MANAGING SIZE AND POSSIBLE GEOMETRY BEHAVIOR USING PARAMETERIZATION OF SKETCH GEOMETRY WITH A TOOLKIT APPLICATION IN THE AUTODESK INVENTOR PACKAGE

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Abstract: There was introduced three ways to create the chamfer element in package Autodesk Inventor applying the parameterization of sketch geometry with dependences for size management and possible behavior of geometry. Analysis and mathematical processing of reference data allowed using the reference value "Width Across Fields S" as the source parameter of chamfer element.

Keywords: chamfer, width across flats, parameterization, Autodesk Inventor package.

ACM Classification Keywords: I.3.5. Computational Geometry and Object Modeling – Boundary representations Constructive solid geometry (CSG) Curve.

Introduction

The rapid development of computer technology and the creation of a multi-faceted automated system of computer graphics on its basis is the outstanding achievement in the last decades.

Modern computer graphics has a number of different applications, one of which is the creation of dynamic, virtual, multimedia environments and three-dimensional solid models.

The need to create and develop interactive graphical modeling of three-dimensional objects for various functional purposes has caused the development of hardware-independent software packages of parametric 3D-modeling of parts, surfaces, assemblies and drawings for machine builders. The most popular computer-aided design environment is the Autodesk Inventor package -- the Autodesk Company's product. The graphics system with OpenGL support used by the company allows you to work with three-dimensional assembly drawings that contain a lot of components. The Autodesk Inventor package has an intuitive user interface, the DSS design support system, and an excellent EESW multimedia aid system. To create 3D-models of the elements of part designs and structural elements of parts, assemblies and mechanisms, the Autodesk Inventor package uses a large set of tools. The conditions for the development of adaptive and parametric 3D- models have also been created.

The Structural Chamfer Element

In this paper we consider the issue of enhancing the tool options to create a structural chamfer element.

The word "chamfer" originates from the French word "faccete", which means the beveled portions of assemblies, edges, and so on. The material is cut by way of a plane or conical surface. The bulk of chamfers are designed for blunting sharp edges to ensure the safety of these manufacturing operations or use of products and mechanisms [Anurjev, 2006].

In the technical drawings the chamfers and their geometric parameters are indicated in those cases where it is necessary to clearly indicate their presence due to the technical solution. In other cases, chamfers are not specified, but they should be considered in the manufacturing process.

Preferably, as it was mentioned above, chamfers are designed to ensure the safety of interaction between people and products of their industrial activity, but in some cases chamfers are needed as decorative items that are used by designers as a part of the product.

There are chamfers with quite a flat bevel [Anurjev, 2006] that enable the parts to perform functions providing a guaranteed engagement with the corresponding components of assemblies and mechanisms.

If conical chamfers are cut off from faceted surfaces (squares, hexagons), then they automatically show line crossings which are conventionally depicted in drawings by arcs of circles [Anurjev, 2006], [Mikhaylenko , 2001].

Problem Statement

To create a chamfer element in the Autodesk Inventor package, three methods [Guznenkov, 2009] are used: 1. Distance; 2. Distance and angle; 3. Two distances.

When creating a chamfer on hexagonal surfaces, it is convenient to use the reference data "width across flats S" [Anurjev, 2006] as the initial parameter for the chamfer element in the Autodesk Inventor package.

After the analysis and mathematical treatment of the reference data for creating a chamfer element in the Autodesk Inventor package by the second method (distance and angle), we propose three ways of building up chamfers:

      1. *0,27 S, α=30⁰;*   **2.** *0,475 S, α=30⁰;*   **3.** *0,1 S, α=30⁰.*

The First Method to Build up a Chamfer Element in the Autodesk Inventor Package

First, we make a basic sketch using the tools to create a sketch profile – a regular hexagon (Fig. 1). The hexagon is circumscribed around a circle with a diameter specified by the parameter taken from the reference data [Anurjev, 2006] "width across flats S" ($d0=S$).

Then we use the tool "Extrude" to create the extruded element on the basis of the existing sketch profile (Fig. 2).

Next, we make an additional sketch using the tools to create a sketch profile – a triangle (Fig. 3). The additional sketch is created in the symmetry plane of the extruded element and perpendicular to the plane of the base sketch (Fig. 1). In building up the triangle it is necessary to observe the conditions listed below:

-    The angle between the sides at the top of the triangle is equal to 300 [Anurjev, 2006];
-    The vertex of the triangle at an angle of 300 is to be on the axis passing through the center of the inscribed circle $d0 = S$ (Fig. 1, 3);
-    One side of the triangle is to be perpendicular to the axis passing through the center of the inscribed circle $d0 = S$ (Fig. 3);
-    The two sides of the triangle at an angle of 300 are to be at least half the diameter of the circum circle of the extruded element (Fig. 3).
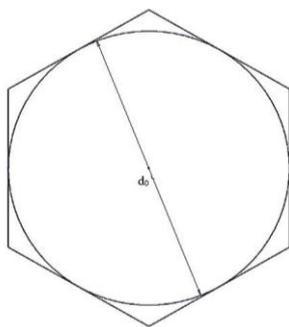
Figure 1. Creating a basic sketch   *Figure 2. Creating the extruded element*
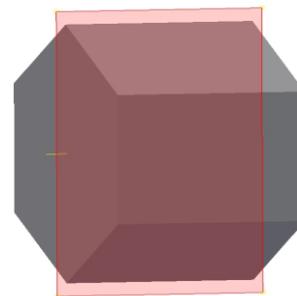
The next step is to place the triangle at a distance from the top (bottom) base of the extruded element at a distance of 0,27 d0 (Fig. 3)

| *a* | *b* |
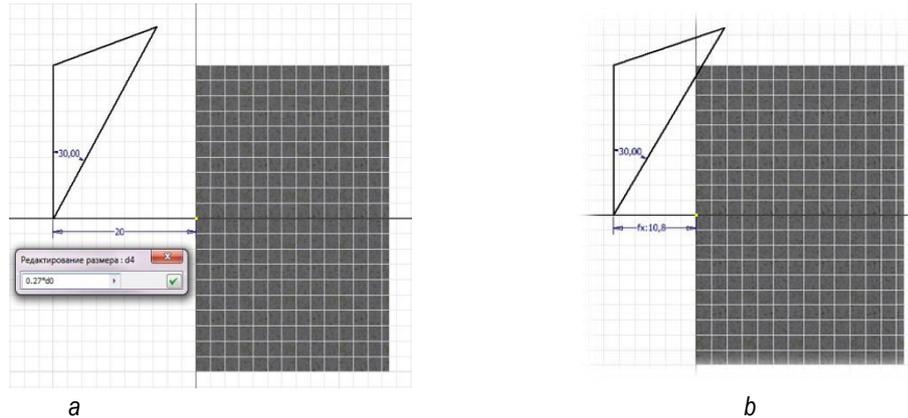
Figure 3. Creating a sketch profile – triangle

For the termination of chamfer building we use the tool "Revolve" to create the element of rotation based on the existing sketch profile (Fig. 3b). The profile is rotated at an angle of 3600. The tool "Revolve" is used with the option "Delete" (Fig. 4).
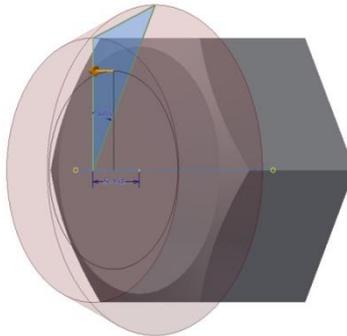
Figure 4. Termination of creating a chamfer element

For creating a sketch, the Autodesk Inventor package provides parameterization, i.e. the sketch geometry with superimposed constrains for managing size and possible geometry behavior. These features allow you to use parameterization for creating a chamfer element. Then, when you create a basic sketch expressing the distance L as the parameter "width across flats S" (Fig. 5) in a sub-sketch, with the given distance of the triangle location from the top (bottom) base of the extruded element, it is sufficient to enter the parameter L in the dialog box "Edit the size" (Fig. 3a).
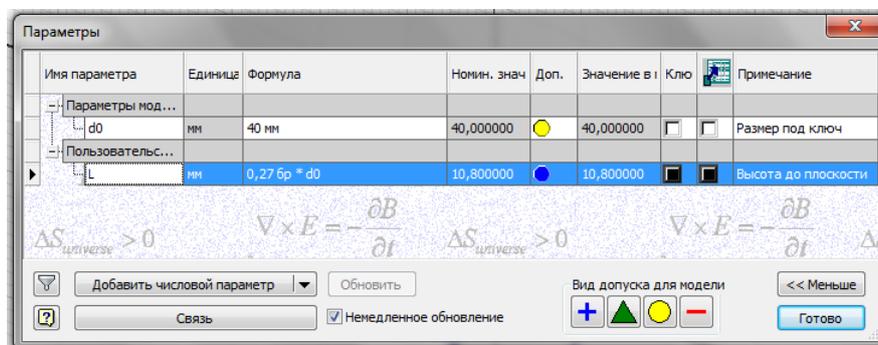
Figure 5. Parameterization in the basic sketch

The Second Method to Build up a Chamfer Element in the Autodesk Inventor Package

First, as it was the case with the first method of building up a chamfer, we create a basic sketch (Fig. 1) using the parameter "width across flats S" and the extruded element (Fig. 2). Then, we create an additional sketch (Fig. 6). In building up the triangle the following conditions must be observed:

- The angle between the sides at the top of the triangle must be equal to 300 [Anurjev, 2006];

- The vertex of the triangle at an angle of 300 must lie on the top (bottom) base (Fig. 6);

- One side of the triangle must be perpendicular to the axis passing through the center of the inscribed circle d0= S (Fig. 1, 6);

- The two sides of the triangle at an angle of 300 must be outside the contour line of the circumcircle of the extruded element (Fig. 6).

| *a* | *b* |
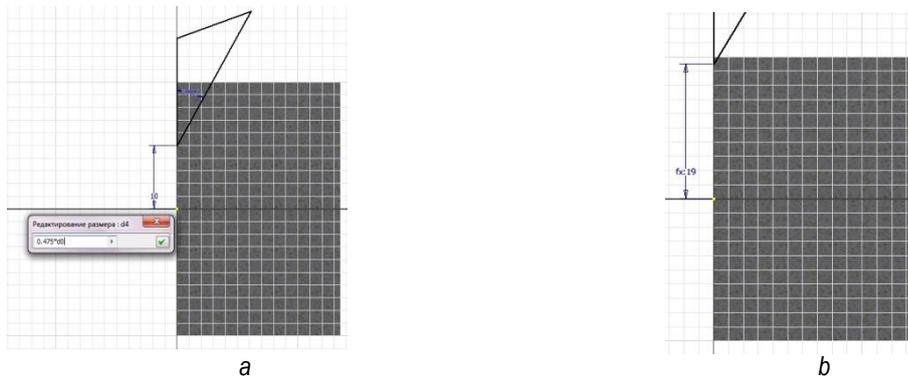
Figure 6. Creating a sketch profile – triangle

The next step is to place the triangle at a distance from the axis passing through the center of the inscribed circle d0 = S (Fig. 1, 6) on the top (bottom) base of the extruded element, at a distance of 0,475 d0 (Fig. 6).

For the termination of building up the chamfer, we use the tool "Revolve" to create the element of rotation on the basis of the existing sketch profile (Fig. 6 b). The profile is rotated at an angle of 360^0. The tool "Revolve" is used with the option "Delete" (Fig. 7).
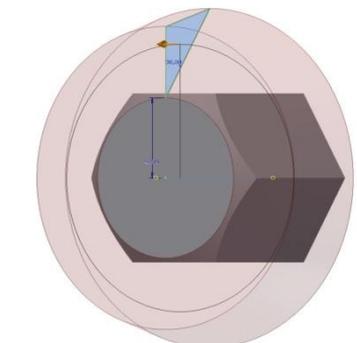
Figure 7. Termination of creating a chamfer element

The second method to build a chamfer in the Autodesk Inventor package also enables the use of parameterization for creating a chamfer element.

Then, when you create a basic sketch expressing the distance M as the parameter "width across flats S» (Fig. 8) in the sub-sketch, with the given distance of the triangle location from the axis passing through the center of the inscribed circle d0 = S, it is sufficient to enter the parameter M in the dialog box "Edit the size" (Fig. 6a).
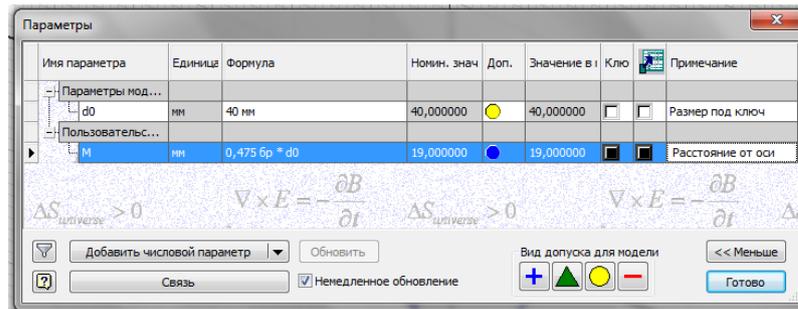
Figure 8. Parameterization in the basic sketch

The Third Method to Build up a Chamfer Element in the Autodesk Inventor Package

The third method has significant differences from the previous ones. Firstly, in the base sketch a circle is built up and described (Fig. 9), secondly, a cylindrical surface is extruded (Fig. 10), and thirdly, there is no need for an additional sketch.

To build up a chamfer by using standard built-in tools provided by the Autodesk Inventor package, we select the tool "Chamfer", the method "distance and angle." In the field "Length" we indicate 0,1 d0,, and in the field "Angle" we indicate the angle of 300 (Fig. 10).
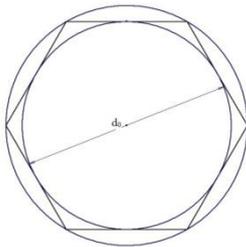
Figure 9. Creating a basic sketch   *Figure 10. Creating the extruded element*
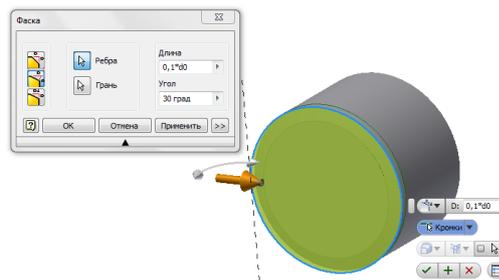
To finalize we use the tool "Extrude" with the option "Delete" (Fig. 11).
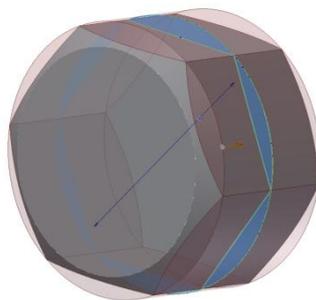
Figure 11. Termination of creating a chamfer element

The third method to build up a chamfer in the Autodesk Inventor package also enables the use of parameterization for creating a chamfer element.

In this case, in creating a basic sketch "Length», z acts as the parameter "width across flats S" (Fig. 12).
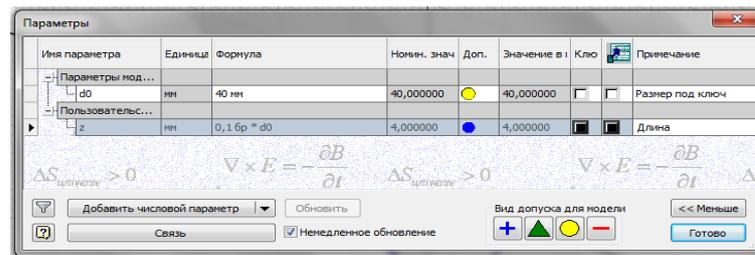
Figure 12. Parameterization in the basic sketch

Next, when we build up a chamfer using standard built-in tools of the Autodesk Inventor package by means of the tool "Chamfer", the method "distance and angle", we indicate the parameter z in the field "Length" (Fig. 10).

Conclusion

The analysis of literary sources has shown that in some cases, when you build up a chamfer element in the Autodesk Inventor package, it is convenient to use the parameter "width across flats S".

None of the options for creating a chamfer element in the Autodesk Inventor package are universal, and they are limited in their application.

The three methods proposed for building up a chamfer element in the Autodesk Inventor package are simple and accessible in creating 3D-models of parts, assemblies and mechanisms;

A method to create a chamfer element in the Autodesk Inventor package is chosen based on the geometric characteristics of parts.

The third method of creating a chamfer element in the Autodesk Inventor package with easy parameterization (z = 0,1 S) and a smaller number of additional constructions is advantageous among all the considered methods.

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