, in order to fix the order of applying the software design features.

Further the part is measured from all angles and the technical design is created. The structure of the part can be seen in figure 1.

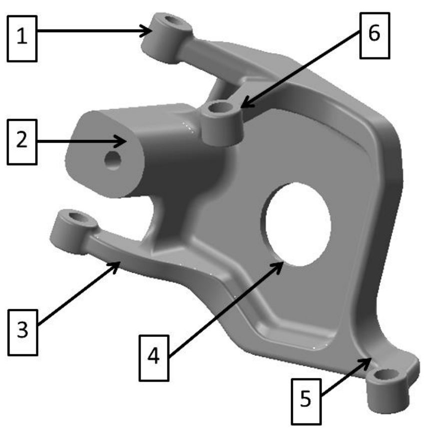

Fig. 1 Main parts of the rear suspension knuckle. 1,3- upper suspension arm joint attachments, 2- damper attachment arm, 4- knuckle body, 5- lower arm attachment, 6- anti-roll bar support

From a structural point of view, the body of the knuckle on which the wheel is mounted through a bearing has upper and lower arms that assure the connection with the other suspension elements. The upper suspension arm is linked to the knuckle arms 1 and 3 through cylindrical joints. Next an additional arm 6 links the wheel assembly to the anti-roll bar. Element 2 is the spring damper assembly support, and the lower arm 5 connects the knuckle to the lower suspension arm.

3. 3D DESIGN STEPS

In the designing process of a product there is often taken into consideration the possibility of readjusting all dimensions, depending on the change of a certain geometrical value. In this case, due to a careful study of the real part and numerous technical drawings, it was decided to approach a nonparametric design.

The software used for this was Solid Works due to its accessible interface.

The first step was creating the arms of the body. In figure 2 there can be observed the sketch containing the dimensions measured on the real body that was extracted from the software. The closed profile in figure 2 was used for en extrusion operation on a 15 mm length.

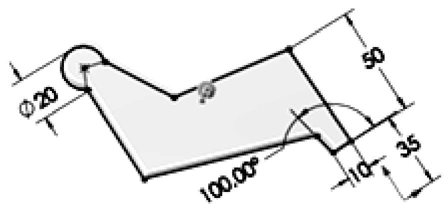

Fig. 2 Sketch of a suspension knuckle arm, including main dimensions

The result for the first two arms, together with the lower arm sketch can be seen in figure 3. The lower arm has a width of 20 mm. The arms have sharp edges and no holes for the joints.

The three arms are generated having each of them one side on a common plane. This plane is then used for creating the sketch of the body.

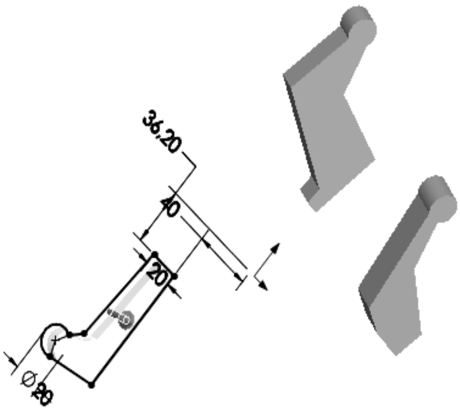

Fig. 3 Dimensioning, sketch and extrude functions for the knuckle arms that link the body to the suspension arms

Next, the sketch of the body is placed on the common plane of the arms. The condition for the extrusion is that the contour to be closed.

In figure 4, there is presented an intermediary extrusion that shapes the body of the knuckle, followed by a second 20 mm extrusion, as shown in figure 5.

After the extrusion the inner shape of the knuckle is obtained by using the 'Shell' command with a 10 mm value.

Figure 6 allows the edges to be headlighted, in order to distinguish the surfaces of the body.

In figure 7 there can be seen the base plane on which the fourth arm was generated, by using the same technique.

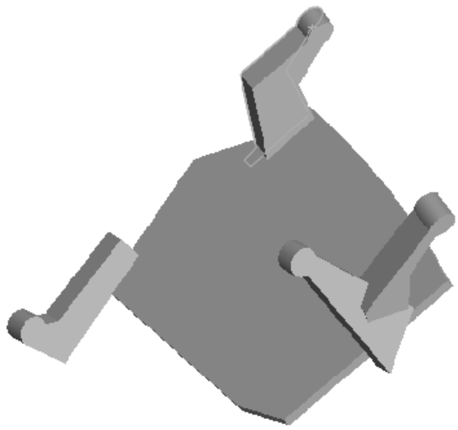

Fig. 4 The knuckle initial shape, obtained only by extrusion

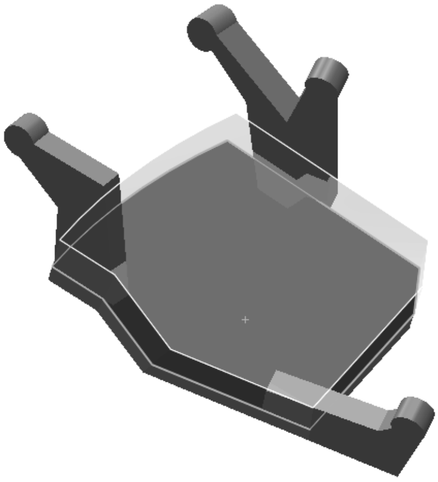

Fig. 5 Shape of the knuckle body without adjustment of the edges shapes

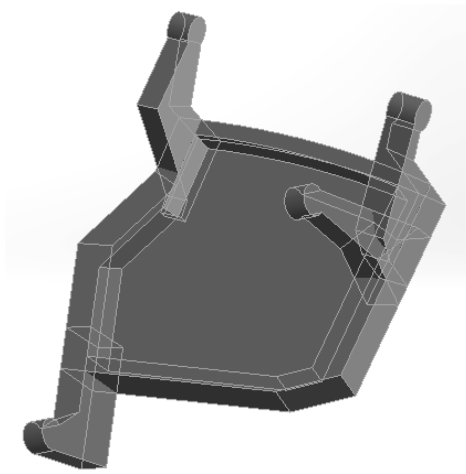

Fig. 6 Creating part of the body of the knuckle with the offset function / wireframe headlight of the part

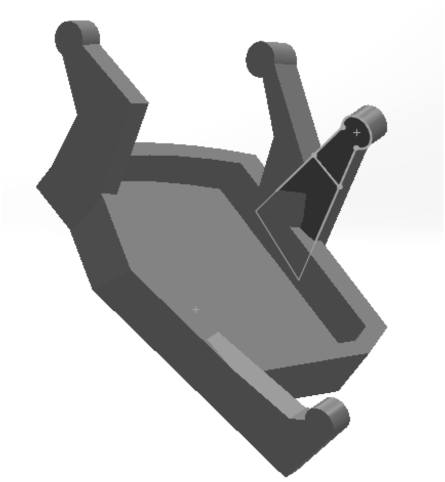

Fig. 7 Generating the shape and dimension of an arm

Other 3D modelling operations are ‘Extrude Cut’, used for creating the holes, as seen in figure 8. This allows an extrusion up to a surface, up to the next element, on a determined distance or through all parts. The input parameter, in this case the diameter of the circle can be readjusted after the operation.

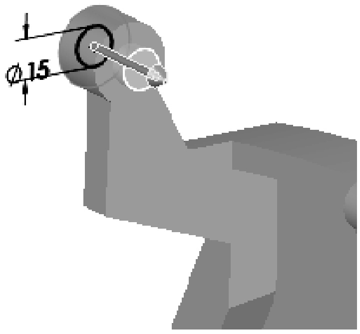

Fig. 8 ‘Extrude cut’ function that creates the interior fillet for linking the spherical suspension joints

Figure 9 gives an example of the adjustment of two neighbouring surfaces through the ‘Fillet’ function with the value of the radius of 20 mm. This name is common in nearly all CAD software, defining the same operation.

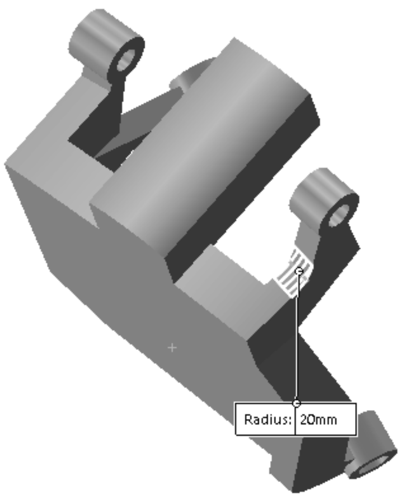

Fig. 9 Example of a fillet with the value of 20 mm radius that connects two intersecting surfaces

Other operations include creating reference geometry, like points, axes or planes starting from existing geometry and using certain distances and geometric characteristics.

One example is given in figure 10, where it is highlighted an operation over the damper support, including a reference plane.

In the same figure, there can be observed that the edges are smoothened. This is obtained by using the ‘Chamfer’ and ‘Fillet’ functions.

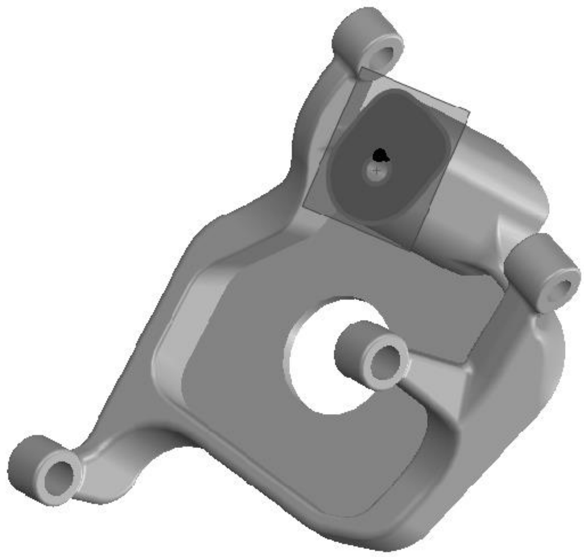

Fig. 10 Sketch on the extrusion plane that generates the damper linking part

Another used function is ‘Variable fillet’. In figure 11 there is given an example of passing from a surface to another by varying the radius from 5 mm to 40 mm.

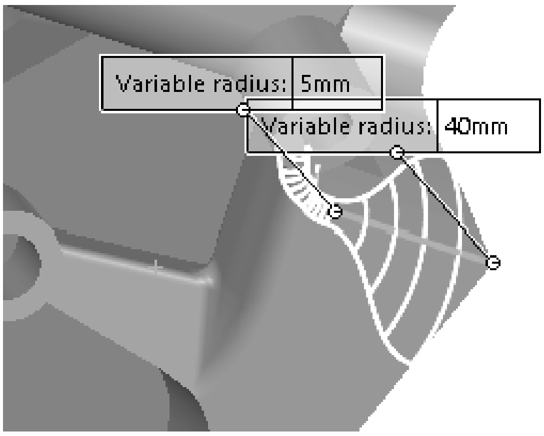

Fig. 11 Variable fillet function that allows joining surfaces with different radii from 5 to 40 mm

An unspecified aspect in the paper is the various angles in different planes that the design has to include, in order to respect the original geometry.

This is where reference geometry plays a crucial role in maintaining the accuracy of the original geometry

4. RESULTS

The result of this process can be seen in figure 12, including also the window of the software. The knuckle is positioned respecting its position on the real model.

The window slightly shows the toolbar on the left that allows accessing all operations and features and other further modifications. This is the determining factor for choosing the Solid Works as software.

From the accuracy point of view, the only differences appear due to measurement or operating the software properly.

The result is a folder that has to be converted into data necessary for 3D Printing.

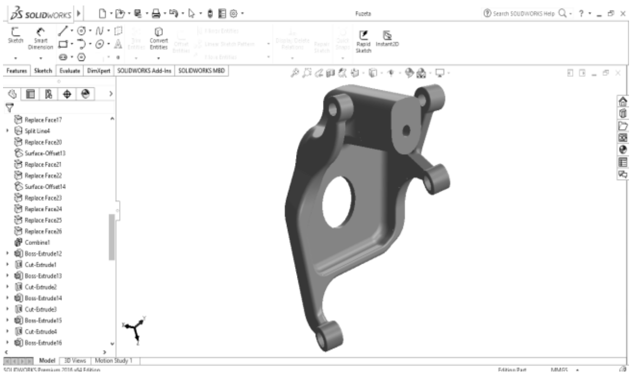

Fig. 12 Final form of the knuckle together with the Solid Works window

8. CONCLUSIONS

The paper is part of a reverse engineering project. It resumes the 3D design of a complex part from the suspension of a medium size vehicle. The basic steps in the design were measurement, establishing operation order, generating the part using Solid Works features, software geometrical measurement and comparison with the real model.

The main skills required for crating complex parts are the ability to extract dimensions from the real model, creating technical design, capacity to establish the steps and operations order and the use of CAD software.

A further possible use of this work is the simulation of suspension loads of the entire suspension system. The FEM software can perform static or dynamic analyses and optimisation of the parts can be operated.

The second use, which is the real one following this design is the remanufacturing using the metal 3D printing technology. After having mechanical parts with exact geometry and using the same material as the original one, but obtained by metal 3D printing, tests can be operated to analyse the mechanical behaviour of two differently manufactured parts.

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