

OnLine tananyag

Interdiszciplináris tudományok

Sorozatszerkesztő: Dr. Balázs László

29. *CAD (English)*



DUNAÚJVÁROSI EGYETEM
UNIVERSITY OF DUNAÚJVÁROS

Table of Content

1. fejezet	5
1.1 Overview of SolidWorks 6	
2. fejezet	15
2.1 The first part	16
2.2 Navigating on the screen	26
2.3 More Features – cuts	28
2.4 The Feature Manager	35
2.5 Features without sketches	36
2.6 Editing the model	41
3. fejezet 55	
3.1 Modeling a ratchet mechanism	56
3.2 Modeling the parts of the mechanism	59
4. fejezet	91
4.1 Housing	92
4.2 Ratchet Wheel	116
4.3 Pawl	131
4.4 Spring	142
5. fejezet	155
5.1 Practices	156

6. fejezet	164
6.1 Modeling a Mug	165
6.2 Practices	199
7. fejezet	204
7.1 HEXAGON CASTLE NUT	204
8. fejezet	207
8.1 Assemblies	208
8.2 Inserting the second component	213
8.3 Inserting more Components	219
9. fejezet	243
9.1 ASSEMBLY OF A MECHANISM I.	244
9.2 ASSEMBLY OF A MECHANISM II.	245
10. fejezet	246
10.1 Test (Assemblies)	247
11. fejezet	248
11.1 Drawings	249
11.2 Creating a projection	253
11.3 Creating a projected view	260
11.4 Creating a sectional view	271
11.5 Dimensioning	278
11.6 Editing the dimensions	281
11.7 Editing the drawing sheet	286
11.8 Creating a detail view	290
11.9 Creating a breakout	294
11.10 Creating a shortened view	297
11.11 Creating a pure section	299

12. fejezet	303
12.1 Task I.	304
12.2 Task II.	305
12.3 Task III.	306
13. fejezet	307
13.1 Creating drawings of assemblies00	308
13.2 Creating an item list (Bill of Material)	310
13.3 Ballooning	313
13.4 Editing the item list	316
14. fejezet	329
14.1 Practices	330
15. fejezet	331
15.1 Test (Drawings)	332
16. fejezet	334
16.1 Parts and drawings	335
16. 2 Parts, assemblies and drawings	336
16.3 Synthetical practices	337
Drawings	338

1. fejezet

1.1 Overview of *SolidWorks*

About CAD-systems in general

CAD

The title of the subject (CAD) is an abbreviation. Although this abbreviation is widely known in the engineering practice and maybe sounds familiar in the everyday life, many people don't know that it can have two resolutions: *Computer Aided Drafting* and *Computer Aided Design*. The goal of both techniques is to help the engineering work, but their approaches are significantly different.

COMPUTER AIDED DRAFTING

Computer Aided Drafting can be classified as the traditional technique. The history of this technique is surprisingly old. The first software which realized the fundamental functions of *Computer Aided Drafting* was SketchPad in 1963. This software interacted with the user on a graphical interface. The user was able to draw *Lines*, circles and other planar shapes. It was also possible to manipulate the drawn objects: copy or move them, create the intersection of each other, resize them etc.

Sketchpad handled two dimensional objects – it was a 2D software. Later as the computer hardware and the graphical algorithms were improved, *Computer Aided Drafting* softwares were prepared to create, describe and manipulate spatial (three dimensional, 3D) objects (eg. cylinders, cubes, spheres, cones, torii etc) as well, but this remained just a possibility – the main goal of such softwares is to create 2D engineering drawings.

Nowadays the most known representatives of *Computer Aided Drafting* techniques are *AutoCAD*, *CADKey*, *MicroStation* etc.

COMPUTER AIDED DESIGN

When an engineer designs a part or a machine he imagines it as a spatial object. In the past the only possibility to record his thoughts was to draw the projections of the imagined object on a drawing sheet. This situation was changed by the appearance of *Computer Aided Design* softwares. The aim of these softwares is to create and manipulate three dimensional objects. The user builds the geometrical model of a machine part and if it is necessary he can create the engineering drawing of the part.

THE USES OF 3D MODELS

Creating 2D drawing documentation is not the only possible use of 3D geometrical models. It is possible in a modern manufacturing environment that no paper based drawing is created about a product during its lifecycle – only electronic documents are transferred between the stations of the design and the manufacture.

- The model of the part can be analysed using the *Finite Element Method*, to determine what stresses and strains occur as the results of the loads (forces, torques) acting on the part. If the analysis shows that some region of the part is overstressed the geometry of the part can be easily modified and the analysis can be performed again.
- A physically existing instance of the part can be produced on a *Rapid Prototyping equipment*, eg. to see if the part is handy or not.
- The geometrical model can be processed in a Computer Aided Manufacture (CAM) software to generate the controlling program of a *Computer Numerical Control (CNC)* machine which manufactures the part.
- The manufactured part and the geometrical model can be compared on a CNC measuring equipment.
- Of course the engineering documentation of the part can be created using the *Computer Aided Design* software itself.

Because the *Computer Aided Design* technique is more close to the engineering train of thought this technique will be reviewed during the subject.

PROPERTIES OF 3D CAD-SYSTEMS

There are many 3D CAD softwares on the market (eg. *ProEngineer*, *Catia*, *Inventor*, *Solid Edge*, *SolidWorks*, *TopSolid* etc). The functionality of these softwares is very similar - the differences between them appear typically on the user interface level.

These softwares are usually mentioned as 3D, engineering, feature-based, parametric and associative ones.

3D

This property was previously mentioned. The user creates three dimensional geometrical models of machine parts and assemblies.

Engineering

These softwares are developed to help the work of mechanical engineers. Using them it is easy to create the typical *Features* of engineering parts like bored holes, slots, grooves, chamfers, fillets, etc. Sheet metal parts and revolved bodies can be created by a few clicks. Screws, bolts, nuts, washers, roller bearings can be inserted from a toolbox to an *Assembly*.

Feature based

Features are the building blocks of a part model. Some *Features* add material to the model while others remove. Sometimes a feature is originated as a sketch (eg. adding/removing material by extrusion or rotation); other features, such as fillets, chamfers or shells are created without sketching.

The model of the part is the sequence of its features. The *Features* can be deleted, modified or reordered at any time during the modeling process.

Parametric

Let's suppose that a cylinder is to be created. In a Computer Aided Drafting software this can be done by extruding a circle. The result is the surface model of the cylinder (*Fig1*). The software stores the coordinates of the cornerpoints (vertices) of the object and the necessary topological information – which points are connected by edges and which edges are bordering the facets of the object. The information, that the object is a cylinder is not stored, so if it is necessary to modify the model, it can be performed only by dragging the vertices.

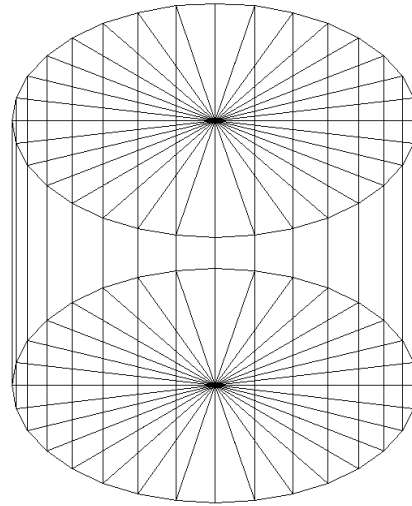


Fig1

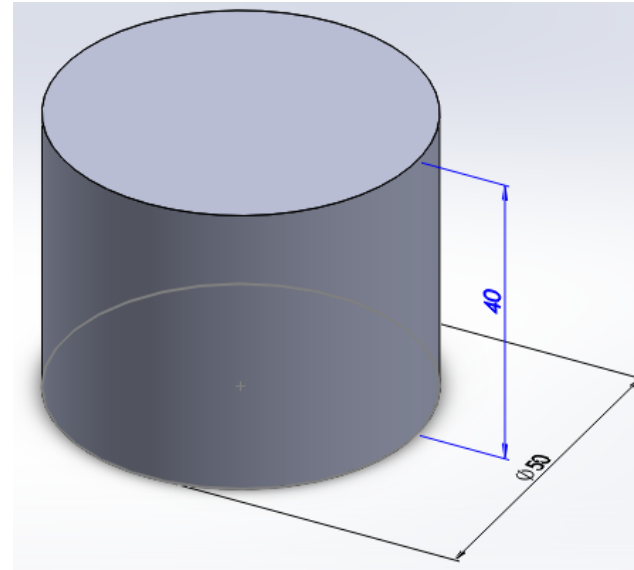


Fig2

The same task in a *Computer Aided Design* software is preformed by sketching a circle on an appropriate plane then extruding the sketch to the necessary height (Fig2). The software generates the boundary representation to display the cylinder, but in this case the diameter of the base circle and the height of the extrusion are stored. These numerical values are the parameters of the model. Modification of a part can be performed by overwriting its parameters.

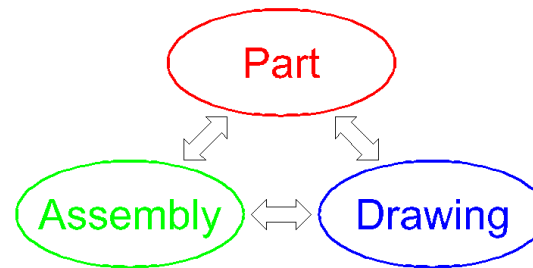
Associative

A 3D CAD software typically handles at least three kind of documents – parts, assemblies and drawings.

- Parts (as it was previously mentioned) are the combinations of their features.
- An *Assembly* is the combination of several parts or subassemblies.
- A drawing document contains projections of parts or assemblies.

These documents are associated. If a part is modified, the instance of the part built into an *Assembly* will be also modified automatically. The previously created drawing of a part or an *Assembly* will also show the changes of the design. Moreover editing a drawing document means the modification of the described part or *Assembly* (Fig3).

Fig3



Introduction to the use of SolidWorks

During the course the methods and the techniques of *Computer Aided Design* will be acquired through the use of *SolidWorks*. *SolidWorks* is a typical 3D CAD software. *SolidWorks* is easy to learn but it can be used also to design complicated equipments. *SolidWorks* is developed for Windows environment.

PURCHASING SOLIDWORKS

To practice the CAD techniques using *SolidWorks* download the fully functional students' edition of the software. Visit the http://www.SolidWorks.com/sw/education/SDL_form.html page on *SolidWorks*' site. Fill the *Student Acces Form* according to Fig4.

Fig4

School License ID, SDK-ID or SEK-ID: 93201EFT

Are you a: Student

First Name: Your First Name

Last Name: Your Last Name

School Name: Budapesti Műszaki és Gazdaságtudomány

School Type: Graduate

Major: Mechanical

Month of Graduation: December

Year of Graduation: 2015

City: Budapest

State:

Postal Code: 1111

Country: Hungary

Version: 2014-2015

VAR Name(if available): eurosolid

Email Address: Your Email Address

Request Download

The above data was provided by the Hungarian retailer of *SolidWorks* (Eurosolid Ltd). The full content of the *School Name* field: Budapesti Műszaki és Gazdaságtudományi Egyetem.

After providing data press the *Request Download* button. Accept the license agreement then select the appropriate version to start download (Fig5).

Fig5

Student Access Form

 **Download** Win7 32-bit

 **Download** Win7/Win8 64-bit

[Click here](#) for instructions if your browser security settings are blocking this download.

The links above download the SolidWorks Student Access product requested.

This is a very large download of up to 8 GB: a broadband internet connection is required.

Download instructions:

- Before proceeding with the download and installation:
 - Please back up any SolidWorks data you may have generated from any prior use of other versions of SolidWorks on your machine, then uninstall any prior versions of SolidWorks before proceeding.
Please note: after you download and install the SolidWorks Student Access product on your computer, older versions of SolidWorks will no longer operate.
 - Make certain you have at least 8 GB of available disk space to download, install, and begin using the product
Please note, your download will fail if there is not enough disk space for this operation. Files are copied to a temporary directory location during download. Check your browser settings for the temporary directory location, and insure there is sufficient disk space available
- After clicking the appropriate link for the 32-bit or 64-bit version of your requested Student Access product, you will be asked to download **the SolidWorks Installation Manager** (about 20 MB).
- Please run the SolidWorks Installation Manager, which will guide you through the complete download and installation process.
- When the download is complete, click UNZIP then follow the installation instructions.

 Print |  Email

Start the downloaded *SolidWorks* Installation Manager and follow its instructions.

RUNNING SOLIDWORKS

After the succesful installation the starting icon of *SolidWorks* appears on the *Desktop* (Fig6). Double-clicking on it starts the software.

Fig6



THE USER INTERFACE

After a short while an empty *SolidWorks* window appears on the screen (Fig7). Although no document is opened in it, it can be realized that because *SolidWorks* is a *Windows*-based software we will work in a familiar environment.

Fig7



The top row of the screen shows the usual Main drop-down menu, and also the Standard toolbar can be seen with the common tools as *New, Open, Save, Print etc.*

2. fejezet

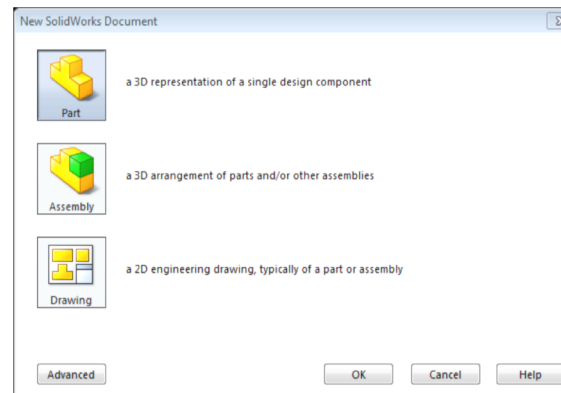
2.1 The first part

In this practice we overview the fundamental workflow of part modeling.

A part is a single 3D object made up of features. A part can become a component in an *Assembly*, and it can be represented in 2D in a drawing. Examples of parts are bolt, *Pin*, plate, and so on. The extension for a *SolidWorks* part file name is *.SLDPRT*.

Create a new document (*File/New...*). This will be the most fundamental kind of document in *SolidWorks* - a part. Choose the Part button on the dialog box and press *OK* (*Fig1*).

Fig1

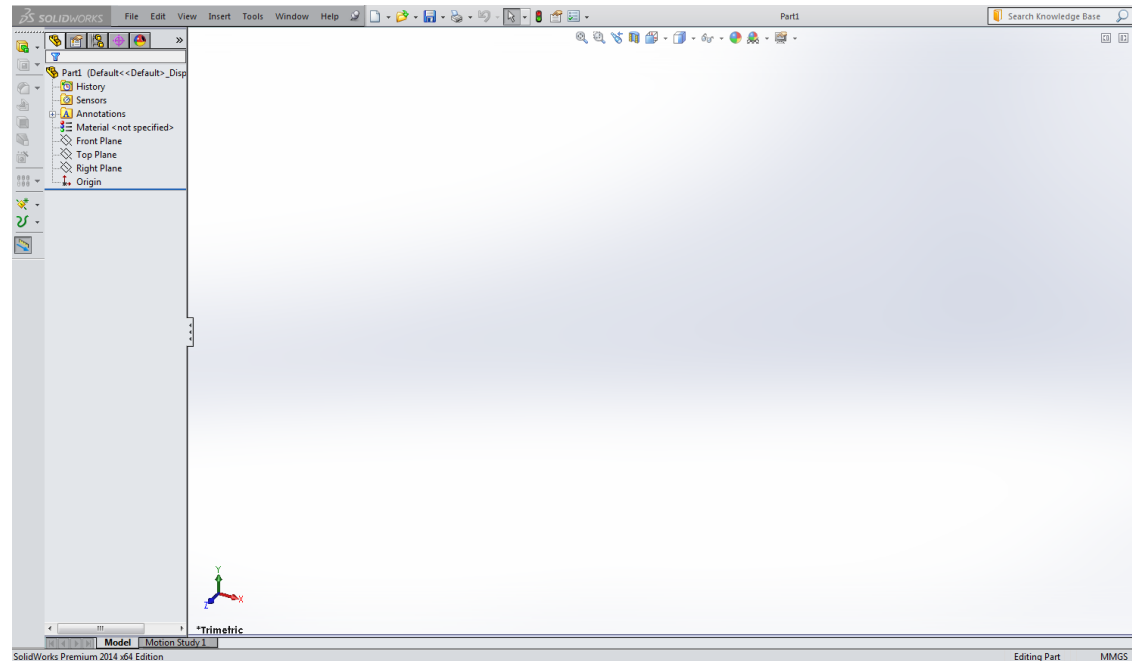


An empty part document window appears. Watch the typical areas of the screen (*Fig2*):

- The large central area is the *Drawing Area*. This is where the picture of the part will be displayed.
- The *Feature Manager* is located to the left of the *Drawing Area*. This is where the *Features* and every other data of the part are listed. The visual representation and the list of the *Features* are equivalent – every detail of the part is available graphically or in the *Feature Manager*. Because the document is empty, no *Features* are listed yet. Notice that three coordinate planes (*Top*, *Front* and *Right*) and their common point (*Origin*) belong to the part. Hover the mouse over these reference elements in the *Feature Manager* to see their *Orientation* on the *Drawing Area*.

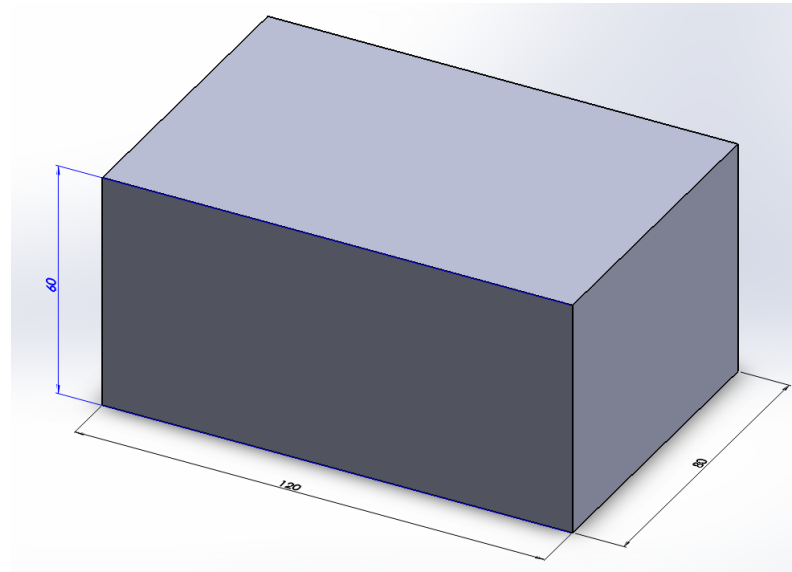
- On the *Right* side of the screen the *Sketch* toolbar can be found. This toolbar is usually available at this place in every *SolidWorks* document.
- In case of part documents the *Features* toolbar is opened on the left side of the screen.
- Below the *Drawing Area* the *Status Line* give us information about the status of the design.
- At the top edge of the *Drawing Area* the *View (Heads-Up)* toolbar makes the acces of the viewing tools easy.

Fig2



Remarks:

- If one of the toolbars mentioned above is not opened, then *Right*-click somewhere outside the document window and turn on displaying of the missing toolbar.
- There is a different layout for the tools. If you prefer, it is possible to choose the *Command Manager* by *Right*-clicking on the toolbar area and selecting *Command Manager*. The figures of this workbook will follow the layout on *Fig2*.

*The first feature**Fig3*

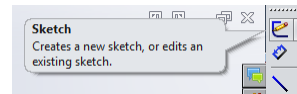
Let's create the cuboid on *Fig3*! This will be the first feature of the part. It means addition of material by extruding a sketch.

The sketch will contain a rectangle.

THE FIRST SKETCH

Before starting to sketch we have to choose the sketch plane. In this case all of the default planes are suitable. Select the *Top Plane* in the *Feature Manager* and click the Sketch tool (the first one in the *Sketch toolbar*) (*Fig4*).

Fig4

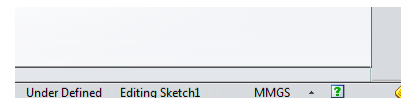


The sketch plane turns in front of us to provide an undistorted view of the sketch. Notice that there are several signs on the screen indicating that we are in a sketch:

Fig5



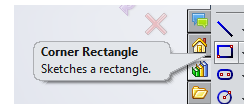
Fig6



- The Sketch button is pressed (*Fig5*).
- The Confirmation Corner appeared in the upper *Right* corner of the Drawing Area (*Fig5*). Using it the sketch can be left either *keep*ing or discarding the changes.
- The Status *Line* indicates that Sketch1 is being edited (*Fig6*).

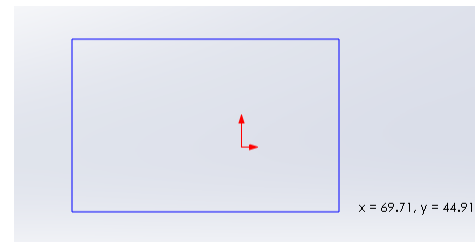
The sketch should contain a rectangle. Click on the *Corner Rectangle* tool in the Sketch toolbar (*Fig7*)!

Fig7



The pointer and the *Corner Rectangle* button show that we are sketching a rectangle. Point and click on the *Drawing Area* to define the rectangle by its two opposite corners (*Fig8*).

Fig8



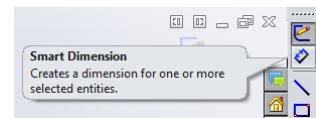
Important! Although *SolidWorks* displays the coordinates, it is not necessary to take care of them. Remember that this is just a sketch! It is not important to draw precisely. The result does not depend on the accuracy of the sketch.

After placing the second corner point, the rectangle appears on the screen. Press Esc on the keyboard to stop sketching more rectangles.

Observe the colour of the rectangle! *Right* after sketching it its color was pale blue indicating that the sketched elements are selected. After pressing Esc the colour changed to blue. The meaning of this colour is that the sketch entity is under defined. The status *Line* also communicates this information.

It is easy to realize what information is missing. The size of the rectangle is not defined yet. Select the *Smart Dimension* tool (Fig9).

Fig9



This tool makes the dimensioning of the sketch possible. The type of the dimension depends on the selected element (eg. length of a *Line*, radius of an arc, diameter of a circle, distance of *Lines*/circles, angle of *Lines* etc.). Notice that a supplementary symbol at the pointer reminds us of being in dimensioning mode.

Click on one of the horizontal edges of the rectangle. Because the selected element is a *Line* SolidWorks realizes that we want to define the length of this *Line* (Fig10).

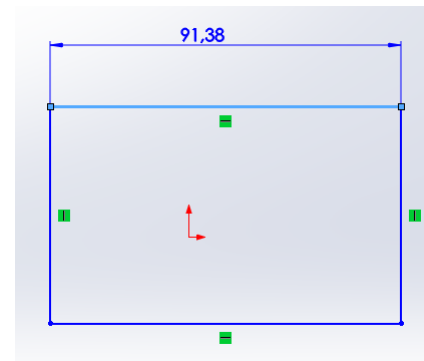


Fig10

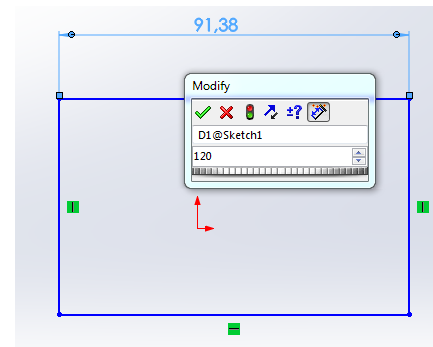


Fig11

Click on an appropriate location to place the dimension number! The actual position, ie. the distance of the dimension *Line* from the rectangle or the position of the dimension number between the extension *Lines* is not important. Remember! This is just a sketch. After placing the dimension number type the necessary value into the dialog box (*Fig11*).

Perform the dimensioning for one of the vertical edges and press Esc to stop dimensioning!

MAKING THE SKETCH FULLY DEFINED

Although the width and the height of the rectangle is defined, its colour remained blue – the rectangle is under defined. It is easy to realize what information is missing. If it is not self-explaining then click on one of the blue entities and try to drag it. The position of the rectangle is not known. The reference to which this position can be defined is the origin of the sketch (Every sketch has its own origin which is the perpendicular projection of the origin of the part on the sketch plane). Click on the lower left corner of the rectangle and drag it over the origin. As the corner point gets close enough to the origin it “jumps” onto it, and a small yellow symbol appears indicating that a relation will be created after releasing the mouse button (*Fig12*). This way the corner point can be defined to be *Coincident* with the origin (*Fig13*).

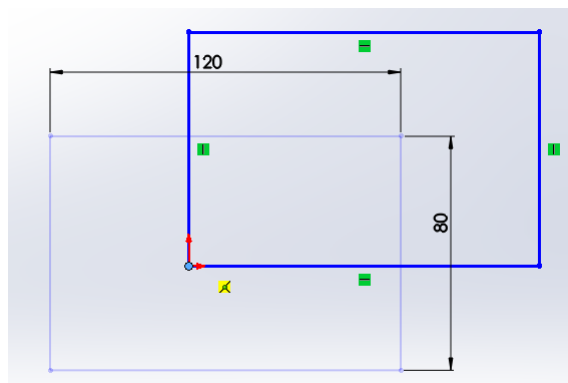


Fig12

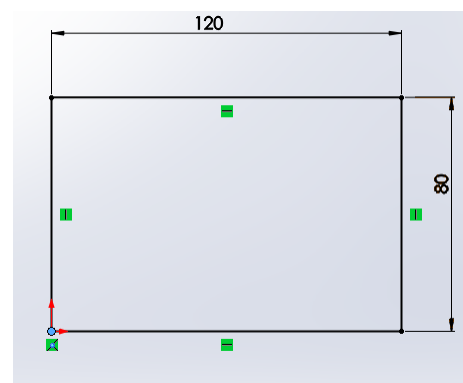


Fig13

Notice that the colour of the rectangle changed to black – it means that the elements are defined. The Status *Line* also indicates this information.

The sketch contains not only the edges of the rectangle and the dimensions but five small green symbols appear throughout the sketch. These symbols are called relations.

If you hover the pointer over these symbols a small pop-up text shows information about the symbolized relation.

The contents of the sketch:

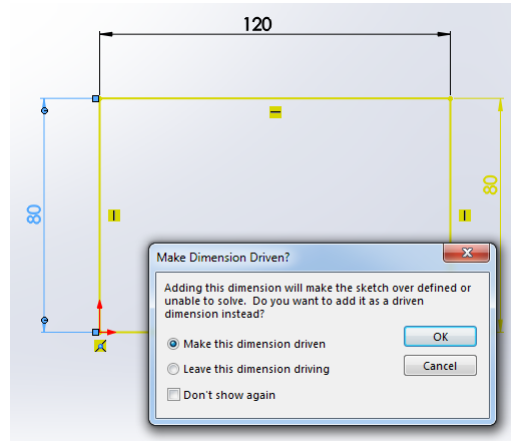
- Four *Lines*;
- Two dimensions;
- Five relations:
 - o two horizontalities,
 - o two verticalities and
 - o one coincidence between the corner of the rectangle and the origin of the sketch.

The horizontal/vertical relations were created automatically during sketching the rectangle. These relations prevent the rotation of the rectangle.

The content of the sketch is defined by the sketched elements (entities), the dimensions and the relations between the elements of the sketch.

If you try to add more dimensions or relations to the sketch it can turn to be *Over Defined*, which is a situation to avoid (*Fig14*). Overdefining dimensions must be deleted or set to driven (informative) dimensions.

Fig14

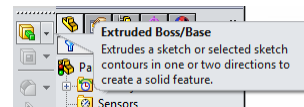


As you can see on Fig14 the colour of overdefining dimensions or relations is yellow. If you have created overdefining or driven dimensions delete them (click on them and press *Delete* on the keyboard) until the sketch turns to black (*Fully Defined*) again.

EXTRUDING THE SKETCH

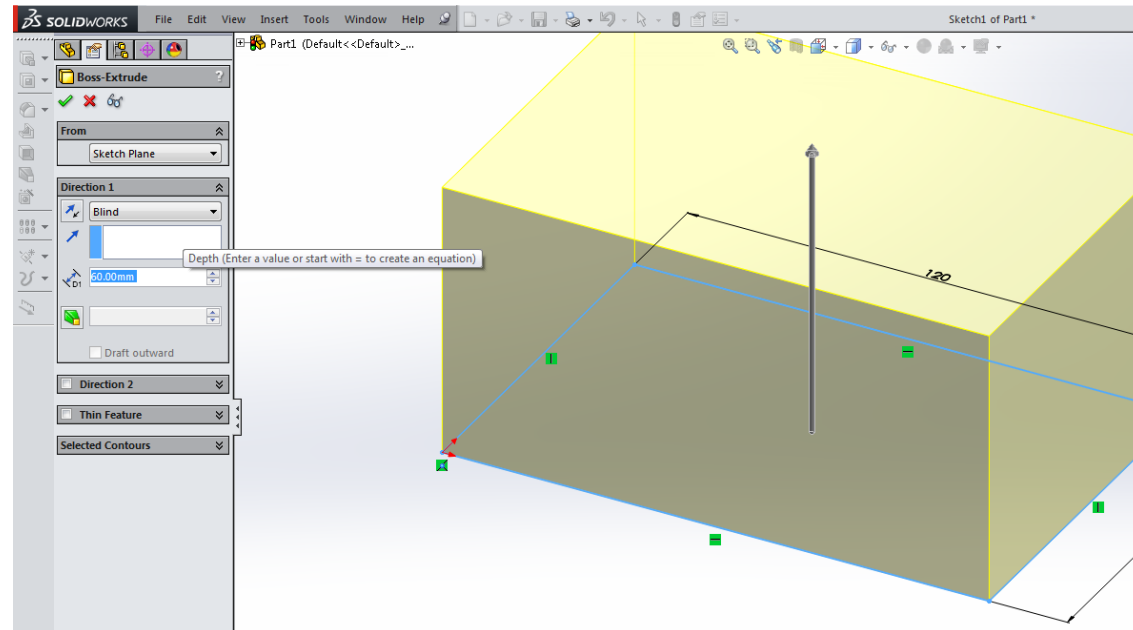
If the sketch is fully defined then it is time to create a feature from it. Select the Extruded Boss/Base tool in the *Features* toolbar (Fig15)!

Fig15



After selecting the tool the viewing direction returns to the original axonometric view so we are able to see the result of the extrusion (Fig16).

Fig16



Type the necessary height into the *Depth* field of the *Boss-Extrude* dialog box and then click the green chekmark (OK)!

The cuboid is finished.

2.2 Navigating on the screen

Let's see what the most frequent tools are, which make possible to modify the view of the part. These tools are collected in the View (*Heads-Up*) toolbar at the top edge of the Viewing Area.

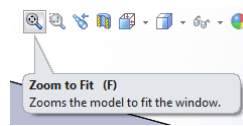


Fig1

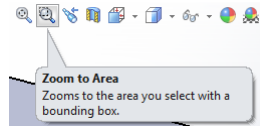


Fig2

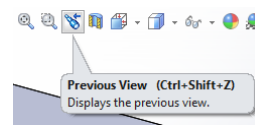


Fig3

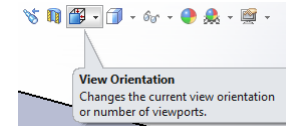
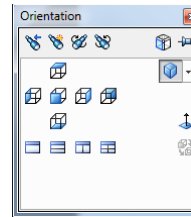


Fig4

- The Zoom to Fit tool finds the best scaling factor to display the part in its entirety (Fig1).
- Using the Zoom to Area tool a rectangular box can be marked on the screen to zoom on (Fig2).
- The Previous View tool returns to the preceding view (Fig3).
- The View Orientation tool (Fig4) brings up the Orientation dialog box in which the six main views (*Front, Top, Left, Right, Bottom and Back*) and three axonometric views (*Isometric, Dimetric, Trimetric*) can be selected. The same dialog box can be reached by pressing Space on the keyboard (Fig5).

Fig5



The zooming can be changed quickly by the mouse if it has a wheel. Rolling the wheel away from us means zooming out - rolling toward us means zooming in.

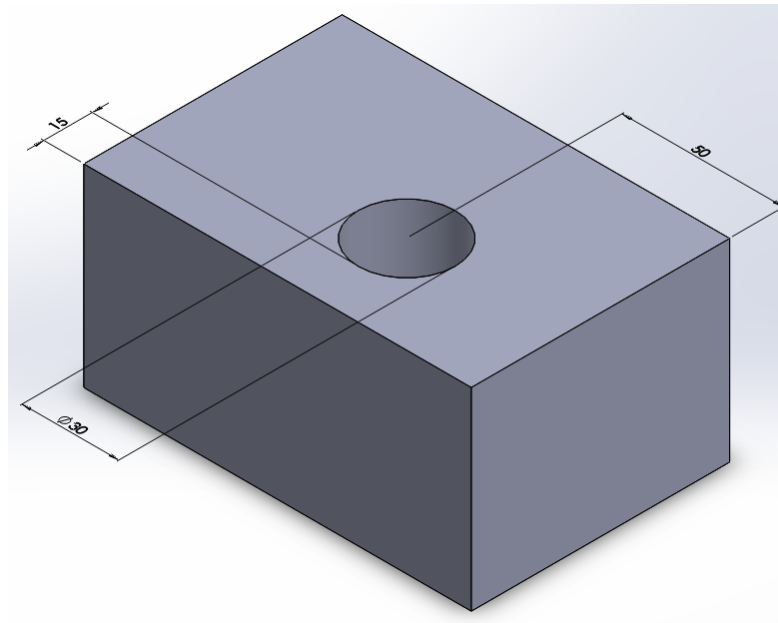
Pressing the middle mouse button or the mouse wheel can be used to change the viewing direction quickly.

Try out the functions mentioned above! If you became disorientated the *Orientation* dialog helps to return to the standard views.

2.3 More Features – cuts

A part typically consists of several features. Let the next feature be a through-hole with dimensions according to Fig1. This feature is a material removal which requires a sketch. Notice that in the case of one of the dimensions the minimum distance between the hole and the outer surface of the part (15 mm) is prescribed.

Fig1



There are three choices for the sketch plane:

- The sketch can be created on the *Top Plane*.
- The sketch plane can be the top face of the part.
- The sketch plane can be the bottom face of the part.

It is always advisable to sketch on the surface of the part therefore the sketch plane will be the top face of the part. Select this face and start sketching. Notice that the viewing direction remains the same. If you prefer to see the undistorted view of the sketch you can select the *Normal To* tool in the *Orientation* dialog box (Fig2).

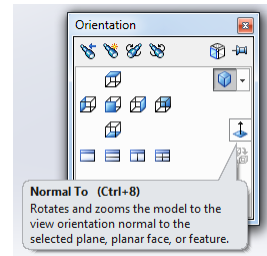


Fig2

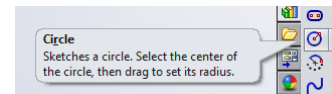


Fig3

The sketch will contain a circle. Click on the Circle tool in the Sketch toolbar (Fig3) then select the center of the circle and drag to define its approximate radius.

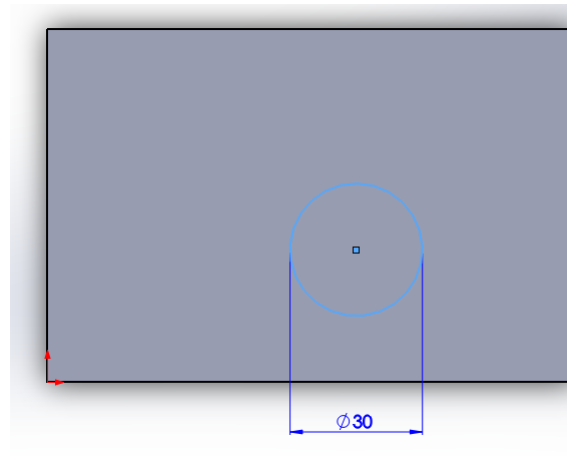


Fig4

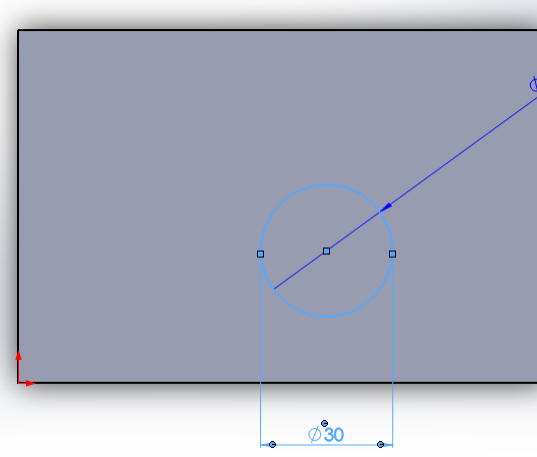


Fig5

Perform the dimensioning of the diameter (Smart Dimension)! Click on the circle! *SolidWorks* realizes that a selected element is a complete circle so the dimension will be a diametral dimension. Place the dimension number and type in the numerical value (*Fig4*).

The next dimension defines the horizontal placement of the circle. Click on the circle again but instead of placing the dimension number select the appropriate edge of the part (*Fig5*). The selected elements define the distance of the circle and the selected edge. By default this distance is interpreted as the central distance. Place the dimension number and type in the value for the distance (*Fig6*).

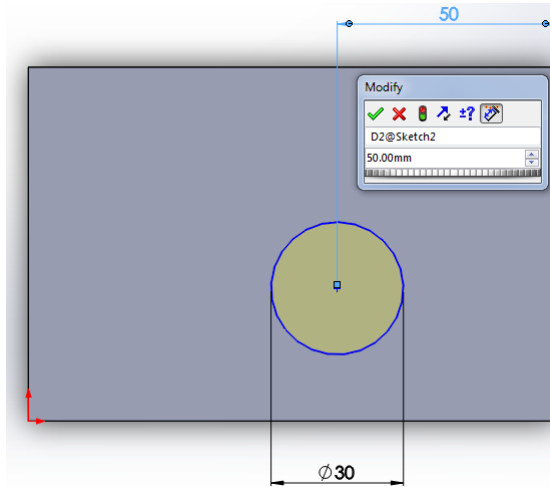


Fig6

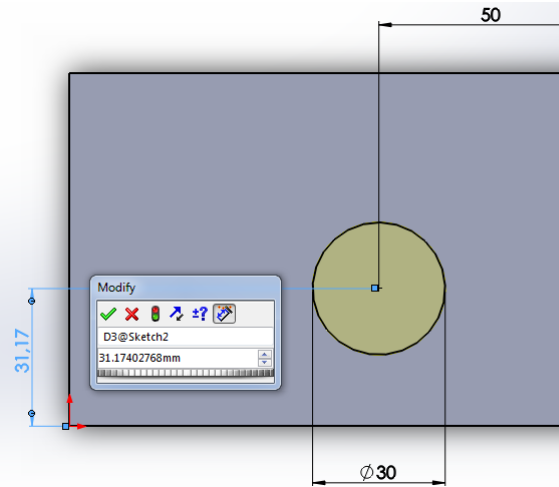
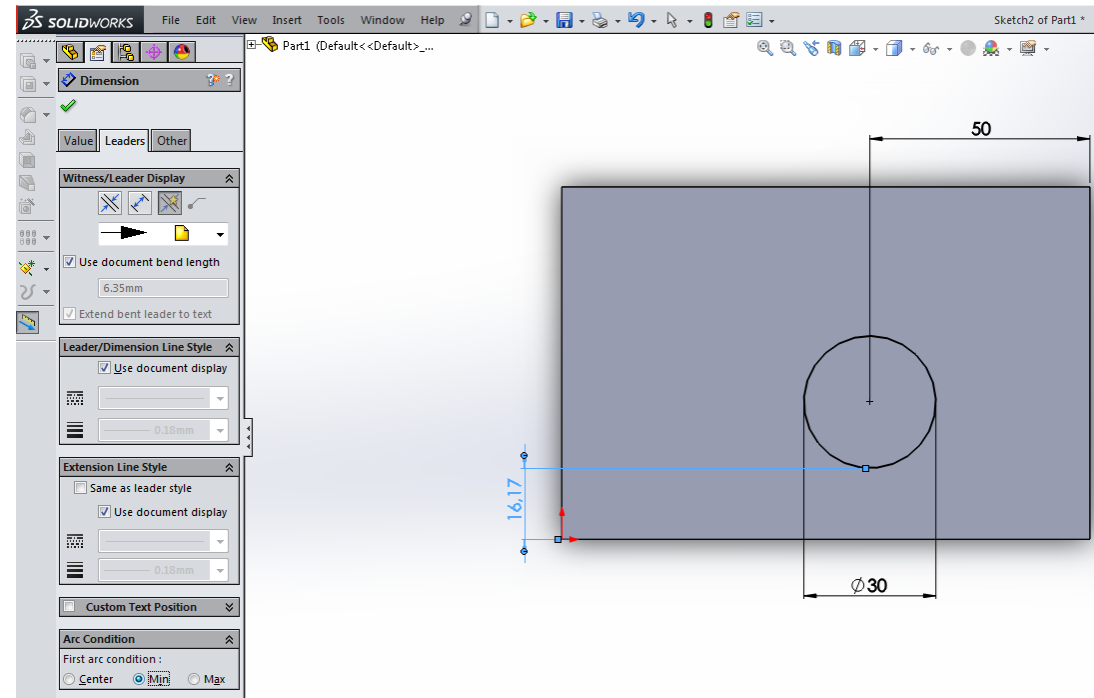


Fig7

The remaining dimension is a bit more complicated to define. Click on the circle (not on its center point) and click on the corresponding edge of the part! Place the dimension number and accept the value (Fig7). This value is wrong yet, but before defining it is necessary to change the interpretation of the dimension.

The dimension is selected – its *Properties* appear on the left side of the screen at the place of the *Feature Manager*. These *Properties* are distributed on three tabs. Select the middle one (Leaders)!

Fig8



Choose the Min option at the bottom of the tab to change the interpretation of the dimension (Fig8)! After changing the interpretation double-click on the dimension number and type in the correct value.

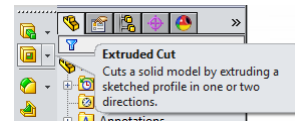
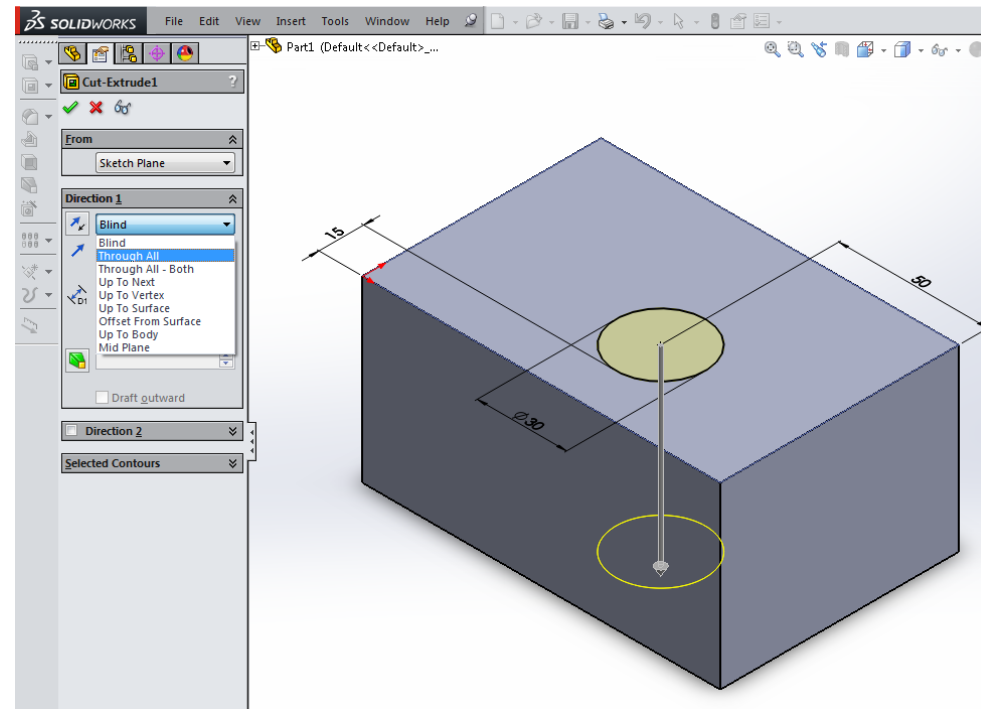


Fig9

The sketch is fully defined it is time to create a feature from it. Return to the isometric viewing direction to see how the result of the feature creation will look like. Find the *Extruded Cut* tool and click on it (Fig9).

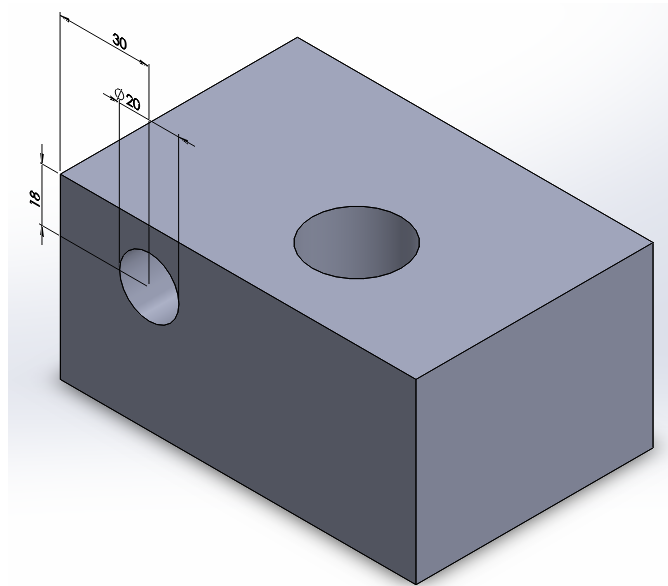
Fig10



Remember that the hole is intended to be a through-hole so change the depth of the cut from Blind to *Through All* (Fig10). This setting makes sure that the hole will remain a through-hole even if the height of the part was changed. After accepting the settings of the dialog box the hole is created.

Create another through-hole according to Fig11!

Fig11



2.4 The Feature Manager

Take a look at the left side of the screen! The *Feature Manager* contains entries for the previously created features. Clicking on the small “+” signs in front of the names of the *Features* the sketches belonging to the *Features* can be revealed. Hovering the pointer over the name of a feature or a sketch the corresponding element gets highlighted on the *Drawing Area* (Fig1). If you click on a face of the part the owning feature is highlighted in the *Feature Manager*. Every feature can be selected either on the *Drawing Area* or in the *Feature Manager*.

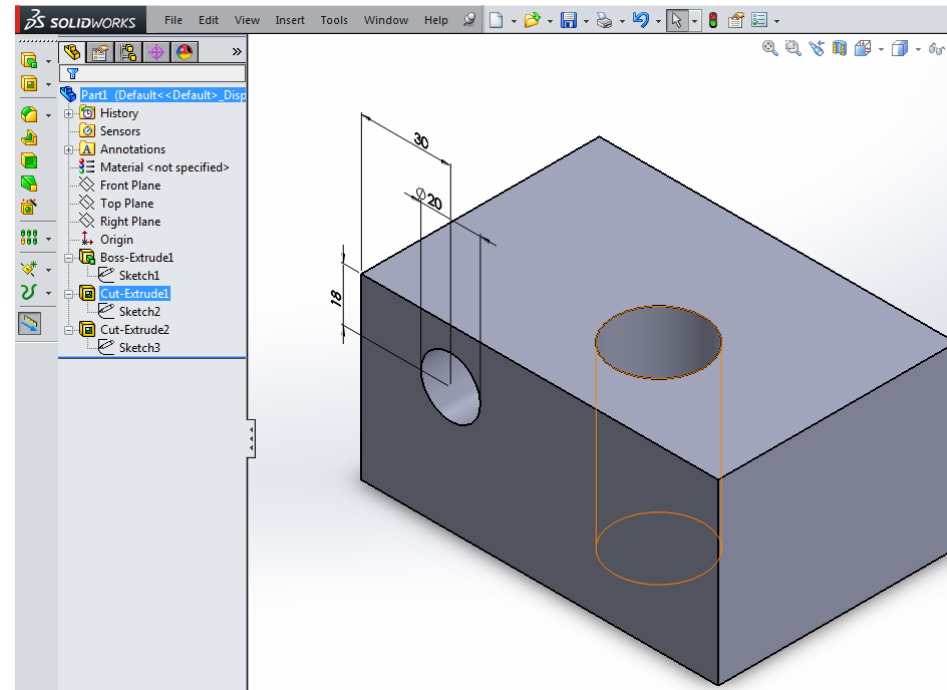


Fig1

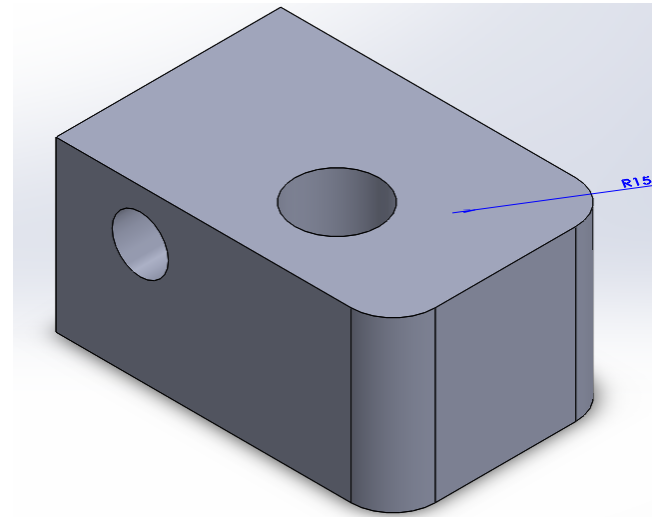
2.5 Features without sketches

All of the previously created *Features* required sketching. There are *Features* for which sketching is not necessary – it is enough to select some edges or faces of the part and to define the necessary data in a dialog box.

FILLET

Fillets are typical *Features* without sketches. Round two edges of the part according to *Fig1*!

Fig1



Choose the Fillet tool in the *Features* toolbar (*Fig2*)!

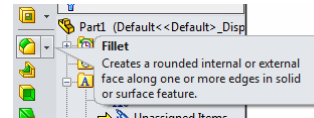
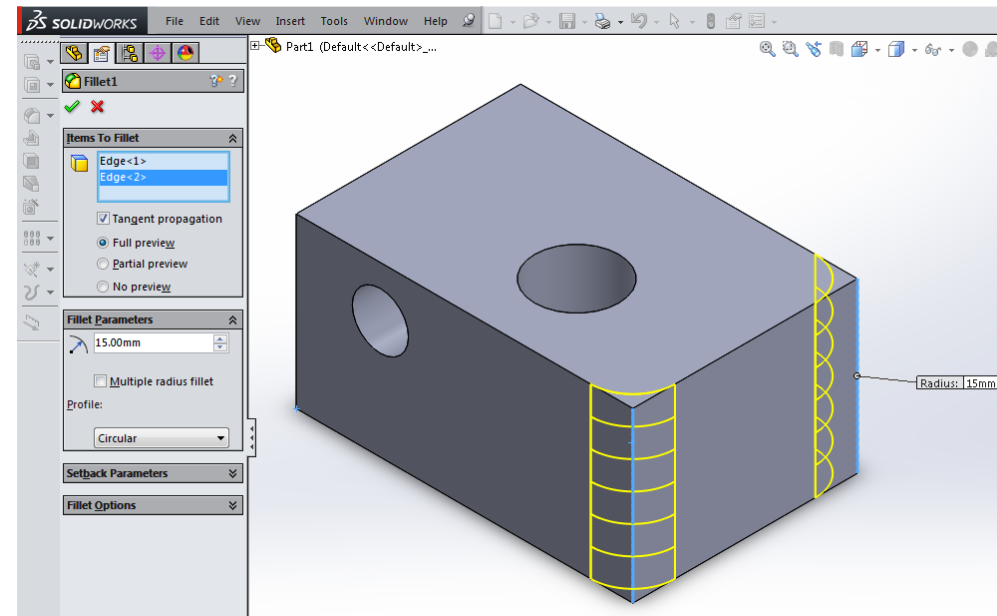


Fig2

Select the two edges to be filleted! Notice that a supplementary symbol shows what element the pointer is aiming.

- A green square marks that the pointer is over a face.
- A green *Linear* segment marks that the pointer is over an edge.
- A green dot marks that the pointer is over a vertex.

Fig3



Check that the names of both edges are listed in the Items To Fillet field then type in the value of the radius to the Radius field among Fillet Parameters (Fig3).

Accepting the settings the edges became filleted.

SHELL

Another example for sketchless *Features* is the Shell feature. It is used to open up some faces of the part and to remove material to create thin walls.

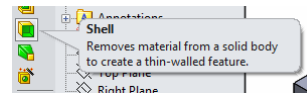
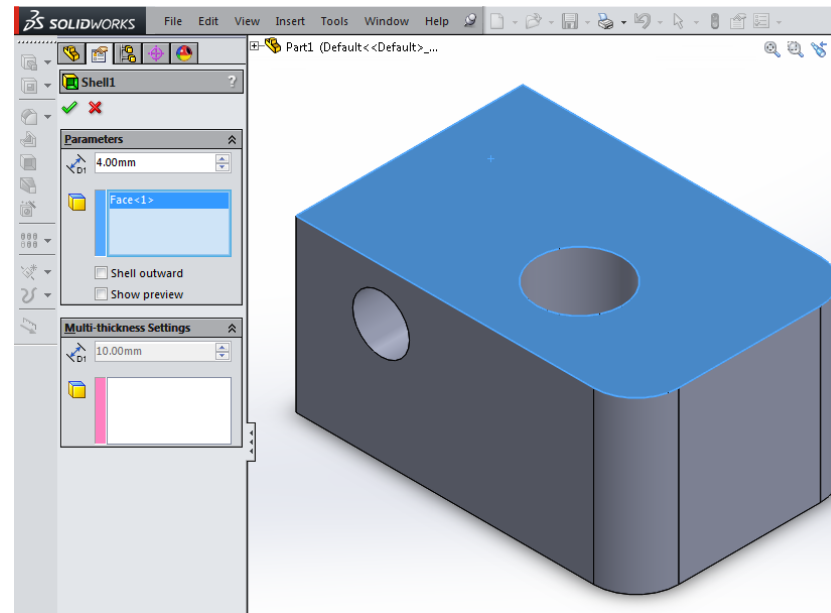


Fig4

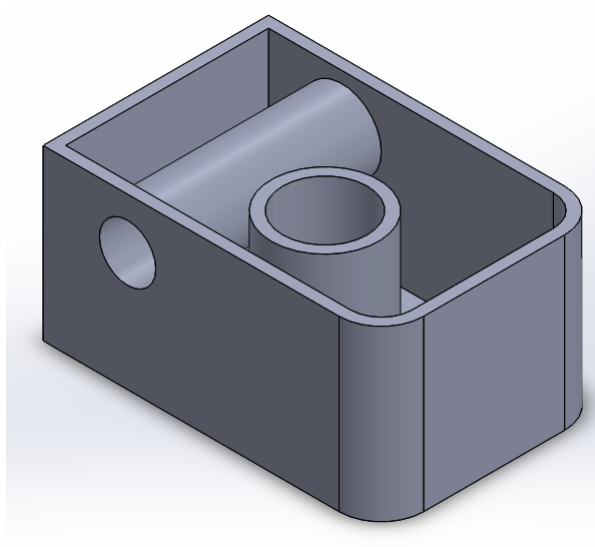
Choose the Shell tool in the *Features* toolbar (Fig4). Select the top face of the part – this face will be opened; and then type the desired wall thickness into the Thickness field among Parameters (Fig5).

Fig5



Accepting the settings creates the Shell feature (*Fig6*).

Fig6



2.6 Editing the model

It is very common situation that the design of a part should be modified. In *SolidWorks* the existing geometry can be edited easily, because every feature and sketch is available in the *Feature Manager*.

EDITING A SKETCH

Let's suppose that the dimensions of the first feature (cuboid) must be changed from 120x80x60 to 100x70x90! The first two values belong to the sketch of the feature so it is necessary to edit that sketch.

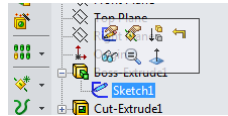


Fig1

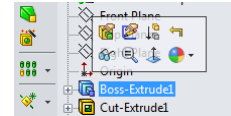


Fig2

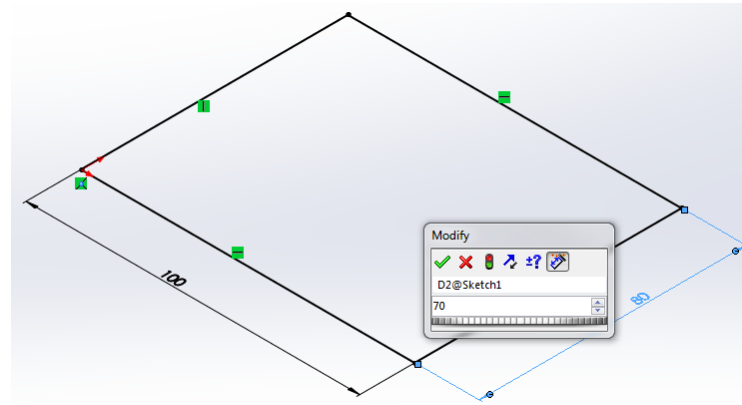
Find the feature (Boss-Extrude1) in the *Feature Manager*!

The modification of its sketch can be initiated two ways:

- Expand the sublist of the feature to see its sketch! Click on the sketch item and choose the first tool (Edit Sketch) in the pop-up menu (Fig1).
- Click on the feature and choose the second tool (Edit Sketch) in the pop-up menu (Fig2).

The sketch opens and the modification of the dimensions is possible by double-clicking on the dimension numbers (*Fig3*).

Fig3



After changing the dimensions use the Confirmation Corner to leave the sketch accepting the changes (*Fig4*).

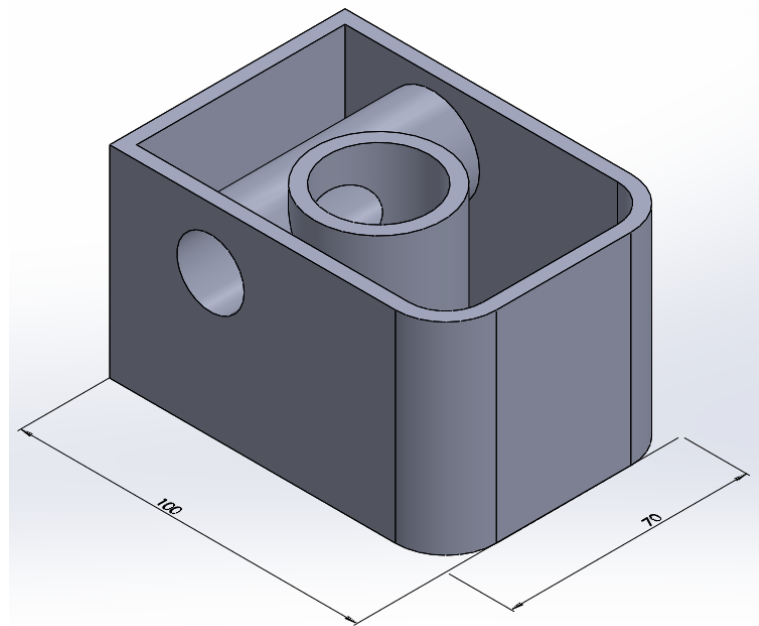


Fig4

SolidWorks rebuilds the model to show the effect of the changes (*Fig5*).

Of course not only the modification of the dimensions but also adding or removing sketch elements is possible.

Fig5



EDITING A FEATURE

The height of the part was defined during the creation of the first feature (Boss- Extrude1) so the definition of this feature is necessary to modify. Go to the *Feature Manager*, locate the feature and click on it! Select the first tool (Edit Feature) from the pop-up menu (Fig6)!

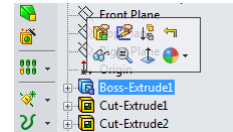
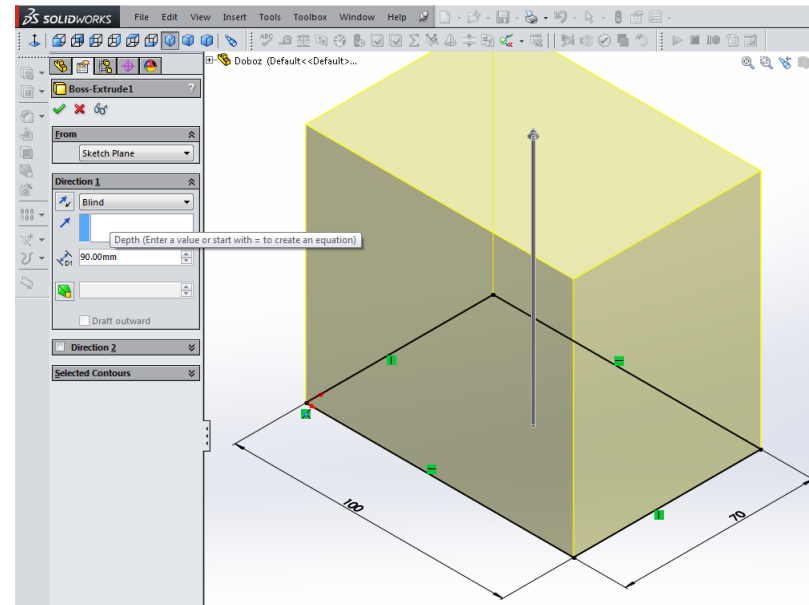


Fig6

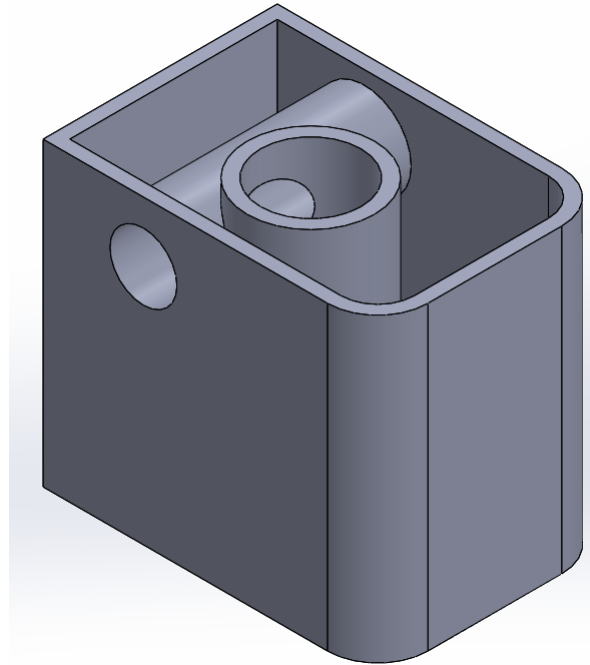
As you can realize we returned to the definition stage of the extrusion. Modify the value of the Depth field to change the height of the cuboid (Fig7).

Fig7



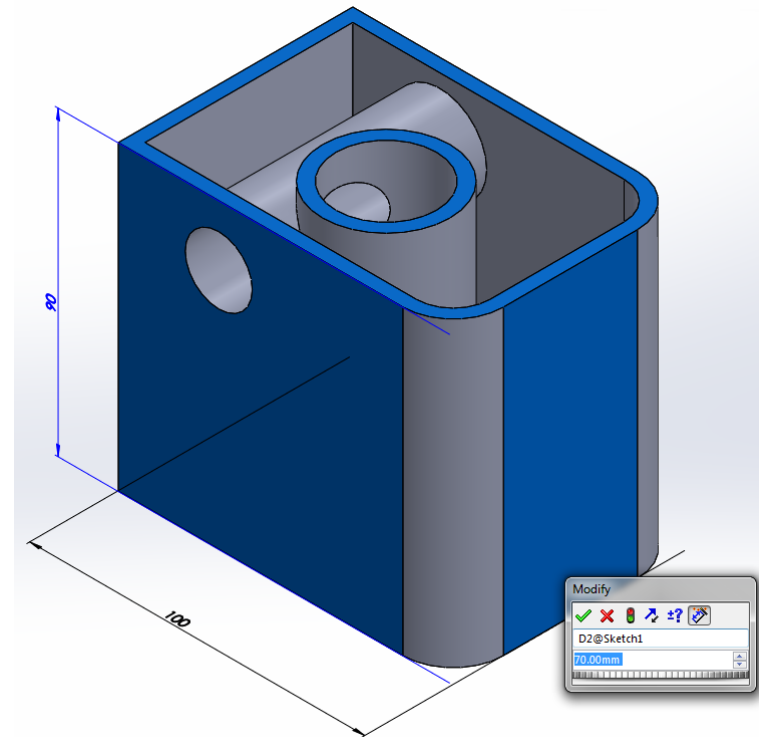
Accepting the settings the modified part appears on the *Drawing Area* (Fig8).

Fig8



Frequently happens that only the dimension values should be modified. There is a quick way to perform these modifications. Double-click on the outer surface of the part! This surface belongs to Boss-Extrude1 so *SolidWorks* reveals all of the dimensions what belong to this feature.

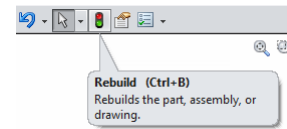
Fig9



Double-clicking on the dimension numbers makes the modification of their values possible (Fig9).

Modify the dimensions of the cuboid to restore their original values!

As you change a dimension the modification does not take effect at once.

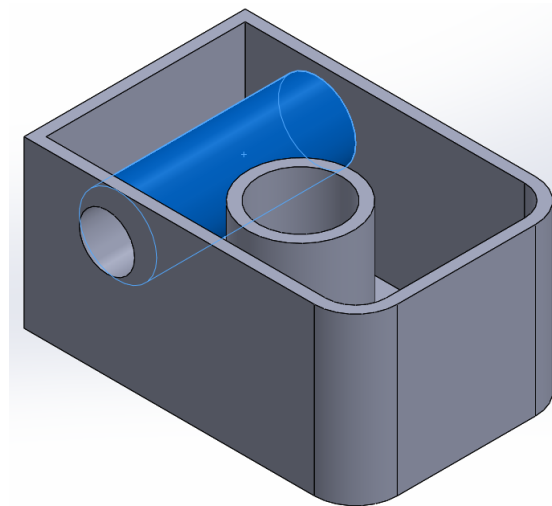
*Fig10*

To see the changes of the geometry let *SolidWorks* to rebuild the model using the Rebuild tool in the Standard toolbar (*Fig10*).

REORDERING FEATURES

Beyond modifying dimensions more sweeping changes can be performed on the model using the *Feature Manager*.

Let's suppose that we want to remove the horizontal tubular shape from the part (*Fig11*).

Fig11

This detail was created during shelling so selecting the cylindrical face and pressing Delete on the keyboard would mean the erasure of the complete *Shell1* feature. This is not what we want – the shell and also the second hole is necessary.

It is very useful if we overview the creation of the part. For that find the thick blue *Line (Rollback Bar)* at the bottom of the feature list. Click on the *Rollback Bar* (the pointer turns to a hand) and drag it to the start of the list – the design shows its earliest state (*Fig12*). Dragging the *Rollback Bar* downward the creation of the *Features* can be overviewed step by step.

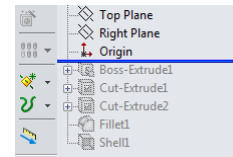
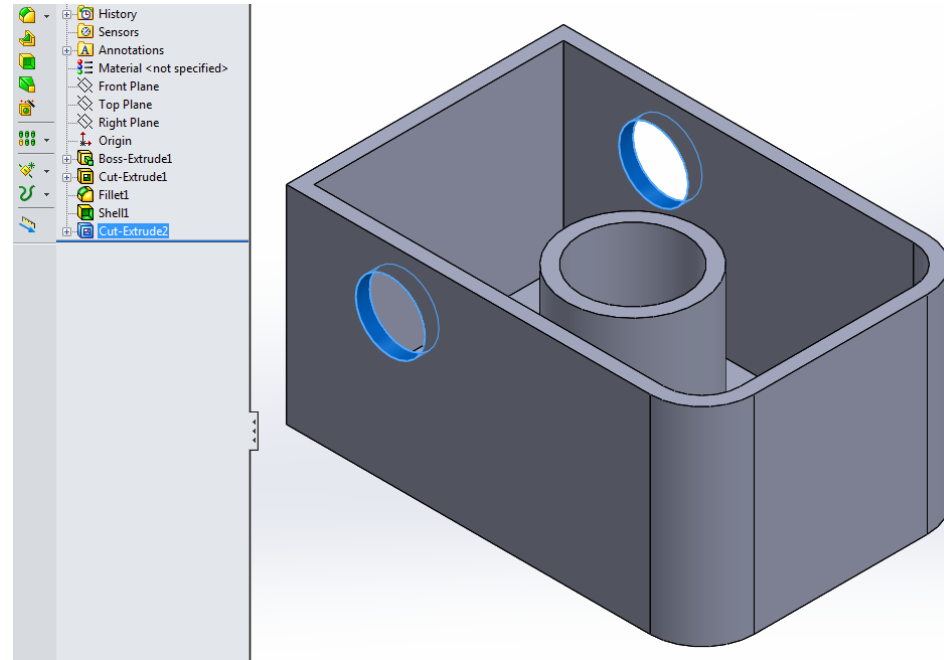


Fig12

Replaying the modeling process we can realize that the order of the *Features* is wrong. The second hole (*Cut-Extrude2*) should be created after the *Shell1* feature. Fortunately the order of the *Features* can be modified – click on *Cut-Extrude2* in the *Feature Manager* and drag it to the bottom of the list!

Now the second hole comes after shelling the part – the *Drawing Area* shows the modified design (*Fig13*).

Fig13



Because the second hole was defined as a through-hole it cuts the back wall of the part. Change the extent of the hole to cut only one wall!

This setting belongs to the definition of the feature. Click on *Cut-Extrude2* in the *Feature Manager* and choose Edit Feature in the pop-up menu.

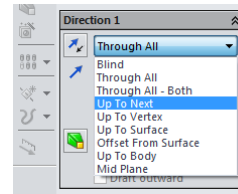
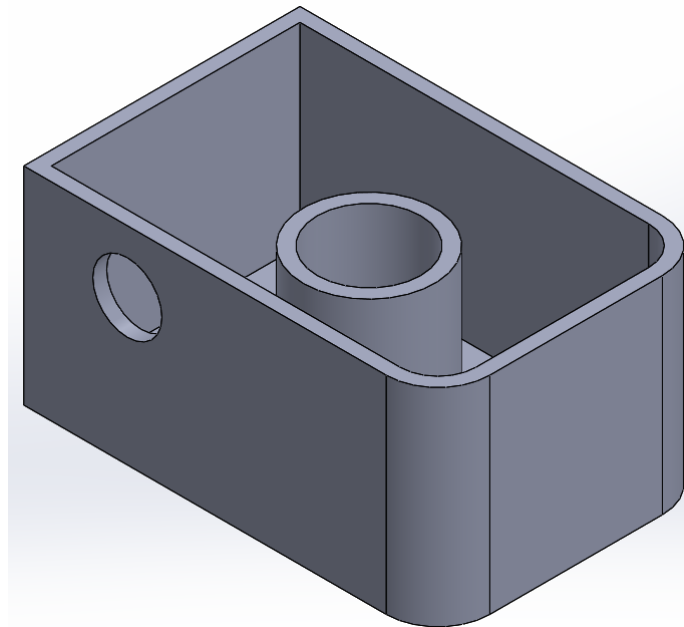


Fig14

Drop down the End Condition list in the *Direction 1* group and change *Through All* to *Up To Next* (Fig14).

Accept the changes and examine the result (Fig15).

Fig15



Now as *Cut-Extrude2* is performed the material removal stops after breaking through the material at the first time.

Reposition the hole to be on the opposite wall! As you can remember the very first momentum of creating *Cut-Extrude2* was the selection of the sketch plane. This decision can be changed. For this go to the *Feature Manager* and find the sketch of *Cut-Extrude2*! Click on it and select *Edit Sketch Plane* from the pop-up menu (Fig16).

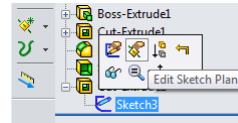
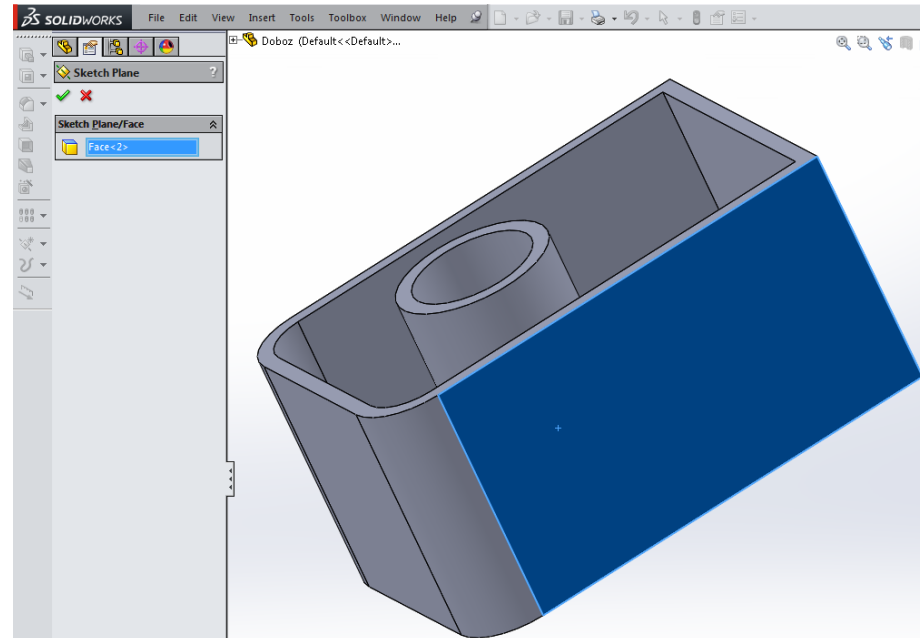


Fig16

The Sketch Plane dialog shows which face of the part was selected as the sketch plane. Rotate the part using the mouse wheel and select the back face (Fig17).

Fig17



Unfortunately the rebuild of the model is unsuccessful. An error message informs us that it was impossible to find the end of the material removal as *Cut-Extrude2* was created (Fig18).

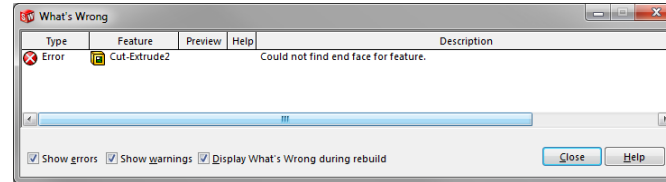
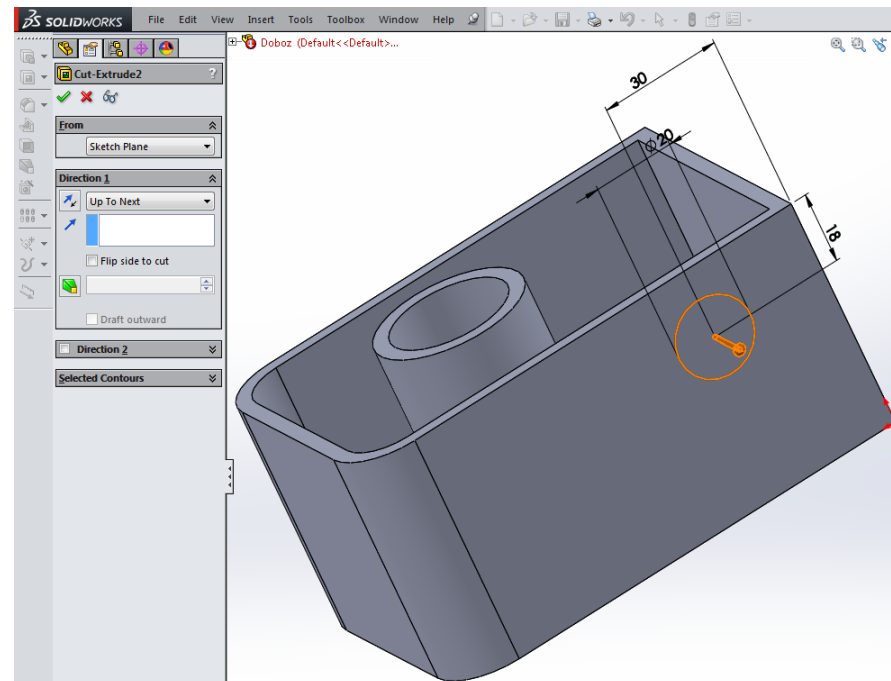


Fig18

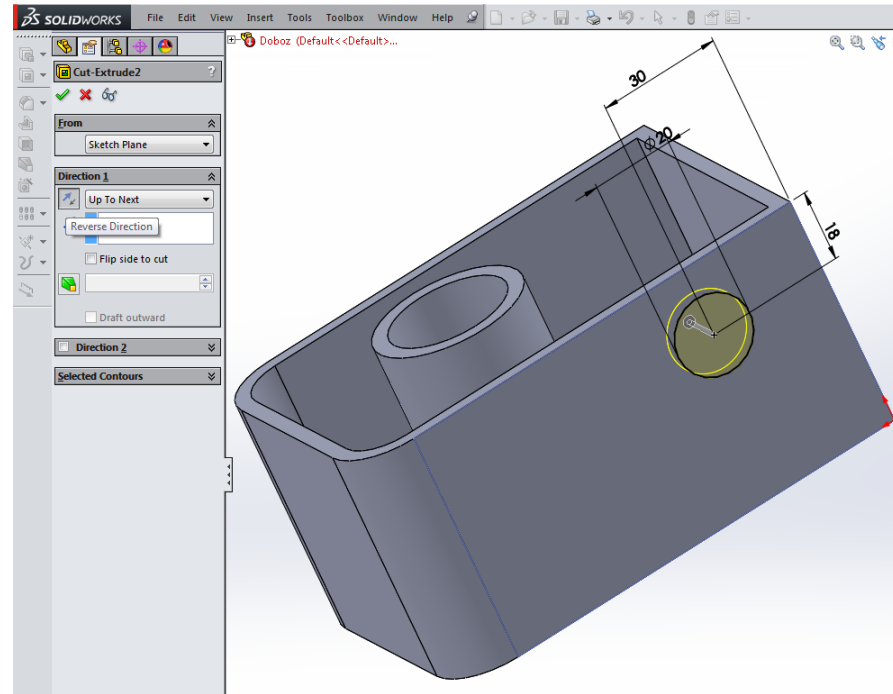
The cause of the error is that the direction of the material removal remained unchanged. Edit *Cut-Extrude2*! The arrow on the *Drawing Area* shows the wrong direction of the cut (Fig19).

Fig19



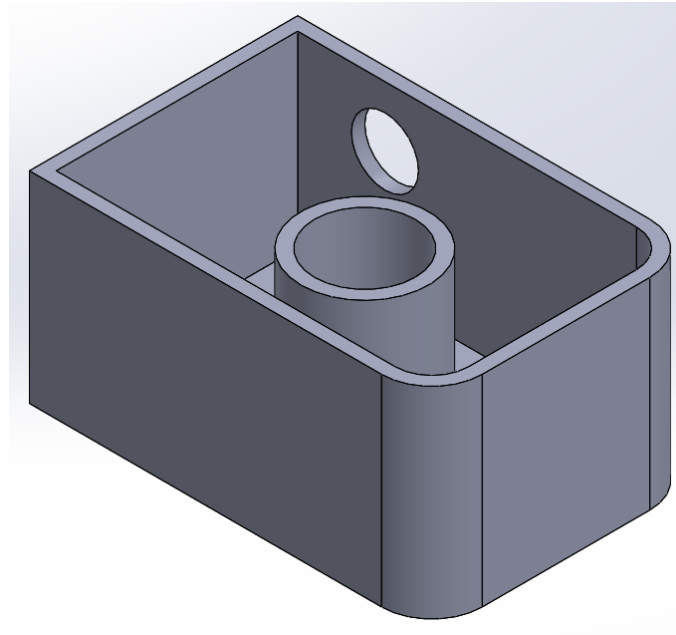
To change the direction click on the Reverse Direction button. As the graphical preview shows the cutting can be performed now (Fig20).

Fig20



Accept the settings and set up an isometric view of the part (Fig21). The modeling process is completed.

Fig21



The part document can be downloaded from the following link: [Box.SLDPRT](#)
This practice illustrated the general workflow of part modeling in *SolidWorks*:

- Simple sketches were created on appropriate planes.
- The sketches were made fully defined using dimensions and relations.
- The sketches were turned to features.
- The sketches and the *Features* were edited to perform changes on the model.

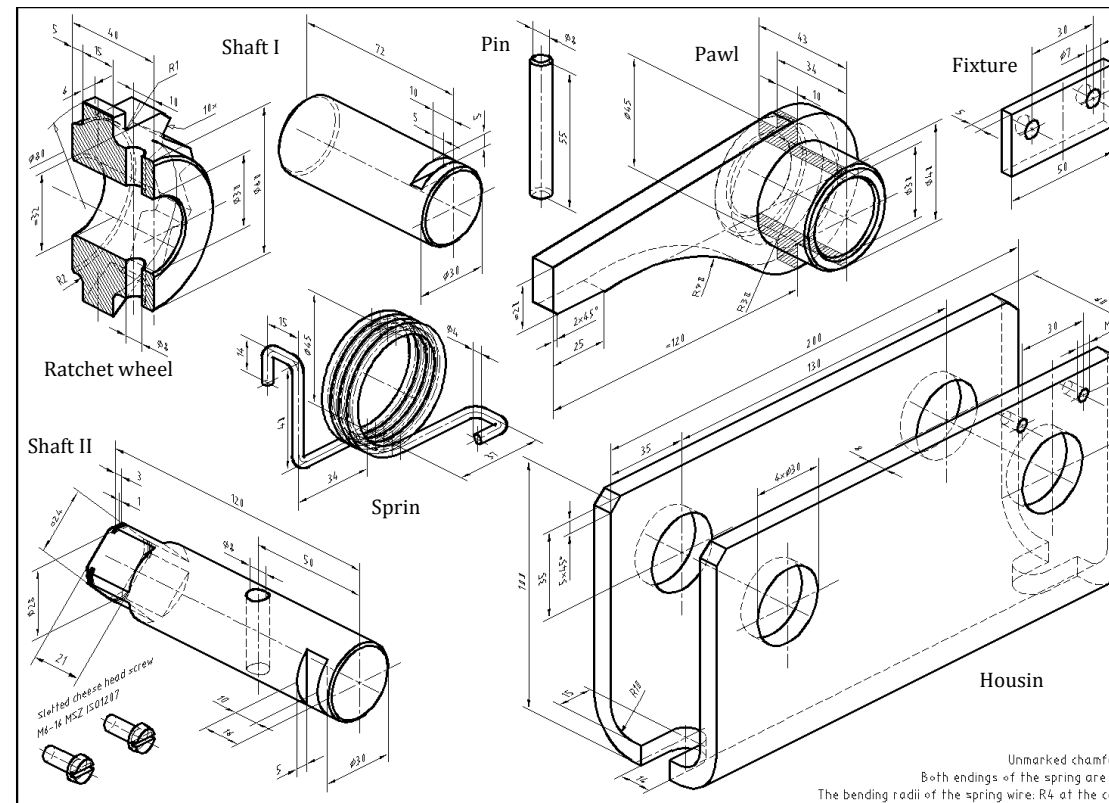
3. fejezet

3.1 Modeling a ratchet mechanism

In the next series of practices the *Components* of an *Assembly* (a ratchet mechanism) will be modeled. The axonometric views and the names of the *Components* can be found on *Fig1*.

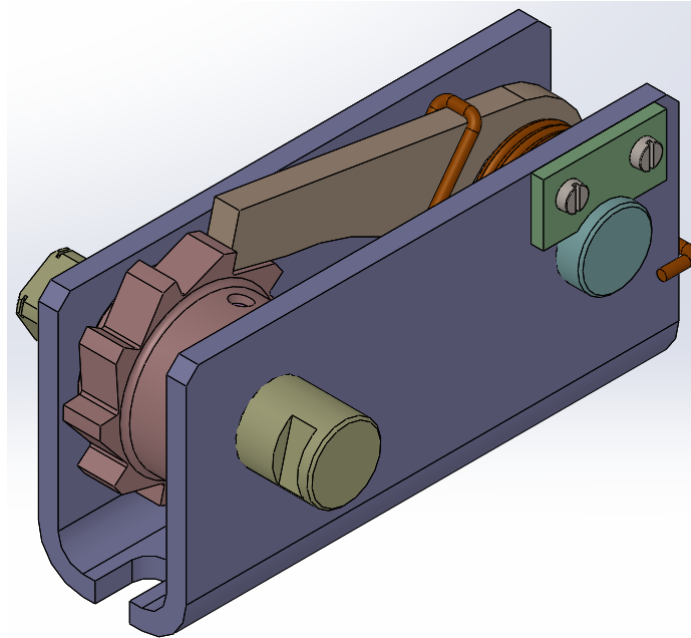
The drawing of *Fig1* can be downloaded from the following link: [Ratchet mechanism - parts.pdf](#)

Fig1



To better understand the functions of the parts the assembled mechanism can be seen on Fig2.

Fig2



During modeling the parts of the mechanism we always aim to create good quality models. What are the *Properties* of a good part model in *SolidWorks*?

- A good model is simple. It contains as few *Features* as possible.
- Even more important that also the sketches of a good model are as simple as possible. Simple sketches are easy to modify. Use more *Features* if it necessary to create the simplest possible sketches. This way the model will be flexible – the modifications can be performed easily.

- Of course the sketches must be fully defined. *SolidWorks* enables to use under defined sketches, but get accustomed to always fully define them.
- The order of the *Features* in a good model is logical. It is a good idea to follow the manufacturing order of features. Another method can be to create the most important *Features* first and then the small details like chamfers and fillets).
- The good model reflects the designer's intent. The dimensions are defined to follow the function of the part. The dimension networks of the parts on Fig1 also created this way so during the modelling processes of the parts always rigorously the same dimensions will be built into the models.
- A good model survives modification. For example through-holes must be defined to go *Through All* of the material of the part even if the thickness would be changed later.
- To prevent getting lost in the three dimensional space it is advisable to keep the *Orientation* of the model as the projections of the parts on Fig1. Always try to orientate the part keeping its natural position.

3.2 Modeling the parts of the mechanism

The following subchapters review the modeling of the *Components* of the ratchet mechanism starting with the simplest one moving towards the more complicated ones. Every component will be created as a separate part document. Don't forget to save the parts because they will be used later to build the *Assembly* of the mechanism.

PIN

The simplest part in the collection is the *Pin*. It is basically a cylinder. Create a sketch on the *Top Plane* and sketch a circle onto it! The center point of the circle is at the origin – the actual value of the radius is not important. Dimension the diameter of the circle (Fig1)! The sketch is fully defined.

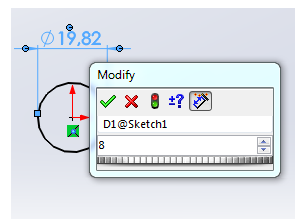
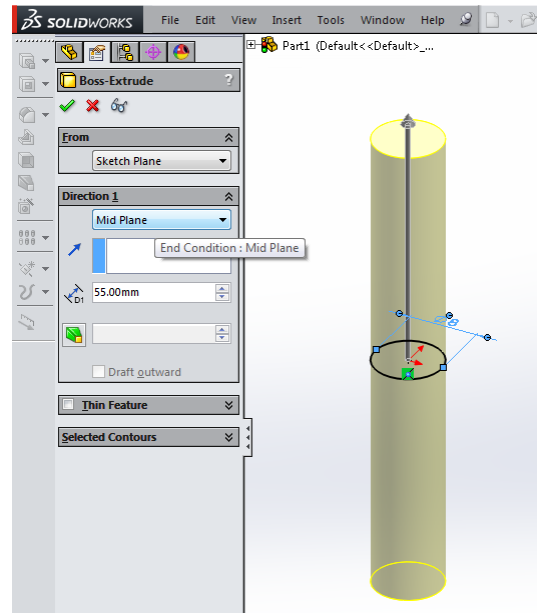


Fig1

Create a cylinder from the circle by extrusion (*Extruded Boss/Base*)! The height of the cylinder is 55 mm.

Notice that the part is symmetrical. It is a good idea to keep the main planes as symmetry planes if it is possible. To do that use the *Mid Plane* option for the extent of the extrusion instead of *Blind* (Fig2). This option means that the extrusion will be performed symmetrically to the sketch plane altogether 55 mm long – the *Top Plane* remains a symmetry plane of the part.

Fig2



To make the insertion easier both ends of the part are chamfered.

The size of the chamfer is defined in the lower *Right* corner of the drawing on **Error! Reference source not found..**

Select the Chamfer tool in the *Features* toolbar (Fig3)!

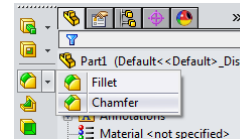


Fig3

The definition of a *Chamfer* feature is very similar to the *Fillet*'s. Type the size of the chamfer into the *Distance* field of the dialog and select the edges to be chamfered (Fig4)! Instead of the edges also the cylindrical face of the part can be selected – it would mean that every edge of the face has to be chamfered.

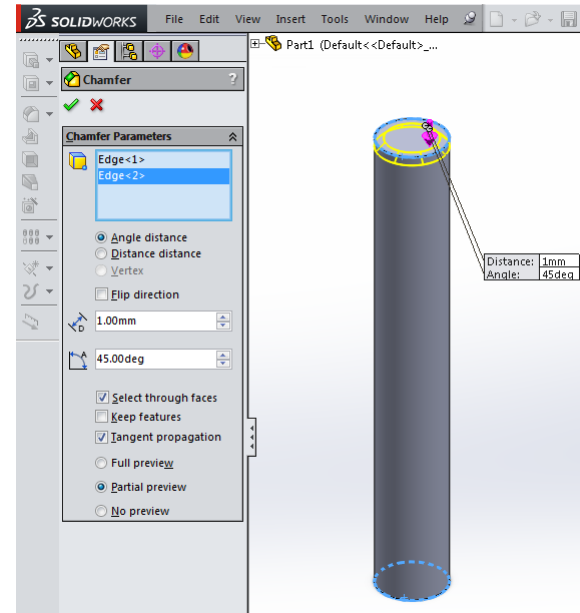


Fig4

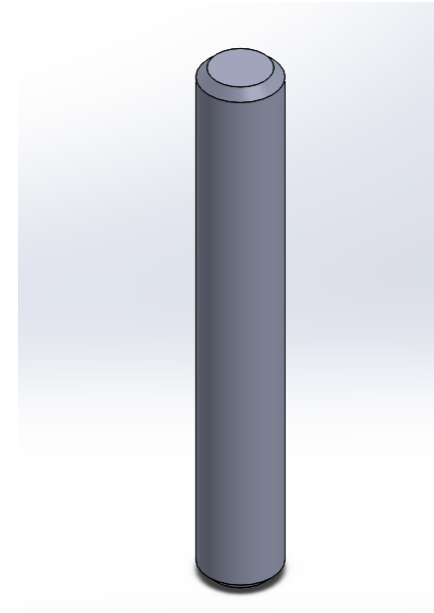


Fig5

After accepting the settings the camfers are created and the model of the *Pin* is completed (Fig5).

The part document can be downloaded from the following link: [Pin.SLDPRT](#)

FIXTURE

The *Fixture* is a cuboid. Sketch a rectangle onto the ! using the *Corner Rectangle* tool! Dimension the width and the height of the rectangle (Fig6)!

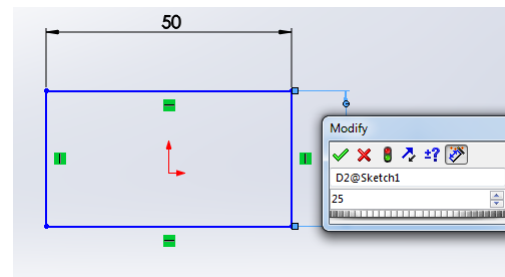


Fig6

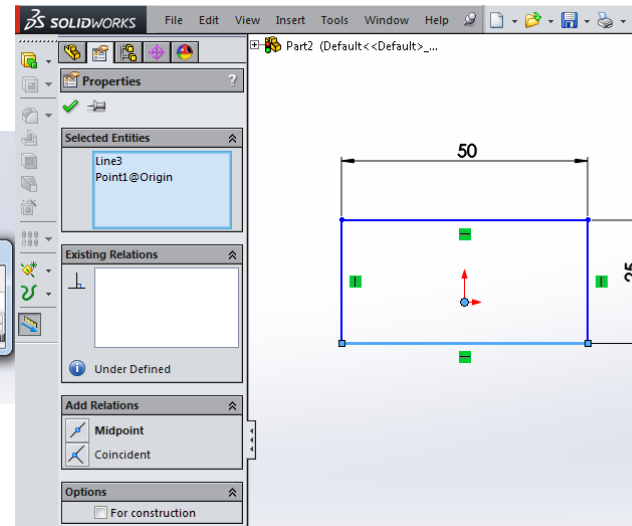


Fig7

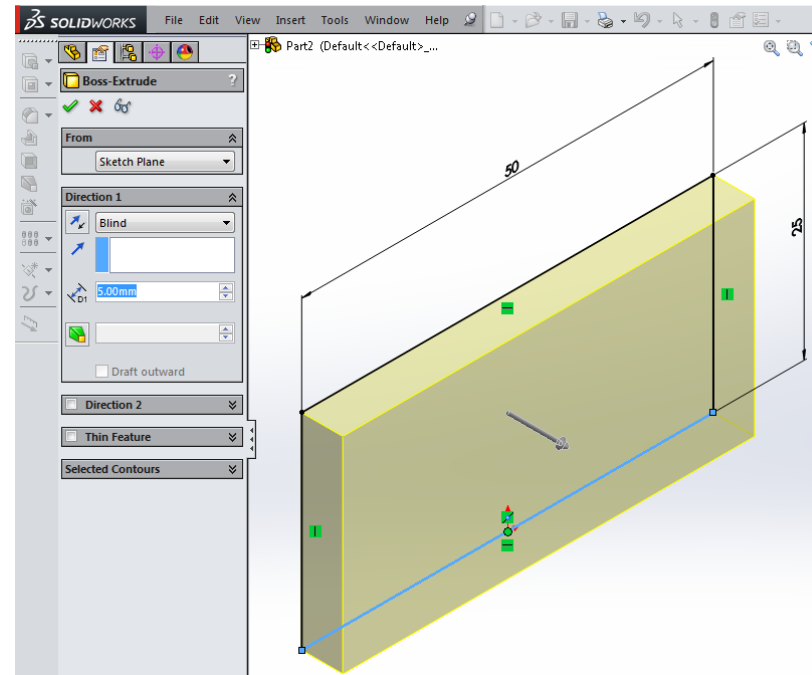
The position of the rectangle relative to the origin of the sketch shall be defined so as to keep the symmetry of the part. Select the origin and pressing *Ctrl* on the keyboard select the lower horizontal edge of the rectangle.

To the left of the *Drawing Area* the list of the selected entities appears (A *Line* and a point at the origin is selected) (Fig7). There are no existing relations for the selected entities.

In the third group on the dialog the definable relations are listed. Choose the *Midpoint* relation!

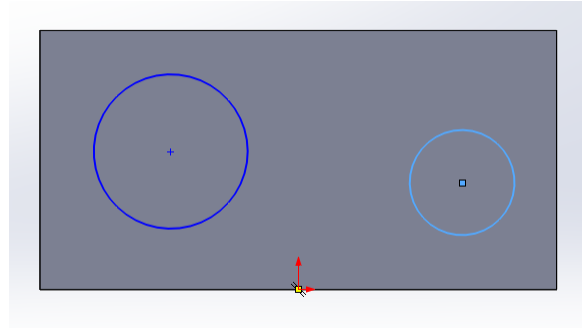
Extrude the rectangle to 5 mm thickness (Fig8)!

Fig8



The next feature is the group of the holes. Select the large rectangular face of the model and sketch to circles onto it (Fig9)!

Fig9



Notice that a symmetry *Line* marks the symmetrical placement of the holes on **Error! Reference source not found.** This symmetry is an information which must be built into the model. First it is necessary to sketch the symmetry axis. This axis must be drawn with a special *Linetype*. Select the *CenterLine* tool in the *Sketch* toolbar (Fig10)! *CenterLines* are widely used in sketches. They can be interpreted as symmetry axes, as rotational axes or even as construction *Lines*.

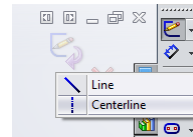
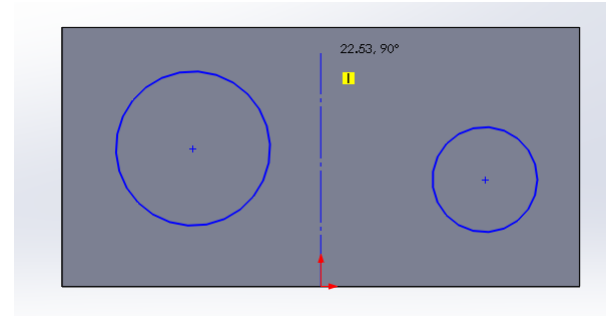


Fig10

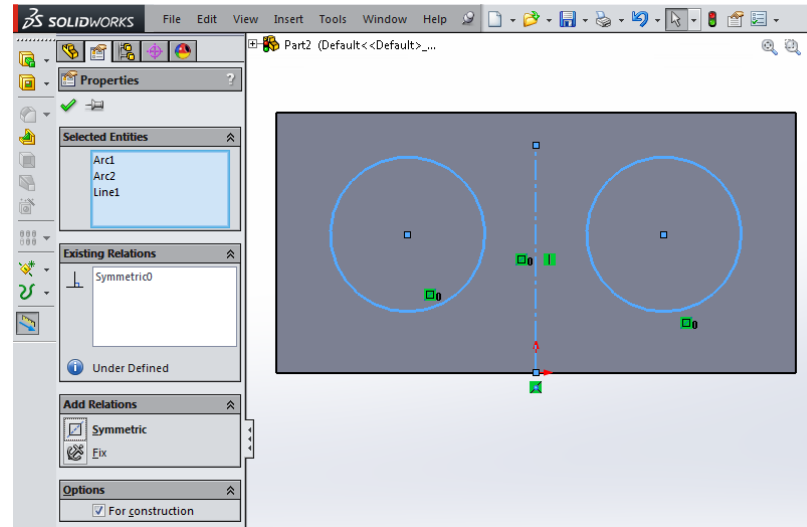
The starting point of the symmetry axis is at the origin of the sketch or at the *Midpoint* of the lower horizontal edge of the part (which has been found first). The direction of it is vertical. Try to sketch the symmetry *Line* vertically! If you are close enough to the *upRight* position the *CenterLine* “jumps” to the vertical direction and a yellow symbol marks that a Vertical relation will be created after placing the endpoint of the segment (Fig11). Place the endpoint – the actual position of it is not important, only the verticality. Press Esc on the keyboard to stop sketching more *CenterLines*.

Fig11



Now the sketch contains a *Line* which can be used as a symmetry axis. Click on it and pressing *Ctrl* on the keyboard select the two circles! Choose the *Symmetric* relation (Fig12)!

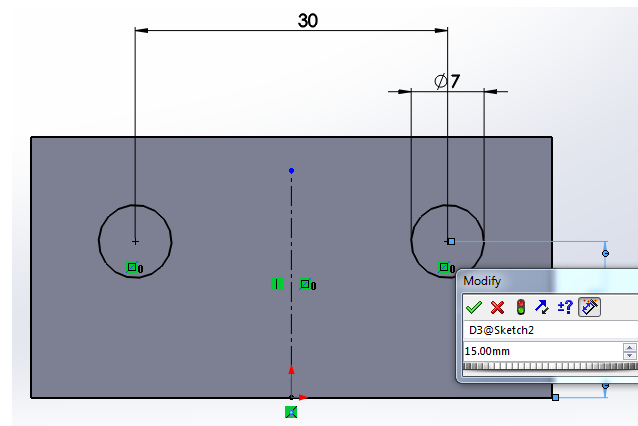
Fig12



Perform the dimensioning of the sketch! Define

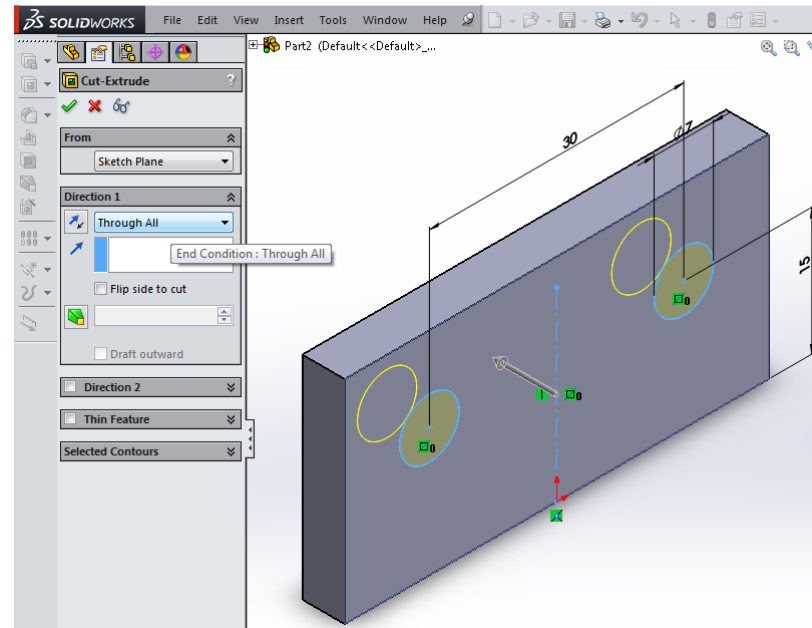
- the diameter of one of the circles,
- the distance between the circles and
- the distance between one of the circles and the lower horizontal edge of the part (*Fig13*).

Fig13



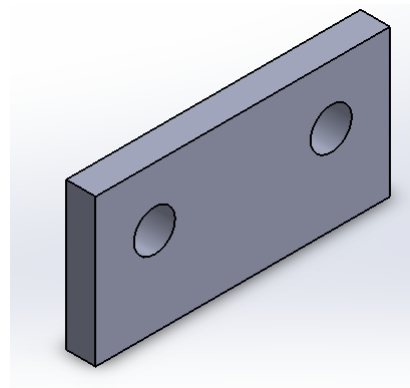
Create the through-holes using the Cut Extrude tool in the *Features* toolbar (*Fig14*)!

Fig14



The *Fixture* is done (Fig15).

Fig15



The part document can be downloaded from the following link: [Fixture.SLDPRT](#)

SHAFT I.

The fundamental shape of this part is a cylinder. Create the cylinder! Sketch a circle onto the ! – the center of the circle is at the origin. Dimension the diameter of the circle (*Fig16*)! Extrude the circle to get a cylinder (*Fig17*)!

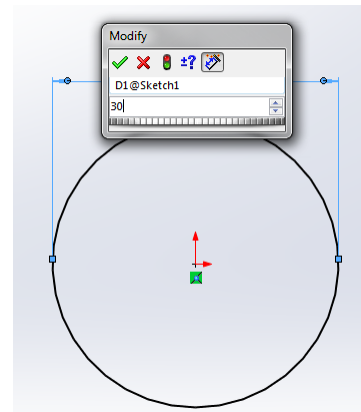


Fig16

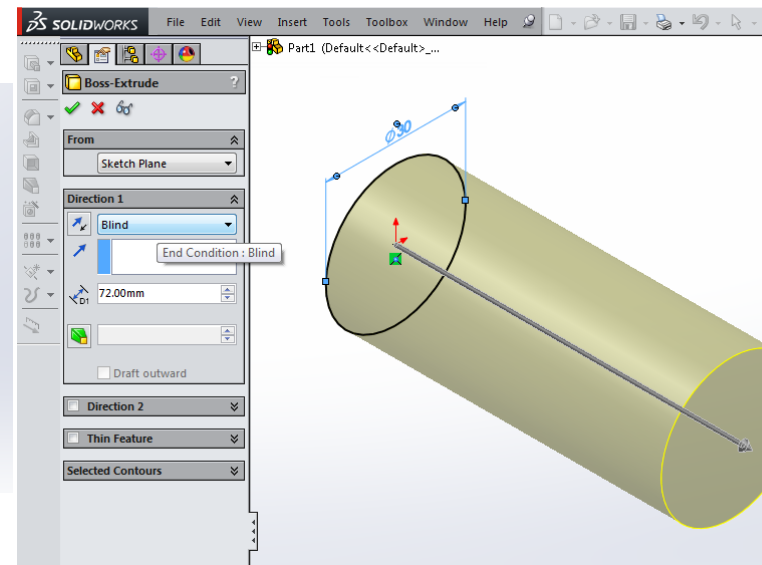


Fig17

The next feature of the part is the slot, because it is more important than the chamfers.

The slot is a material removal by extrusion (*Extruded Cut*) which requires a sketch.

There are three possibilities for the placement and the content of the sketch:

- The sketch can be placed onto a plane which is parallel to the end face of the cylinder. In this case the content of the sketch would be a circular segment (*Fig18*);
- The sketch can be placed onto a horizontal plane which is tangential to the cylindrical surface. In this case the content of the sketch would be a rectangle (*Fig19*);
- The sketch can be placed onto a vertical symmetry plane of the cylinder. In this case the content of the sketch would be a square (*Fig20*).

Try to find out which solution is the best!

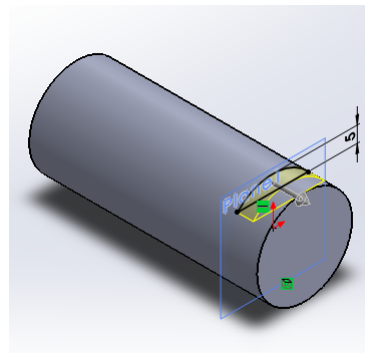


Fig18

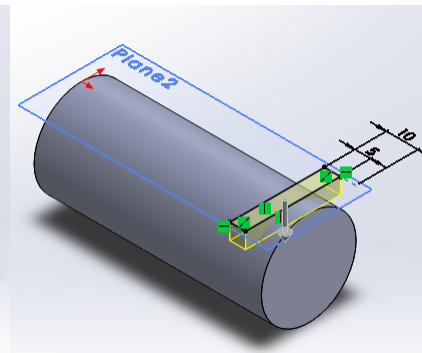


Fig19

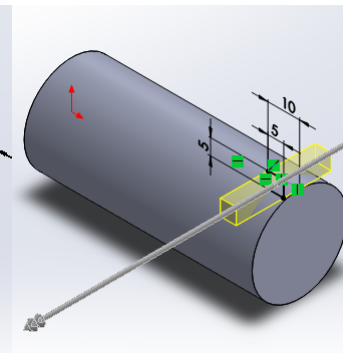


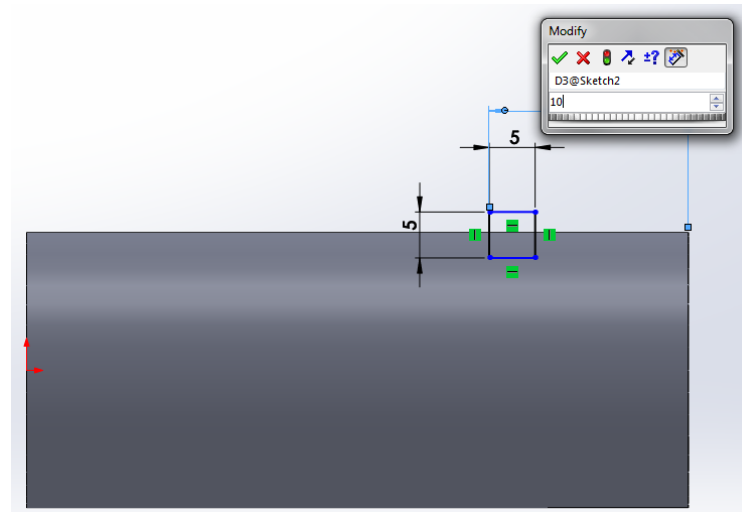
Fig20

The third solution is better than the others because it is simpler. It does not require defining a new plane – the *Front Plane* can be used as a sketch plane.

Select the *Front Plane* and start sketching. It is advisable to apply the *Normal To* tool in the View (*Heads-Up*) toolbar.

Sketch a rectangle using the *Corner Rectangle* tool in the Sketch toolbar. Dimension the width and the height of the rectangle! Dimension the distance between the left vertical edge of the rectangle and the *Right* edge of the cylinder (*Fig21*)!

Fig21



The vertical position of the rectangle (which is a square now) is defined by a relation. Press *Esc* on the keyboard to stop dimensioning! Select the top horizontal edge of the square and pressing *Ctrl* on the keyboard select the top contour (silhouette edge) of the cylinder (*Fig22*)! Relate the two selected elements to be *Collinear* (*Fig23*)! From now on the slot will keep its correct position even if the diameter of the cylinder was changed.

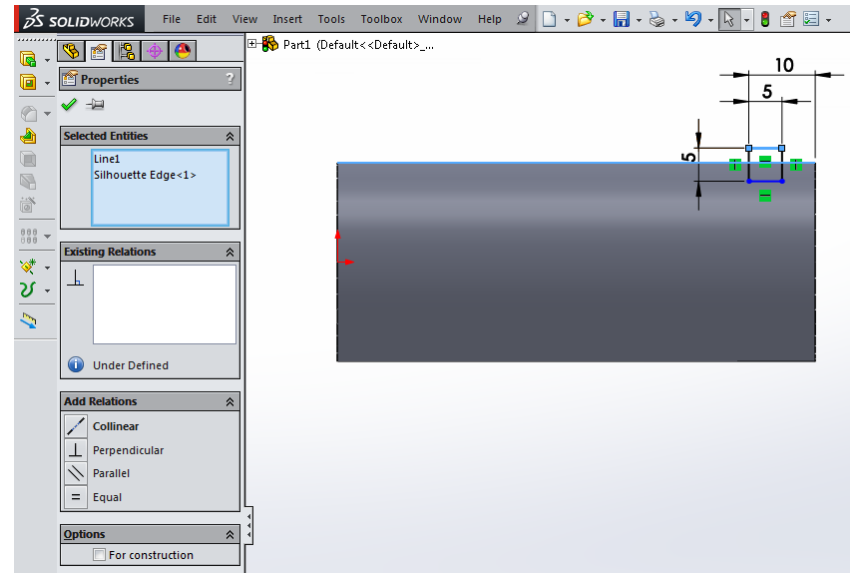


Fig22

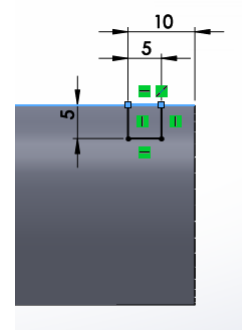
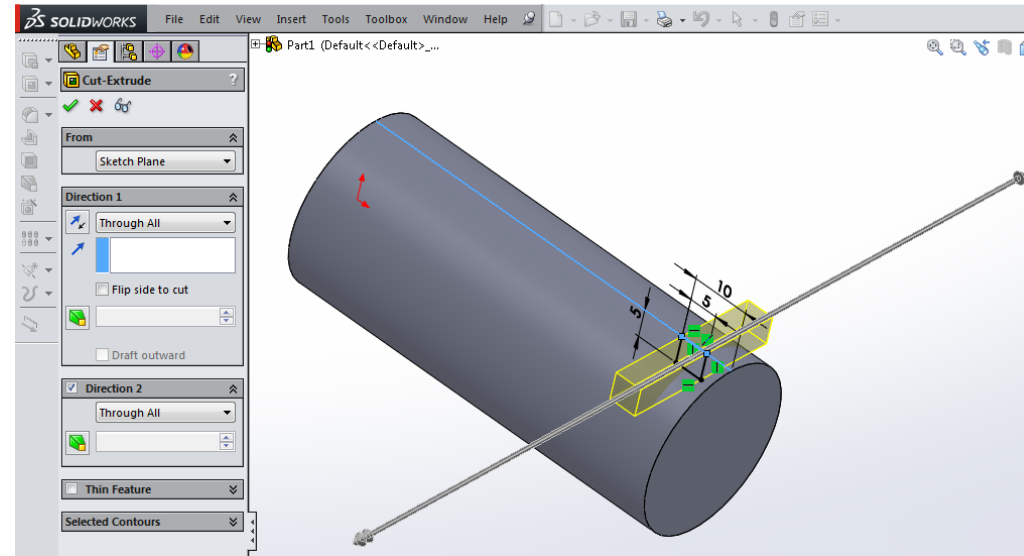


Fig23

Choose a general viewing direction and use the Cut Extrude tool in the *Features* toolbar to create the slot!

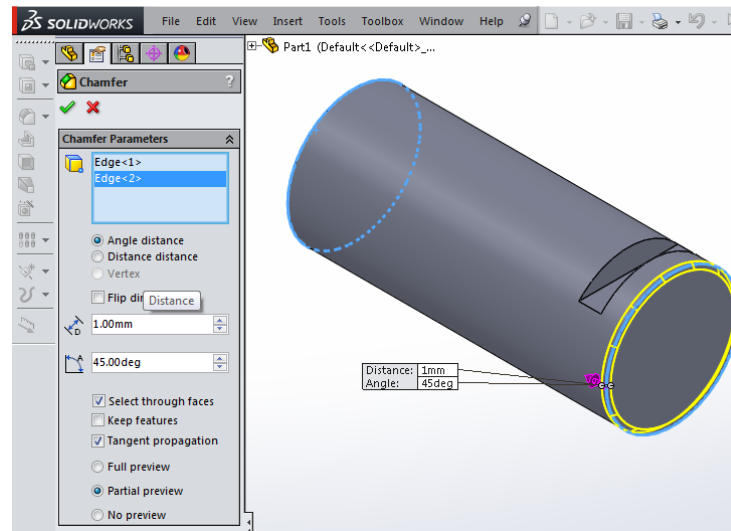
Fig24



Set the extent of the cut to *Through All*! It is necessary to remove material also on the opposite side of the sketch plane so check Direction 2 and set the *End Condition to Through All* again (Fig24). The two directions can be identified according to the arrowheads on the graphical preview.

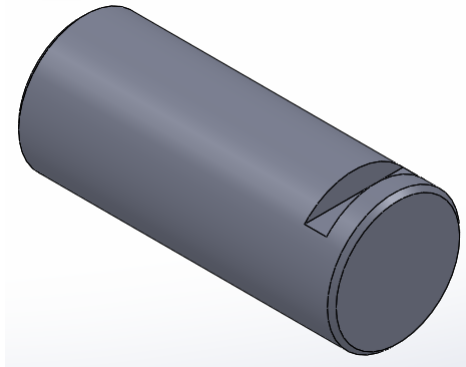
Finally chamfer the edges of the end faces of the cylinder (Fig25)!

Fig25



The part is completed (Fig26).

Fig26

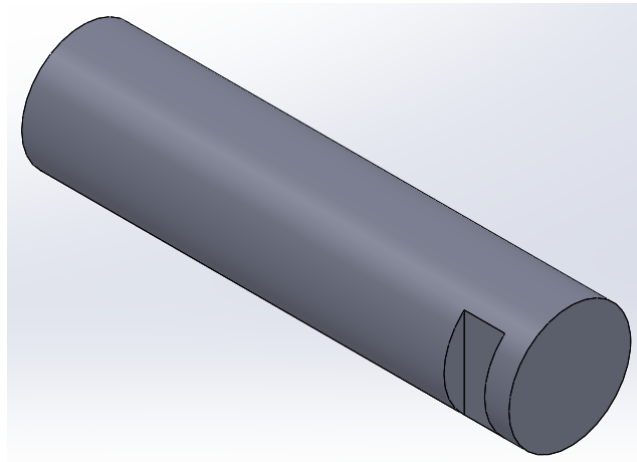


The part document can be downloaded from the following link: [Shaft I.SLDPRT](#)

SHAFT II.

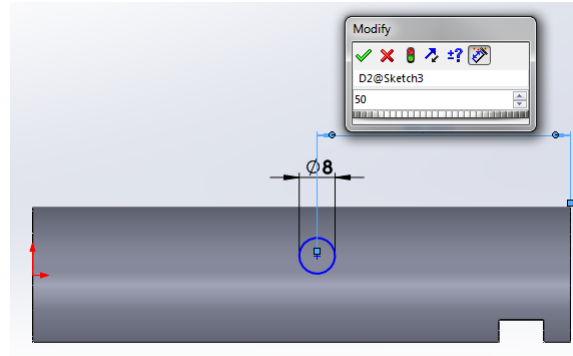
Some details of *Shaft II.* are rather similar to *Shaft I.* Create the base feature (cylinder) and the slot independently! The only difference beyond the dimension values is that the sketch of the slot should be placed onto the *Top Plane* (Fig27).

Fig27



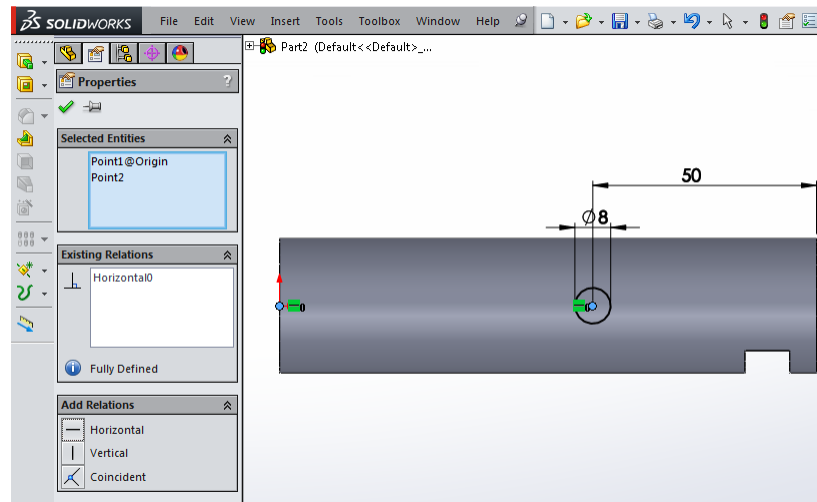
The part contains a vertical hole. Sketch a circle onto the *Top Plane* and perform the dimensioning (Fig28)!

Fig28



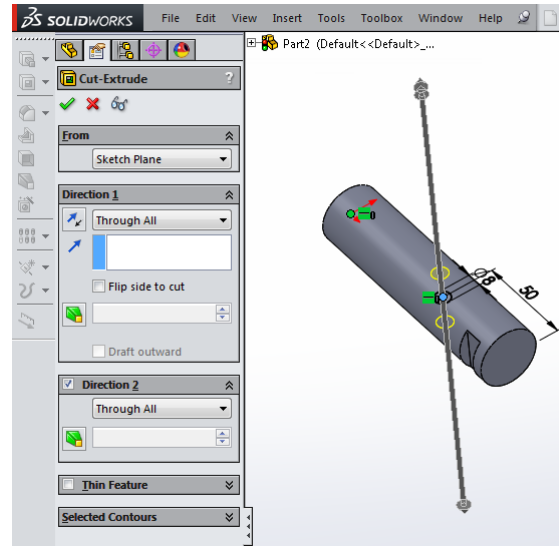
The vertical position of the circle is defined by a relation. Select the origin of the sketch and the center of the circle then choose the Horizontal relation (Fig29)!

Fig29



Create a Cut-Extrude feature – remove material *Through All* in both directions (Fig30)!

Fig30



The drawing of the details on the far end of the part (**Error! Reference source not found.**) is a bit crowded but these details can be realized examining the dimensions carefully.

One of these elements is a circular groove for a snap ring. This groove is rather similar to a slot but the material removal is performed not by extrusion but by rotation.

Select the *Top Plane* and start sketching! Set the viewing direction *Normal To* the sketch plane and zoom onto the left end of the part! Sketch a *Corner Rectangle* (Fig31)!

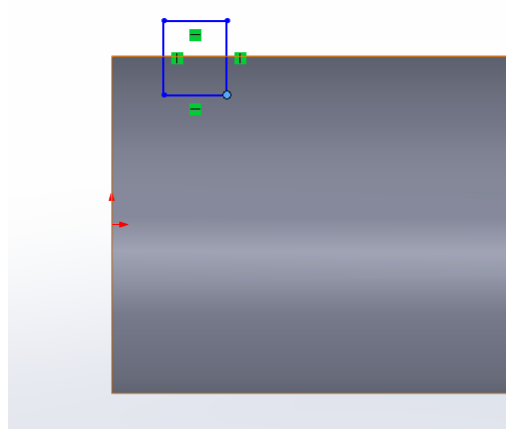


Fig31

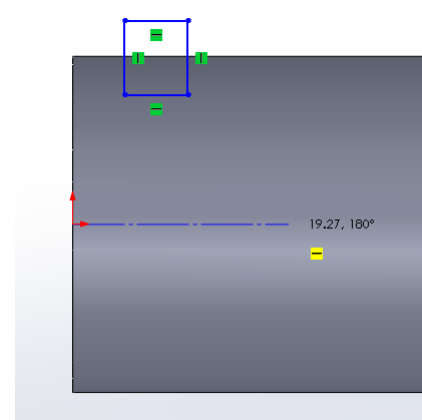
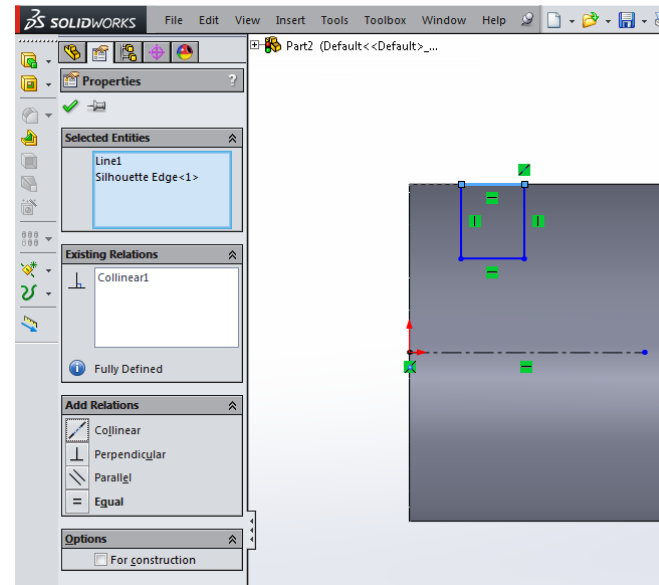


Fig32

Because the sketch will be used for a revolved feature an axis of revolution is to be sketched. Sketch a horizontal *CenterLine* starting at the origin of the sketch (Fig32)! The actual length of the *CenterLine* is not important.

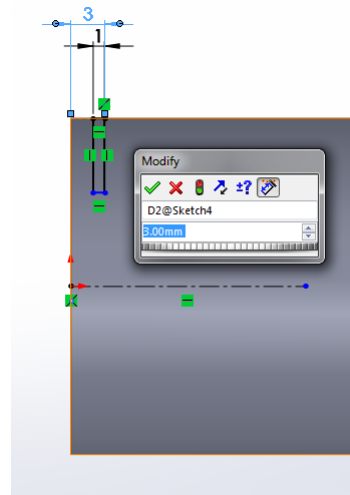
Set the top horizontal edge of the rectangle to be *ColLinear* with the contour of the cylinder (Fig33)!

Fig33



Perform the dimensioning of the horizontal dimensions (*Fig34*)!

Fig34



The only remaining dimension of the sketch is the inner diameter of the groove. It seems to be problematic, because the groove is not created yet so seemingly there is no reference to which the dimensioning can be performed.

There are typical WRONG solutions:

- Dimensioning the height of the rectangle which is the radial difference of the groove and the cylinder.
- Dimensioning the radius of the groove which is the half of the diameter.

Do not follow these methods! Usually if you perform mental arithmetic you can be sure that you are on a wrong way. The important information is the diameter so this is what must be built into the model.

Fortunately *SolidWorks* is prepared to solve the problem. Start to dimension the distance between the *CenterLine* and the lower horizontal edge of the rectangle but don't place the dimension number (*Fig35*)! Drag the dimension number over the *CenterLine*! It can be observed that the dimension toggles to the doubled value (*Fig36*). This is exactly what we need so place the dimension number and type in the necessary value (*Fig37*).

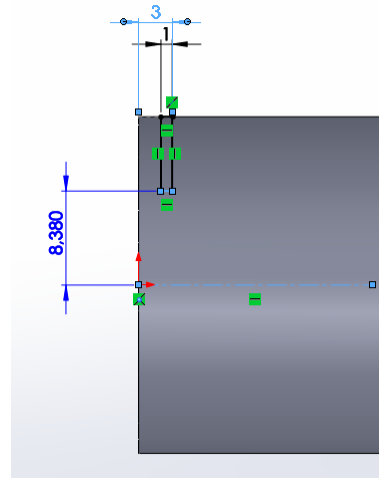


Fig35

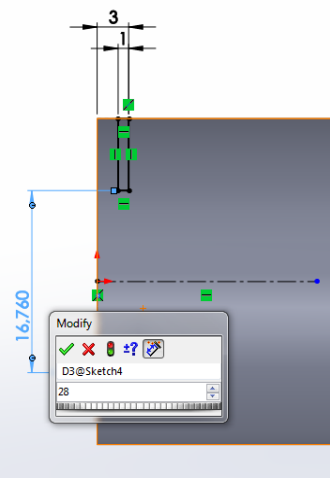


Fig36

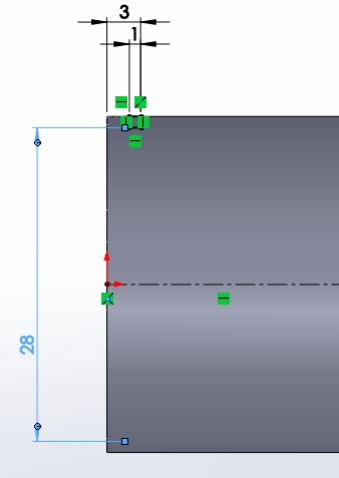


Fig37

Press Esc to stop dimensioning and then select the Revolved Cut tool in the *Features* toolbar (*Fig38*).

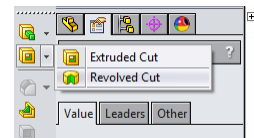


Fig38

Overview the settings of the *Cut-Revolve* dialog (*Fig39*)! There is only one *CenterLine* in the sketch so it is selected as the *Axis of Revolution*; furthermore *SolidWorks* supposes that a complete turnaround (360°) is necessary. Accept the settings! The groove is finished (*Fig40*).

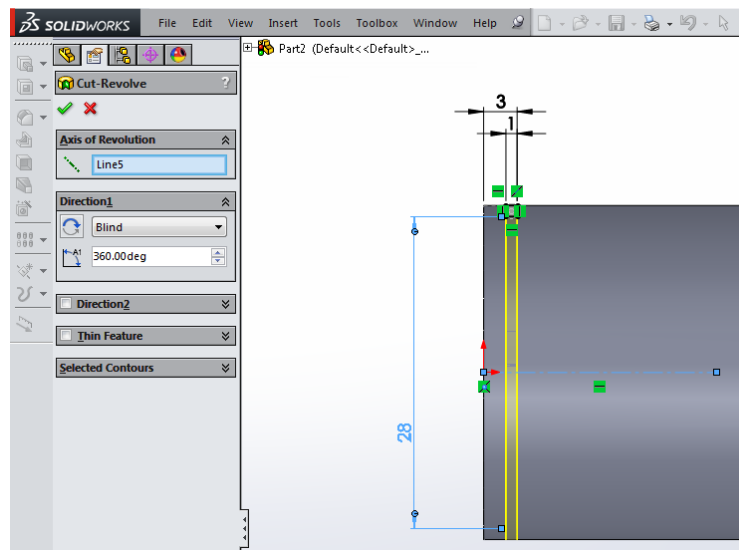


Fig39

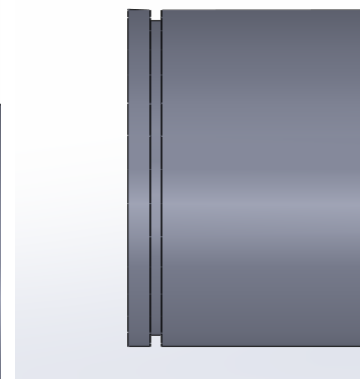


Fig40

The other detail of the far end of the part is a square flattening. This feature is a material removal (*Extruded Cut*) which requires a sketch. Change the viewing direction to see the corresponding end face of the part! Select the circular face and start sketching (*Fig41*)! Set the viewing direction *Normal To* the sketch plane. The content of the sketch will be a square which is standing on one of its corners. The center of the square will be *Coincident* with the center of the selected face.

The *Corner Rectangle* tool is not applicable in this case because the edges of the square are neither horizontal nor vertical.

Use the *Line* tool on the Sketch toolbar (*Fig42*) to draw a general tetragon. Don't accept any offered help (horizontalities, verticalities, parallelities, perpendicularities) from *SolidWorks* – the necessary relations will be defined later. The only aspect to take into consideration is that the sequence of the four *Lines* must be closed (*Fig43*).

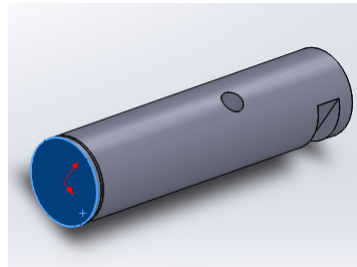


Fig41

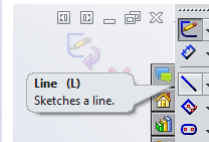


Fig42

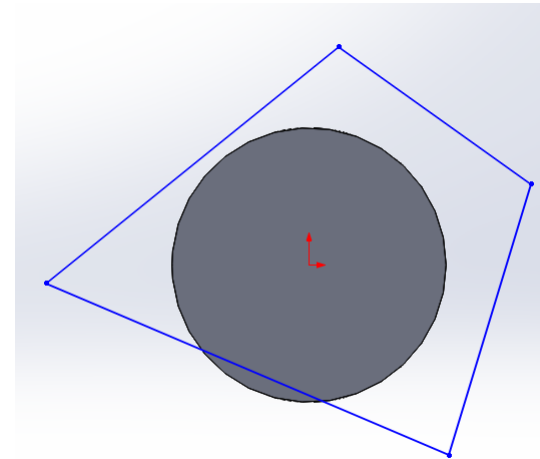


Fig43

Select all of the four Lines! This can be performed several ways:

- Clicking on the *Lines* after each other while pressing *Ctrl* on the keyboard;
- Pressing *Ctrl-A* on the keyboard selects all of the sketch entities;
- Dragging a rectangular window around the tetragon from left to *Right* selects every entity which are completely inside the window (Fig44);
- Dragging a rectangular window around the tetragon from *Right* to left selects every entity which are completely inside the window or intersected by the window (Fig45).

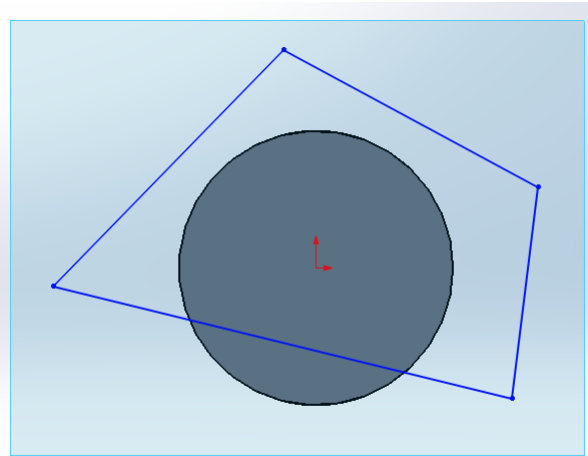


Fig44

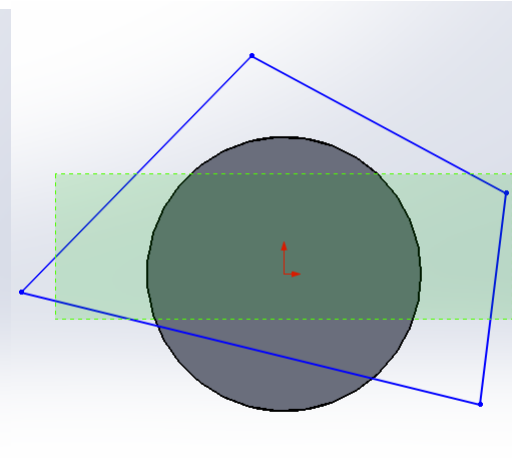
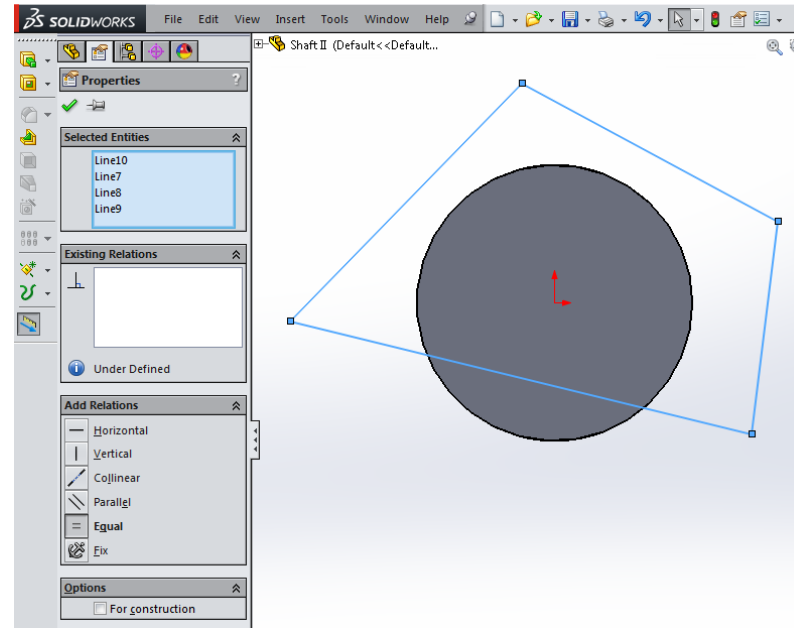


Fig45

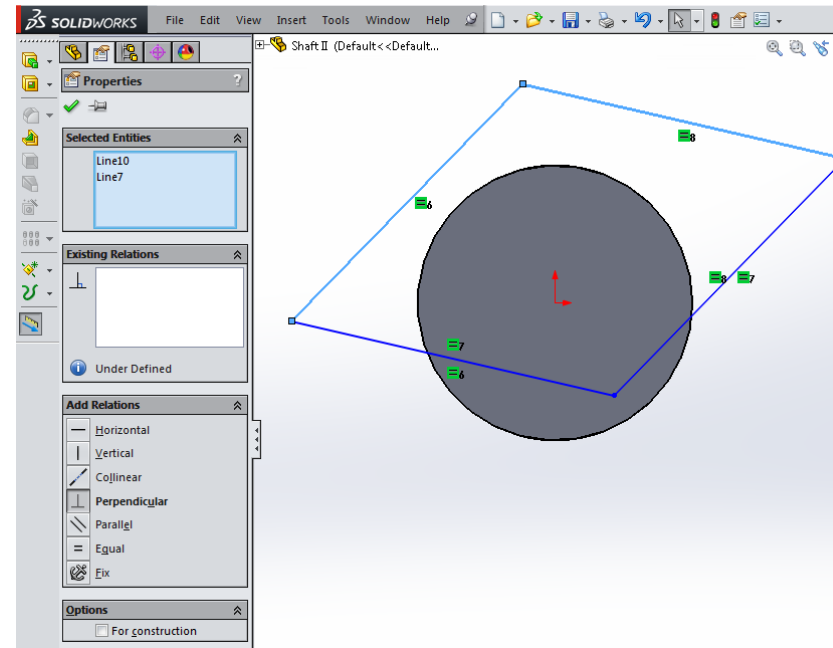
Examine the *Selected Entities* field in the *Properties* dialog. Make certain that all of the four *Lines* are selected. Select the *Equal* relation (Fig46). The tetragon has become a rhombus.

Fig46



A rhombus is a square if one of its corners is perpendicular. Press *Esc* to deselect the entities! Select two adjacent edges of the rhombus and choose the *Perpendicular* relation (Fig47)! The rhombus has become a square.

Fig47



The *Orientation* of the square is not defined yet. The square is standing on one of its corners so its diagonals are horizontal/vertical. Sketch a *CenterLine* which connects two opposite corners of the square (Fig48)! After sketching the diagonal choose the Vertical relation (Fig49)!

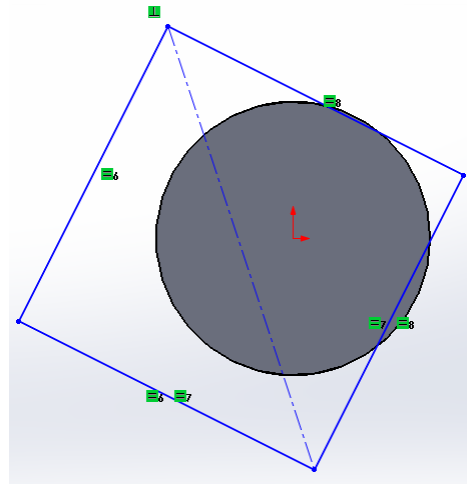


Fig48

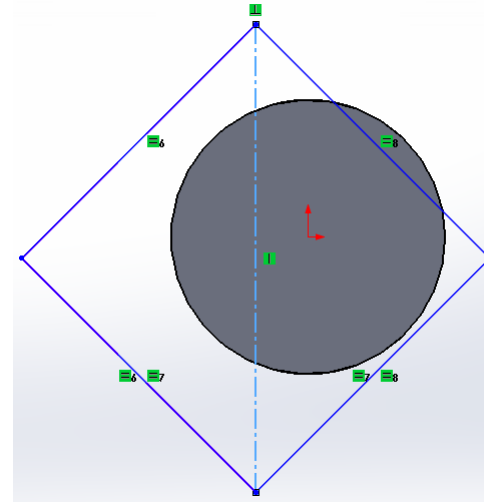


Fig49

The center of the square and the center of the circular face of the part (which is at the origin) are *Coincident*. Unfortunately the center of the square is not displayed in the sketch. Fortunately we know that the center of a square is the *Midpoint* of its diagonals.

The diagonal is selected. Press *Ctrl* and select the origin of the sketch! Choose the *Midpoint* relation! The square is centered (Fig50).

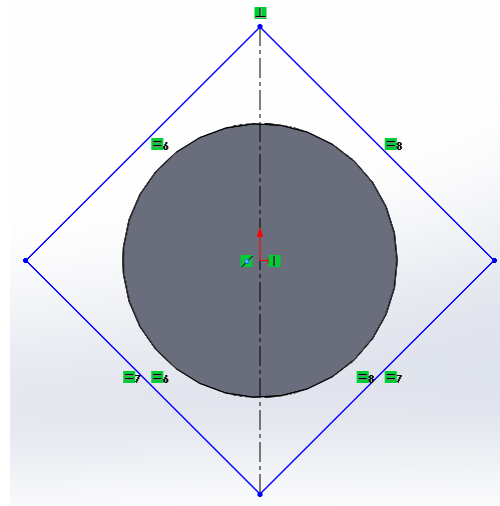


Fig50

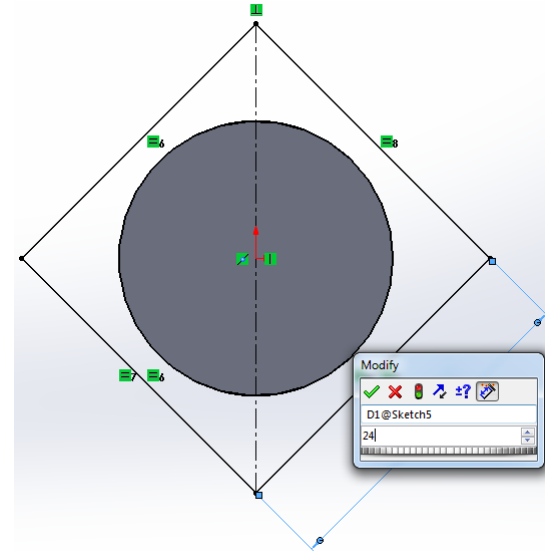


Fig51

Dimension one of the edges of the square! Notice that the *Orientation* of the dimension (horizontal, vertical or parallel to the dimensioned segment) depends on the placement of the dimension number! Place the dimension number so as to dimension the total length of the segment and type in the value (Fig51).

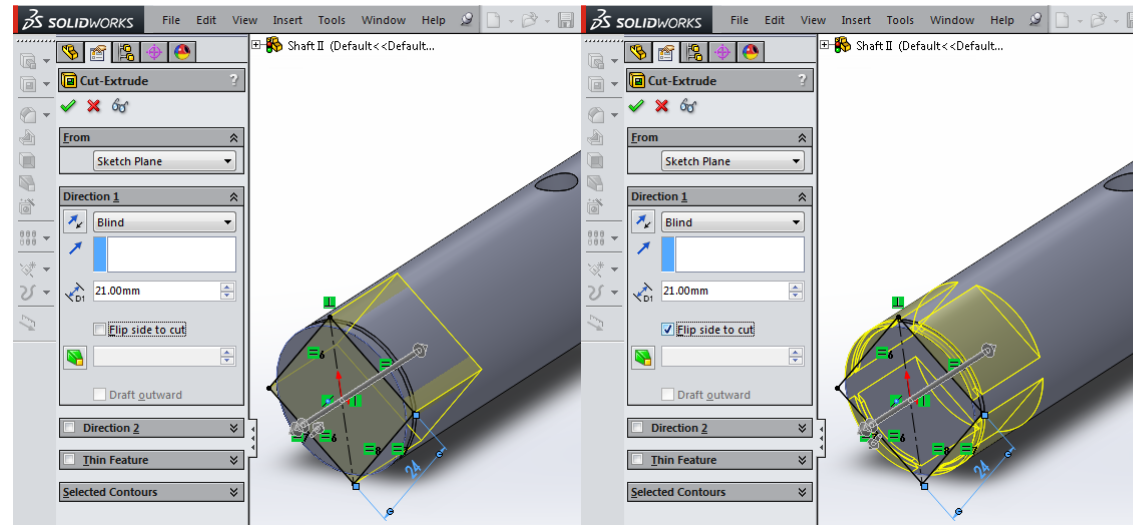


Fig52

Fig53

Select the *Cut Extrude* tool in the *Features* toolbar! The length of the cut is 21 mm. As the graphical preview displays the material of the part will be removed inside the square (Fig52). This is not we want so check the *Flip side to cut* checkbox to turn the cut inside out (Fig53).

The final step would be the chamfering the edges at both ends of the part. It is easier to select the elements to be chamfered if you drag the *Rollback Bar* before the last cut (Fig54).

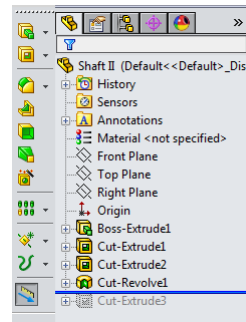


Fig54

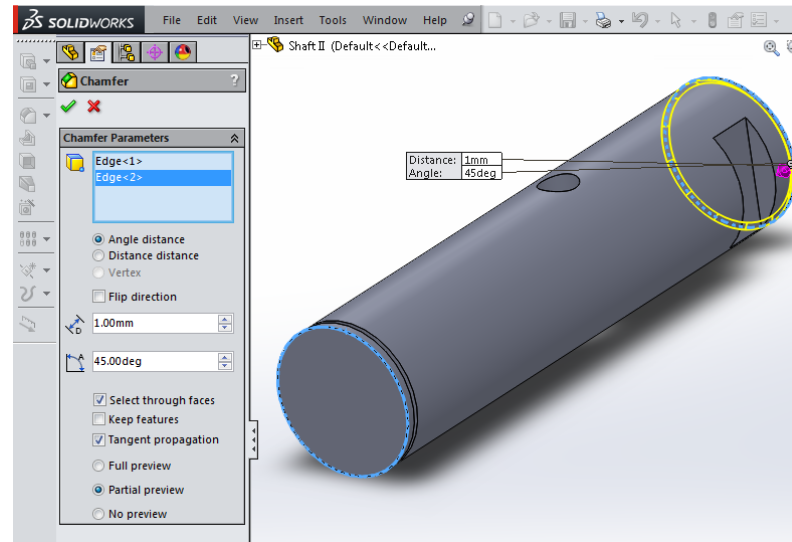
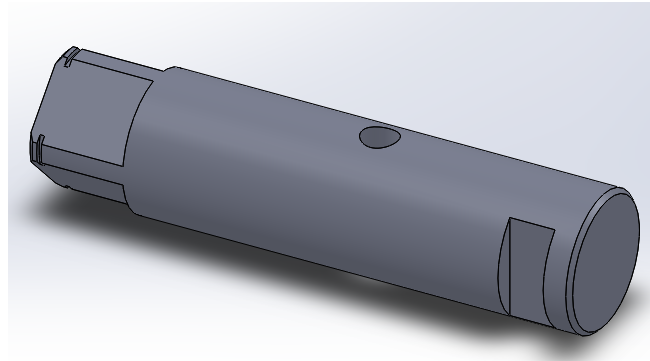


Fig55

Perform the chamfering (Fig55) and drag the *Rollback Bar* to end of the feature list!

Modeling of the part is completed (Fig56).

Fig56



The part document can be downloaded from the following link: [Shaft II.SLDPRT](#)

4. fejezet

4.1 Housing

The *Housing* of the ratchet mechanism is made of sheet metal bent to a U-shape. The constant wall thickness (8 mm) is an important property of the part.

It is instructive to see some wrong methods creating the main volume of the part. We will see that although these solutions create the necessary geometry but they are incorrect.

THE 1ST WRONG SOLUTION

Create a cuboid! The sketch of the feature is placed onto the *Front Plane*. It contains a rectangle. The width of the rectangle is 60 mm while the height is 100 mm. The origin is at the mid-point of the lower horizontal edge of the rectangle (*Fig1*).

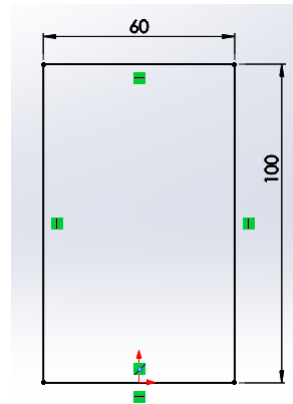
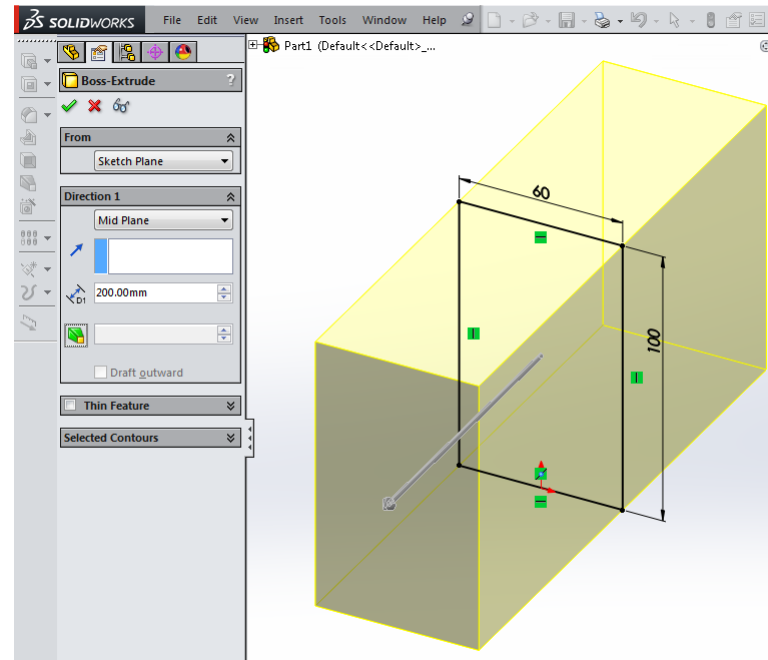


Fig1

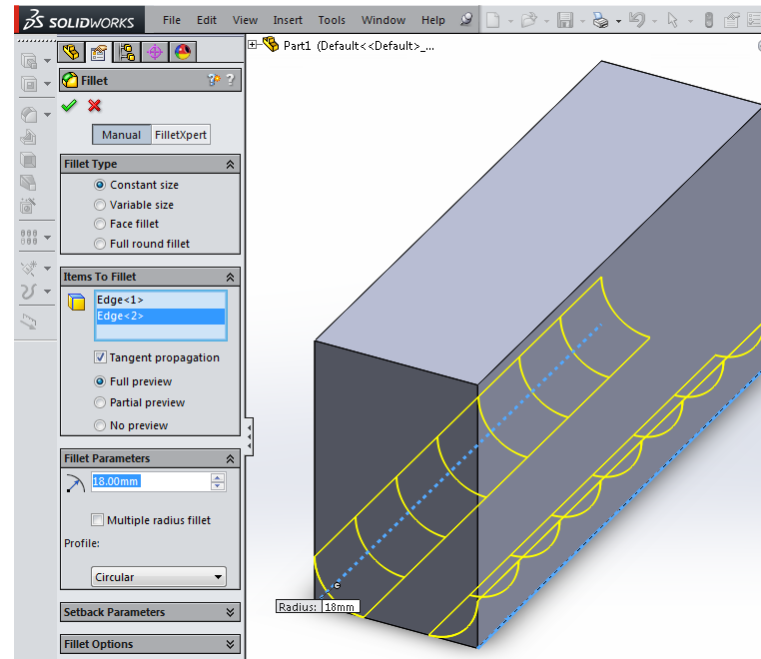
Extrude the rectangle using the Mid Plane option (Fig2)!

Fig2



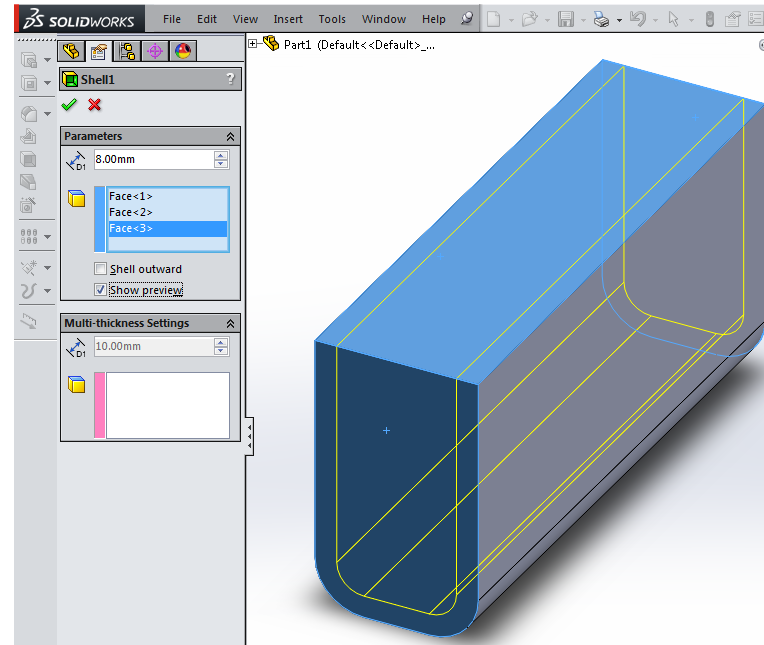
Fillet the lower longitudinal edges of the part (Fig3)! The radius of the fillet is 18 mm (the inner radius of the bend plus the wall thickness).

Fig3



Create a Shell feature to ensure the constant wall thickness (Fig4)!

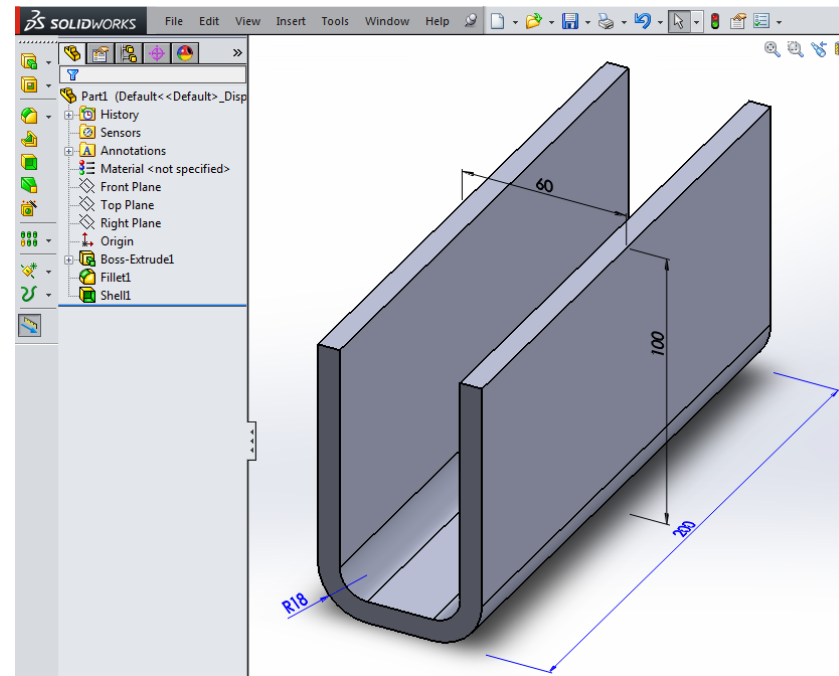
Fig4



Evaluate the model (Fig5)!

- It contains moderate number of features – this is fine;
- It uses simple sketches – this is fine;
- It contains a dimension which does not appear on the drawing – this is problematic. The technologically relevant information is the inner radius of the bend which is determined by the bending tool but the model contains the outer radius. If the wall thickness would be changed later it would alter the bending radius.

Fig5

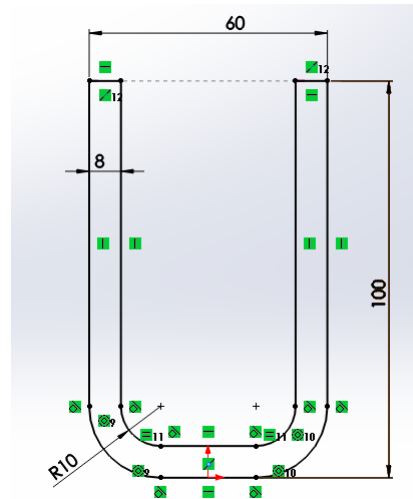


THE 2ND WRONG SOLUTION

Discard the previous part document and start a new one! To eliminate the disadvantage of the previous solution create a sketch which contains the complete cross-section of the part.

The sketch plane is the *Front Plane*. It contains the U-shape of Fig6.

Fig6



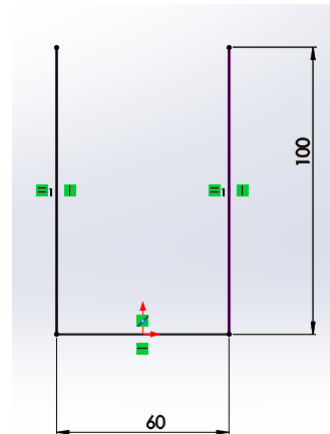
It would be possible to extrude the sketch but it is unnecessary. This solution is surely wrong because the sketch is terribly complicated. It contains a lot of entities (8 Lines and 4 circular arcs) and countless relations. Such complicated sketches are almost impossible to edit and consume too much computer resources.

There must be a solution which uses simple sketches and contains *Right* dimensions.

THE GOOD SOLUTION

Select the *Front Plane* and start sketching! This sketch will contain an absolutely simple U-shape. Draw, relate and dimension three interconnected *Lines* according to Fig7.

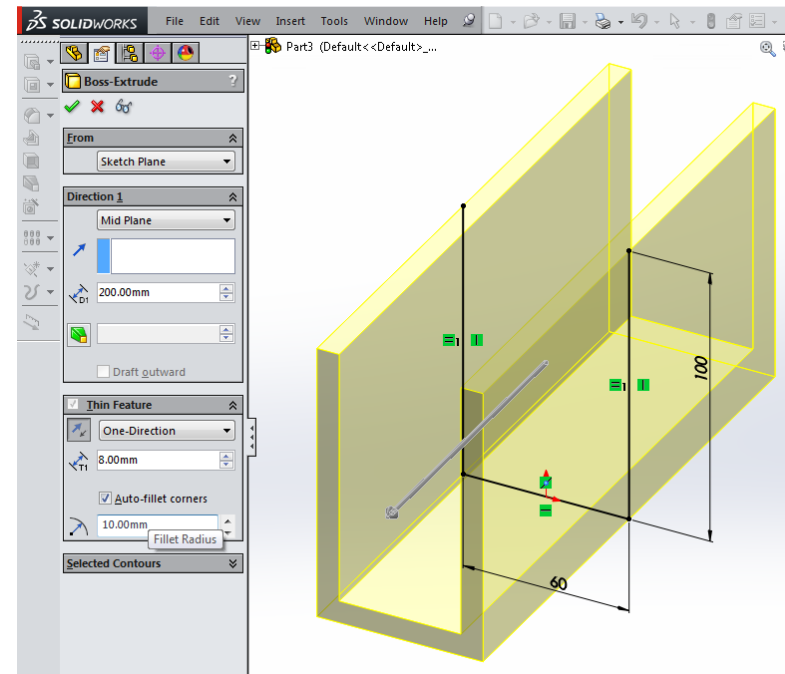
Fig7



- The first and the third segments are *Vertical* – the middle one is *Horizontal*;
- The two vertical segments are *Equal*;
- The origin of the sketch is at the *Midpoint* of the horizontal segment.

Notice that the *Extruded Boss/Base* tool in the *Features* toolbar is active. Although the sketch seems to be overly simple it is possible to extrude it. Because the sketch is open it can be used only to create a Thin Feature (Fig8) – notice that the *Thin Feature* checkbox is grey.

Fig8



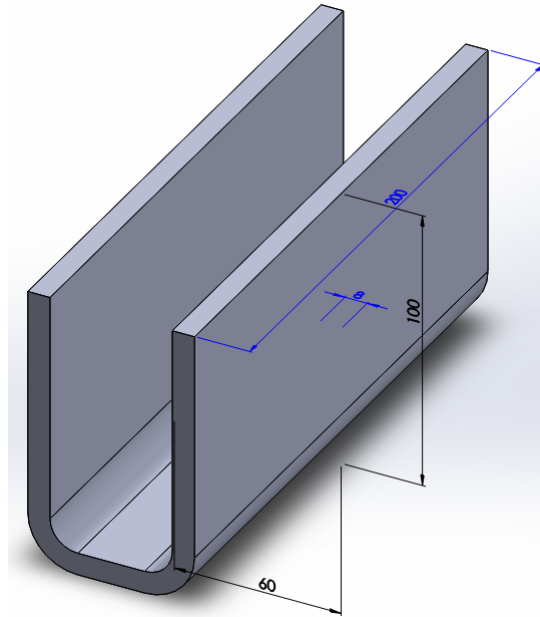
Set the length of the extrusion to 200 mm using the *Mid Plane* option and observe the settings of the *Thin Feature* group!

- The feature will be created applying a thickness on the sketch entities. The thickness is measured along the sketch plane. Set the thickness to 8 mm!
- By default the thickness stretches outward from the sketch. Press the Reverse Direction button to modify this setting. This way the 60 mm width of the U-shape remains the outer measurement of the part.
- The Auto-fillet corners option creates a round at each edge where sketch *Lines* meet at an angle. The Fillet Radius field sets the inside radius of the round.

Accept the settings and examine the result (*Fig9*)! This solution is *Right* because

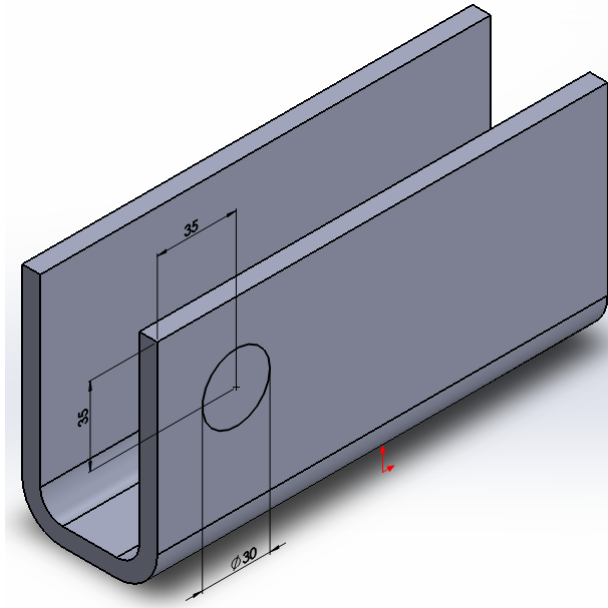
- It contains the simplest sketch possible;
- It uses only one feature;
- It contains exactly the necessary dimensions.

Fig9



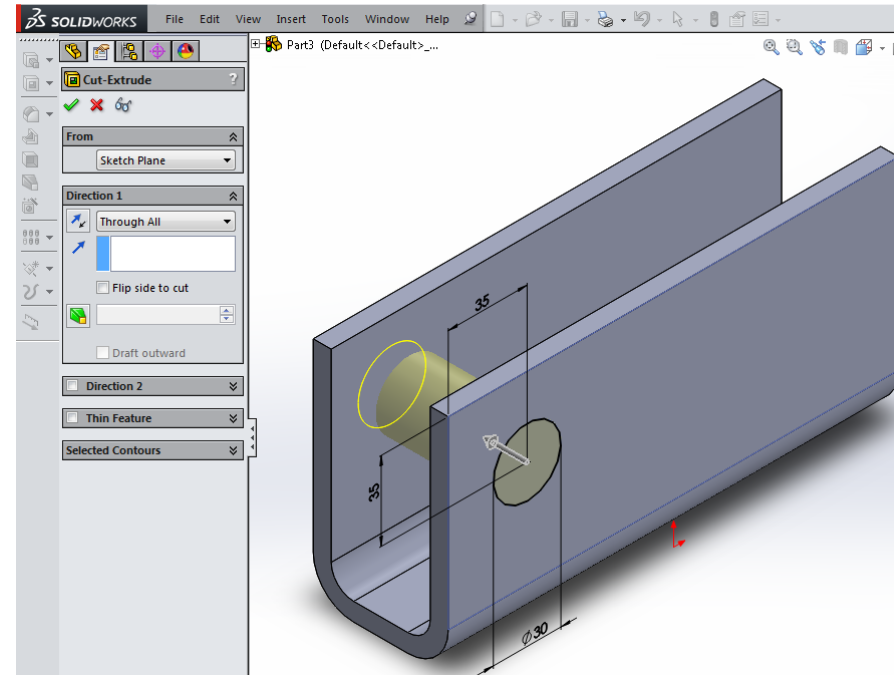
The part contains four identical holes (4 x Ø30). Select the side face of the part and start sketching! Sketch a *Circle* and dimension it according to *Fig10*.

Fig10



Create an *Extruded Cut* feature using the *Through All* option (Fig11).

Fig11



Two of the four identical holes are completed. The remaining holes will be created duplicating the originals by patterning. Patterns are groups of repeated features. This time we use a rather simple *Linear* pattern – it will contain two columns and one row.

Select the *Linear* Pattern tool in the *Features* toolbar (Fig12)!

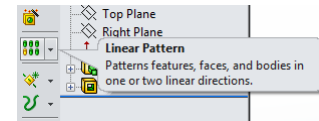
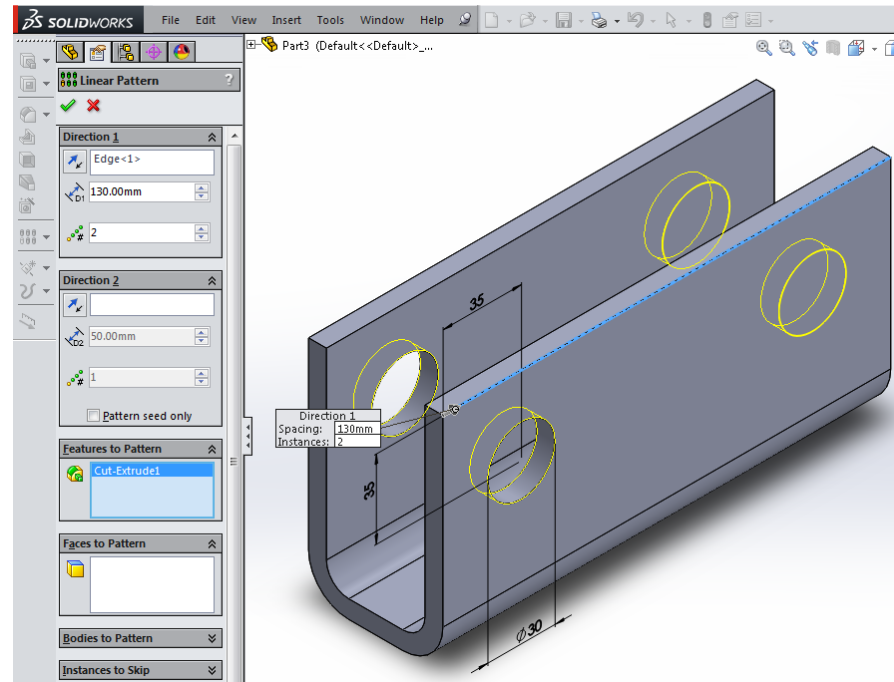


Fig12

Although the *Linear Pattern* dialog defines two directions only Direction 1 is important now because the holes will be patterned in one direction. The first thing is to define the direction of the pattern. This can be done by selecting an edge pointing in the appropriate direction – one of the longitudinal edges of the part is suitable (Fig13). Next type the values for the spacing and the number of the repeated elements into the corresponding fields! Finally click on the cylindrical surface of the original hole to select it for patterning! Accept the settings!

Fig13



The part contains two smaller holes. These holes are M6 threaded holes. Such holes and other types as well can be created using the Hole Wizard. Select the Hole Wizard tool in the Features toolbar (*Fig14*).

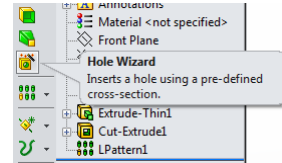
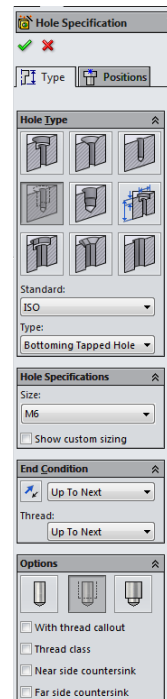


Fig14

Examine the settings of the Type tab of the Hole Specification dialog! For reasons of brevity it is not possible to elucidate all of the options – just copy the setup of *Fig15*.

Fig15



After the hole setup switch to the *Positions* tab of the dialog! Select the face of the part on which the holes will be drilled! As you can realize we are editing a sketch. Turn the sketch plane *Normal To* the viewing direction and place two *Point* entities following the placement of the holes approximately (Fig16).

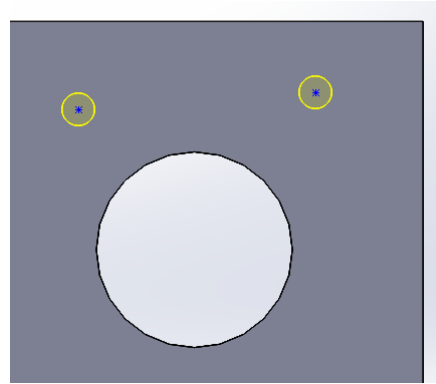


Fig16

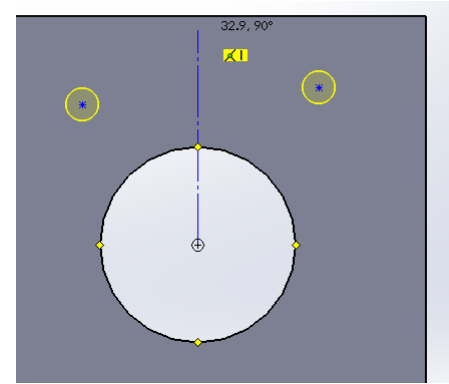


Fig17

The sketch can contain other entities too. The threaded holes are positioned symmetrically to the vertical axis of large hole – notice the elongated symmetry *Line* on the drawing of the part (**Error! Reference source not found.**). Sketch a *CenterLine*! The starting point of the *CenterLine* is the center of the large hole. To find the center hover the pointer over the circular edge of the large hole – *SolidWorks* reveals the characteristic points of the circle. Click on the center and drag the *CenterLine* vertically. Place the endpoint - the actual length of the *CenterLine* is not important. Press *Esc* to stop drawing more segments.

Select the two *Point* entities and the *CenterLine*! Define the Symmetric relation (Fig18)!

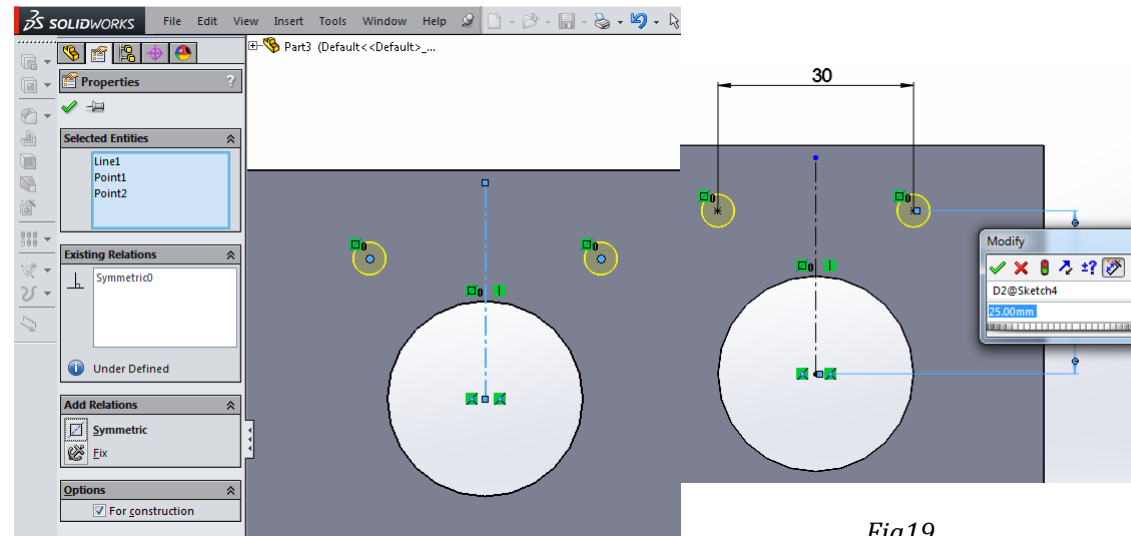


Fig19

Perform the dimensioning (Fig19) and accept the sketch. The pair of the threaded holes is completed.

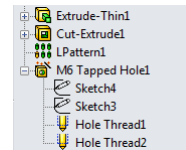


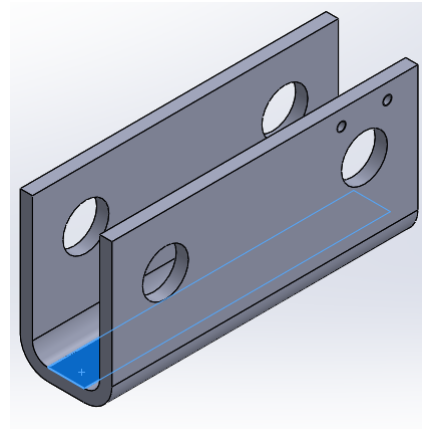
Fig20

Observe the *M6 Tapped Hole1* item in the *Feature Manager* (Fig20)! It contains

- A sketch for the hole centers – this the sketch what has been edited by us previously;
- A sketch for the hole profile – this sketch was created by *SolidWorks* according to the hole type settings;
- Two symbols marking that the holes are threaded ones (*Cosmetic Thread*).

The next feature is the cutout at the bottom of the part. Select the inner horizontal face of the part (Fig21) and start sketching! Set the viewing direction *Normal To* the sketch plane!

Fig21



The sketch will contain *Lines* and a tangentially connected circular arc. Such tangential connections between *Lines* and arcs are very common so there is a built-in trick in *SolidWorks* what makes sketching easier.

Select the *Line* tool and sketch a vertical segment! The rubberband shows the preview of the next segment (Fig22). Drag the rubberband back to its starting position and then move the pointer away – the current *Line* segment temporarily changes to a circular arc (Fig23).

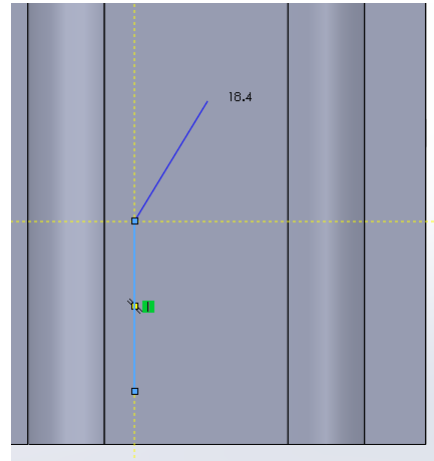


Fig22

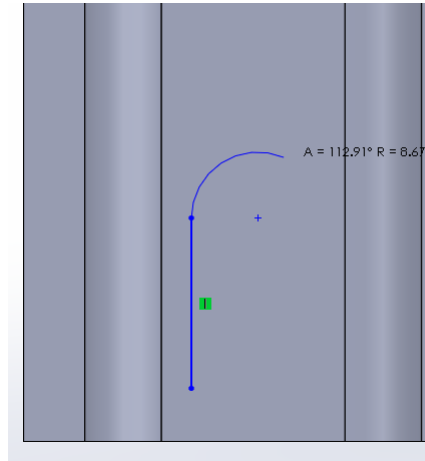


Fig23

Notice that the connection of the two entities depends on the direction in which you move the pointer away from the starting point of the current element.

- If the pointer leaves the starting point approximately in the direction of the previous segment the result will be a tangential arc (Fig23);
- If the pointer leaves the starting point approximately along the previous segment the result will be an oppositely orientated tangential arc (Fig24);
- If the pointer leaves the starting point approximately perpendicular to the previous segment the result will be an arc which is connected to the previous segment perpendicularly (Fig25);

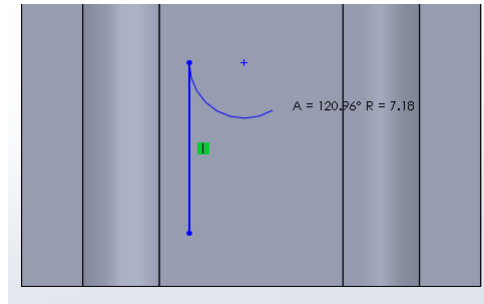


Fig24

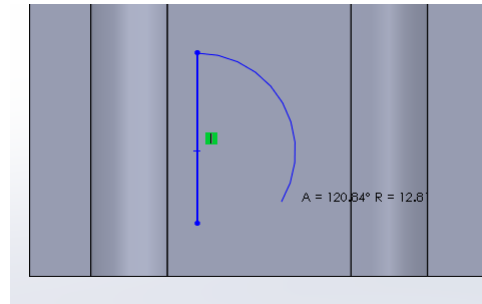


Fig25

Move the pointer so as to get a tangential half circle (Fig26).

- The endpoint of the arc is located horizontally to the starting point;
- The central angle of the arc is 180°.

Place the endpoint of the half circle! The next entity in the chain is a *Linear* segment again – the transition was temporal. Sketch a segment vertically downward. Take care to place the endpoint

- vertically below the starting point and
- horizontally to the starting point of the very first element of the chain (Fig26).

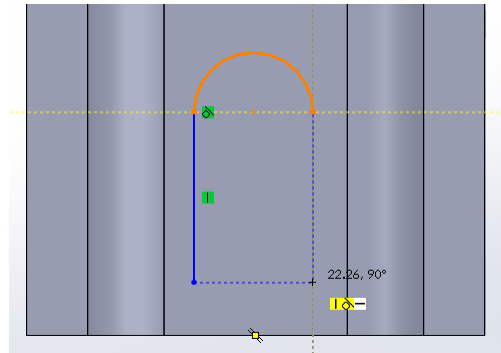


Fig26

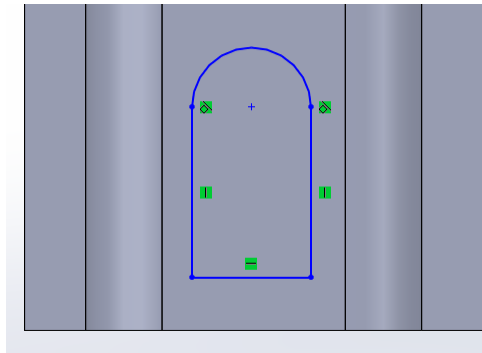
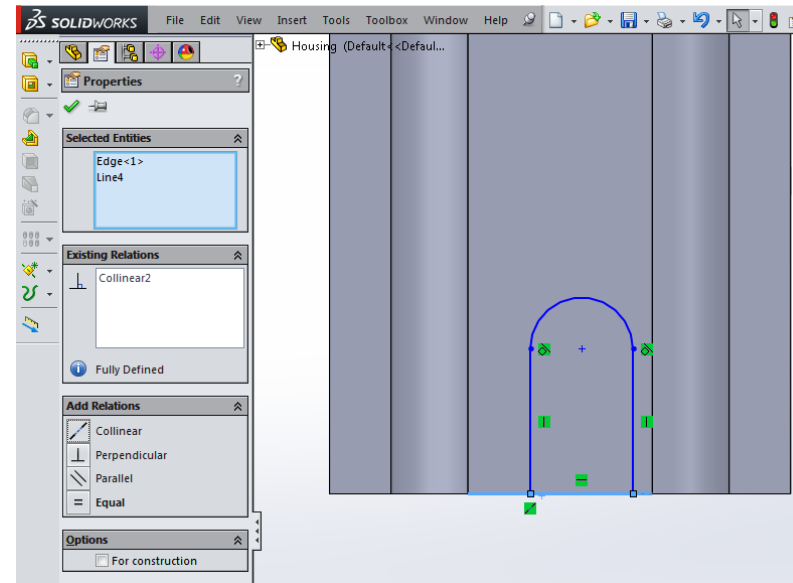


Fig27

Close the chain of entities by a horizontal *Linear* segment. Check Fig27 whether all of the five relations are present on your sketch!

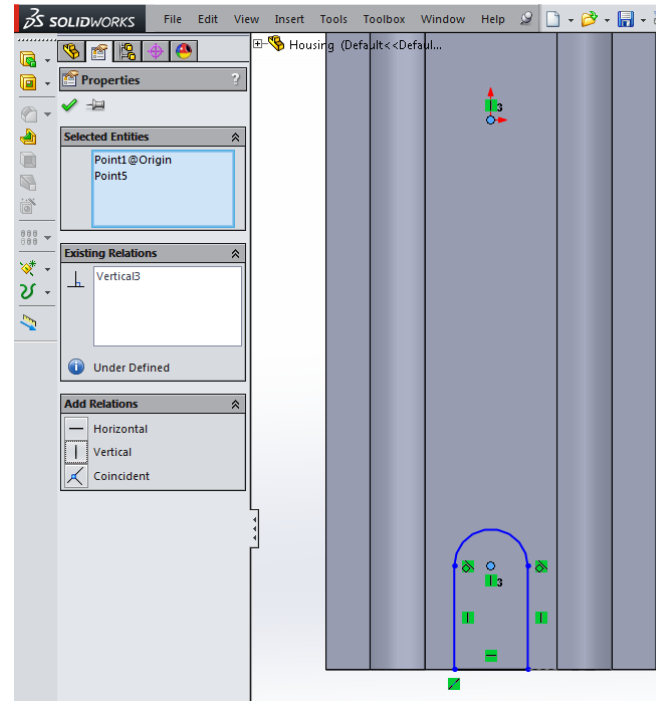
Define the vertical placement of the shape by making the horizontal segment and the lower horizontal edge of the part *ColLinear* (Fig28).

Fig28



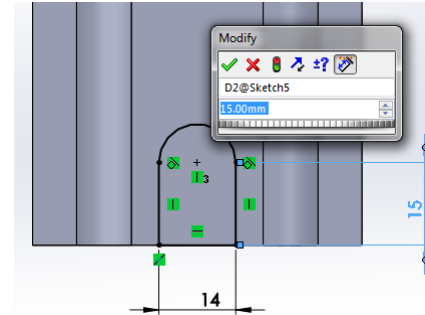
The shape is horizontally centered. Select the center of the half circle and the origin of the sketch then make them Vertical (*Fig29*)!

Fig29



Perform the dimensioning (*Fig30*) and create an *Extruded Cut* feature using the *Through All* option!

Fig30



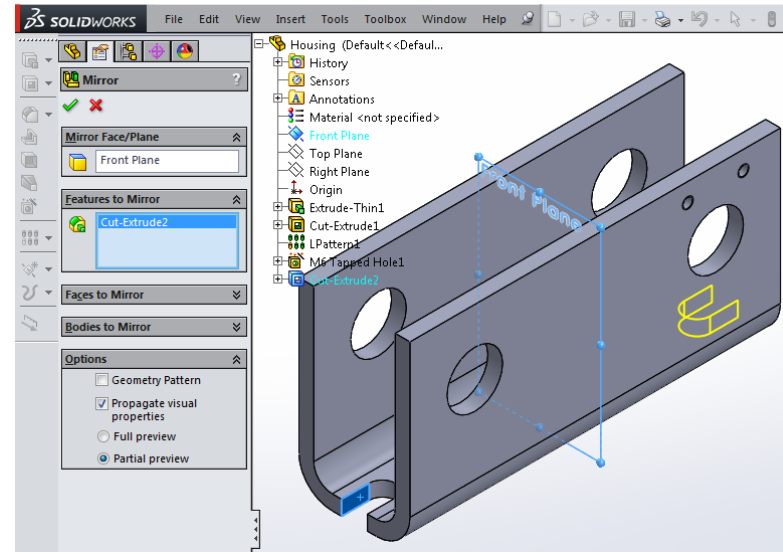
The other cutout is the mirror copy of the first one. Select the Mirror tool in the Features toolbar (Fig31)!



Fig32

The mirror plane is the *Front Plane* – this is why we used the *Mid Plane* option during the first extrusion. To select this plane expand the list of the features which is hovering in the upper left corner of the *Drawing Area* (Fig32) then click on the name of the plane in the list.

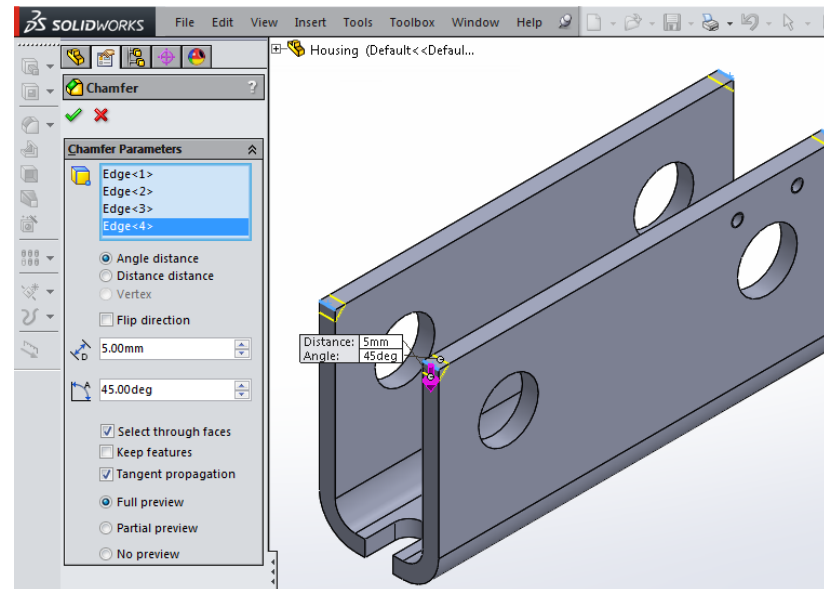
Fig33



The Features to *Mirror field* of the Mirror dialog shall contain the names of the features to be mirrored. The cutout can be selected either clicking on its surface in the *Drawing Area* or clicking on its name (*Cut-Extrude2*) in the list of the features (*Fig33*). Accept the settings!

Chamfer the upper short edges of the part to finish the modeling process (*Fig34*)!

Fig34



The part document can be downloaded from the following link: [Housing.SLDPRT](#)

4.2 Ratchet Wheel

The fundamental shape of the *Ratchet Wheel* is a revolved body. Select the *Front Plane* and start sketching! Set the viewing direction *Normal To* the sketch plane!

Because the sketch will be used for rotation it must contain a *CenterLine*. The starting point of the *CenterLine* is at the origin of the sketch. The direction of it is horizontal (*Fig1*). Change the drawing tool to ordinary *Line* and continue the previously sketched *CenterLine* (*Fig2*)! Take care to close the chain of *Lines* with a vertical segment which ends at the origin of the sketch!



Fig1

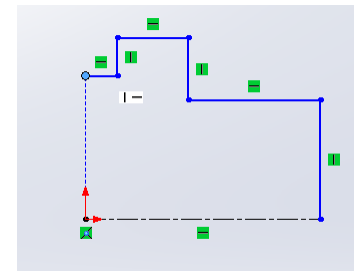
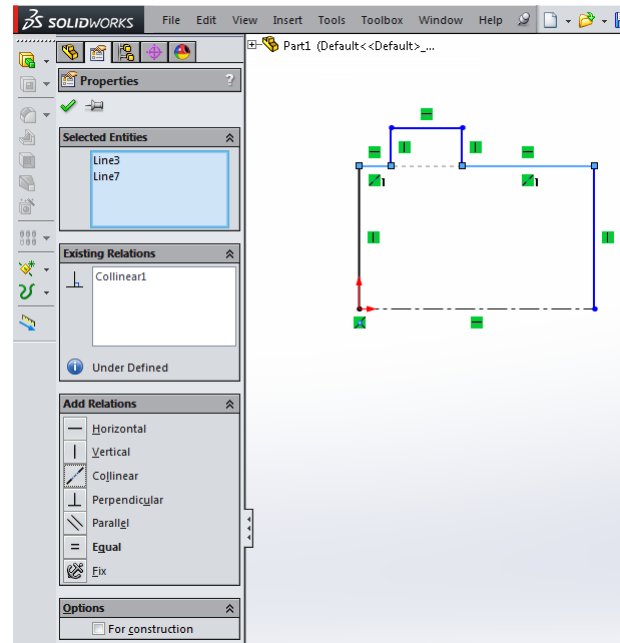


Fig2

Notice on the drawing of the part (**Error! Reference source not found.**) that a local symmetry is defined at the tothing – the diameter of the hub is the same on both sides of the tothing. Build this information into the model – make the left and the *Right* horizontal *Lines Collinear* (*Fig3*).

Fig3



Perform the dimensioning of the horizontal dimensions (Fig4)! Create the diametral dimensions (Fig5)!

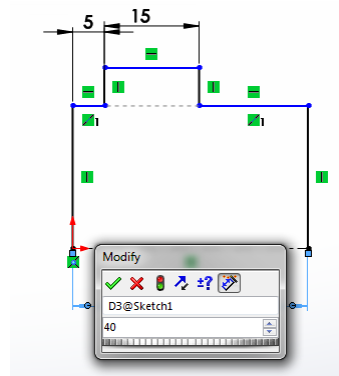


Fig4

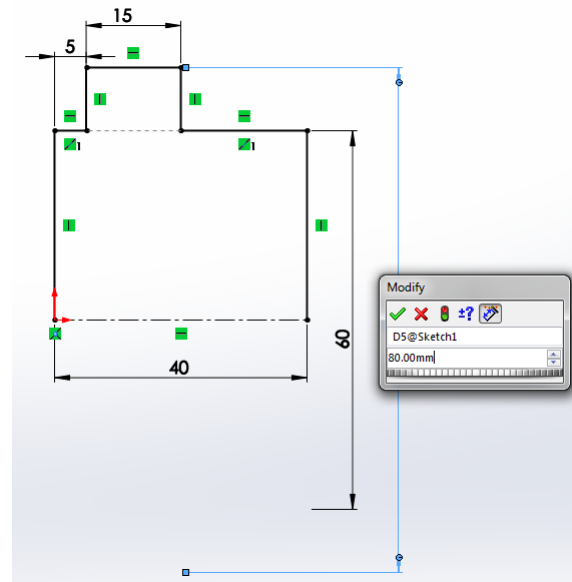


Fig5

The sketch is fully defined. Select the *Revolved Boss/Base* tool in the Features toolbar (Fig6)!

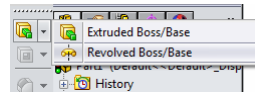


Fig6

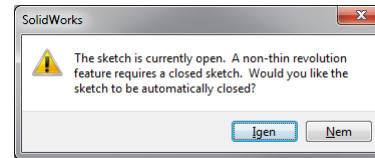
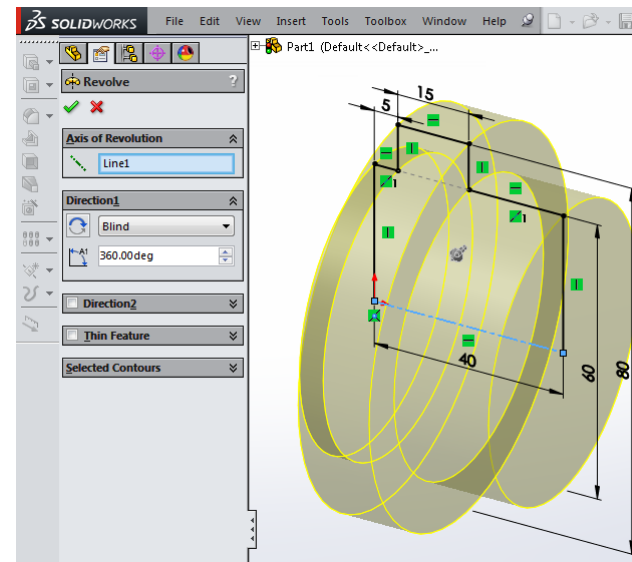


Fig7

A message informs us that the sketch is open – the chain of the continuous *Lines* is not closed (Fig7). This was intentional because this way the selection of the *CenterLine* during dimensioning of the diameters was easier. We want to create a solid body so let *SolidWorks* to complement the sketch – answer Igen (“Yes” in Hungarian).

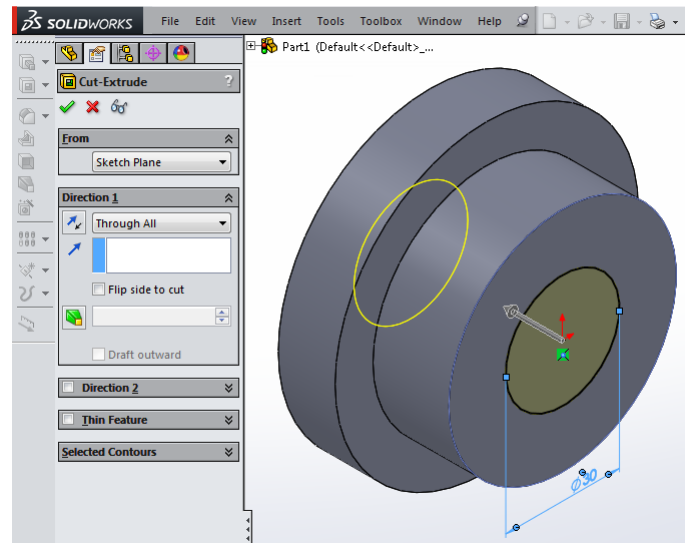
Fig8



The default settings are just fine (one *CenterLine*, complete turnaround) so accept the settings.

Create the large axial hole! Sketch a Circle onto the end face of the part. The center of the circle is at the origin of the sketch. Dimension the diameter and create an *Extruded Cut* feature using the *Through All* option (Fig9).

Fig9



Create the smaller hole! The sketch plane is the *Top Plane*. Sketch a Circle and make the center of the circle and the origin of the sketch *Horizontal*! Dimension the diameter of the circle and the distance between the circle and the end face of the part (Fig10)!

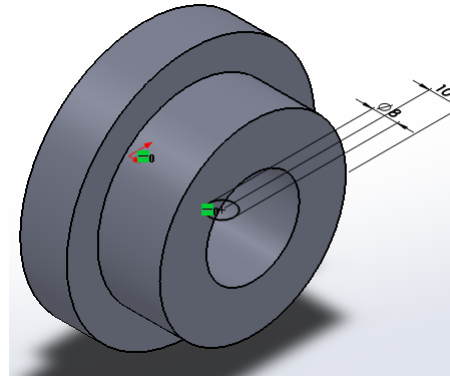


Fig10

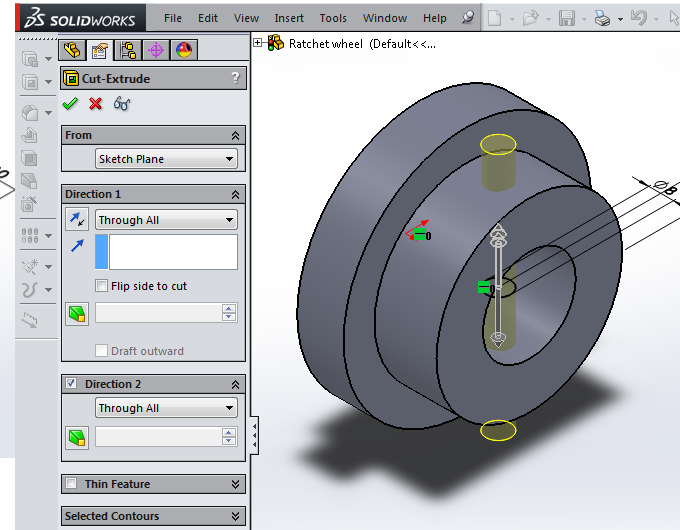


Fig11

Transform the sketch to an *Extruded Cut* feature using the *Through All* option in both directions (Fig11).

The toothing of the gear consists of ten identical teeth – and ten identical spaces between the teeth. The first space will be created as a cut and then the remaining ones will be patterned around.

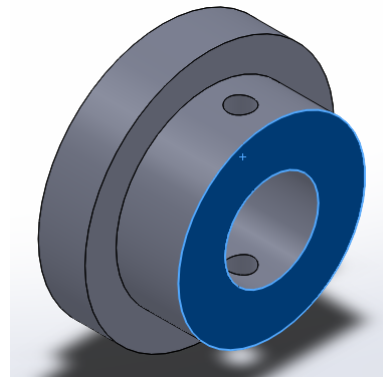


Fig12

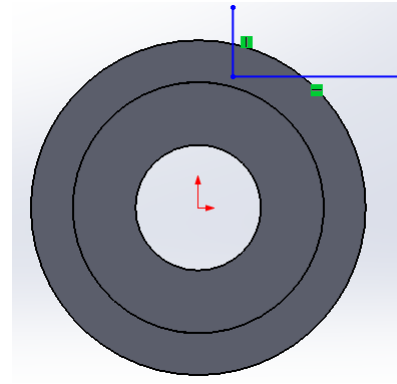


Fig13

Select the end face of the part and start sketching (Fig12)! Set the viewing direction *Normal To* the sketch plane! Sketch two interconnected *Lines* according to Fig13. The sketch is not closed yet – it shall contain a portion of the outLine of the part.

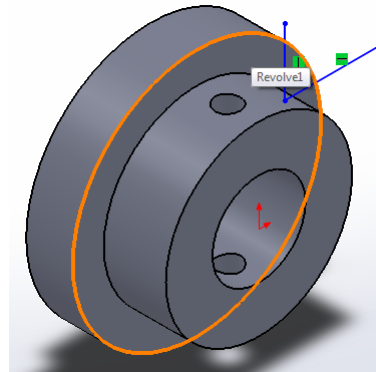


Fig14

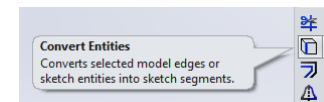


Fig15

Return to the previous axonometric view and select the large circular edge which forms the outLine of the part looking perpendicularly to the sketch plane (Fig14)! Select the *Convert Entities* tool in the *Sketch* toolbar (Fig15)! The projection of the selected edge appears on the sketch plane as a sketch entity (Fig16). Notice the *On Edge* relation which makes sure that this converted entity will always follow the changes of the selected edge (Fig17).

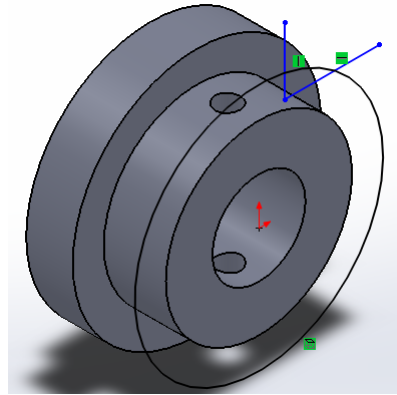


Fig16

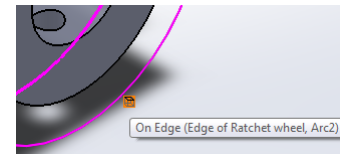


Fig17

The content of the sketch is too much. The *Lines* are too long and only a smaller arc of the converted circle is necessary. The unwanted elements can be removed using the *Trim Entities* tool in the *Sketch* toolbar (Fig18).

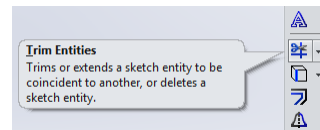


Fig18

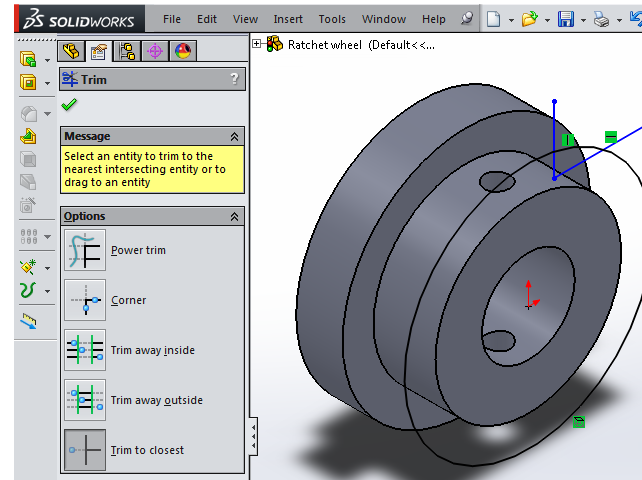


Fig19

Select the Trim to closest option and click on the unnecessary portions of the sketch entities (Fig19)!

Perform the dimensioning of the remained shape according to Fig20! Create an Extruded

Cut feature using the *Through All* option (Fig21)!

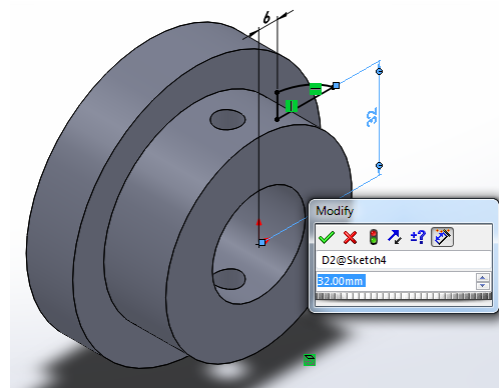


Fig20

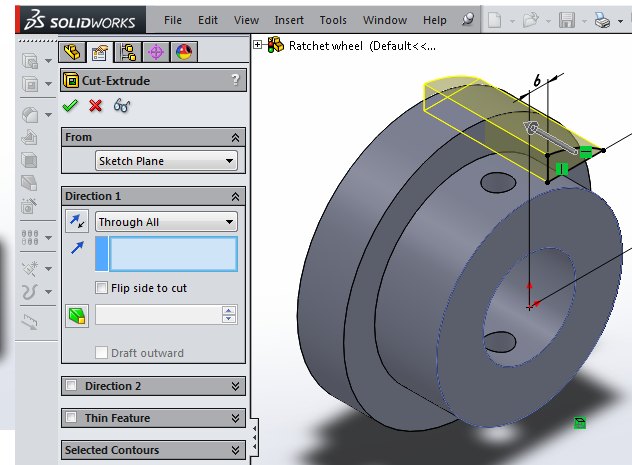
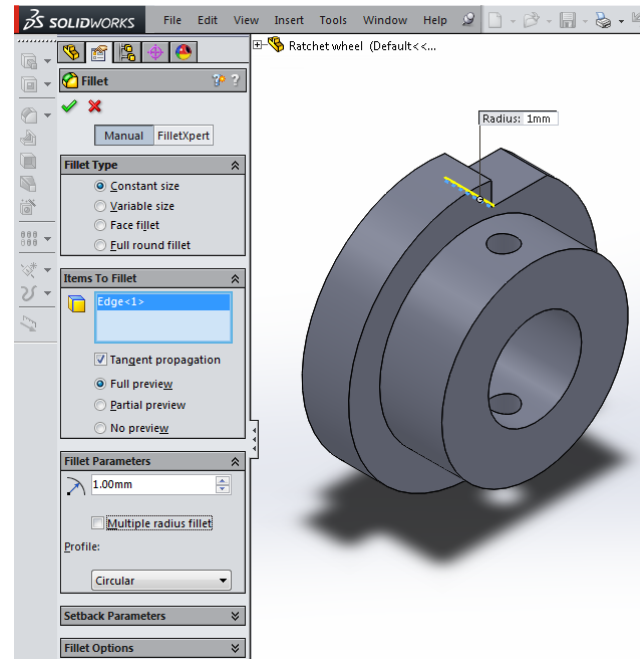


Fig21

The roots of every tooth are filleted. Fillet the edge of the tooth space (Fig22);

Fig22



Select the Circular Pattern tool in the Features toolbar (Fig23)! Select the last two features – the cut and the fillet (Fig24)!

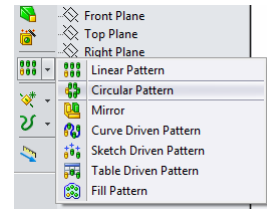


Fig23

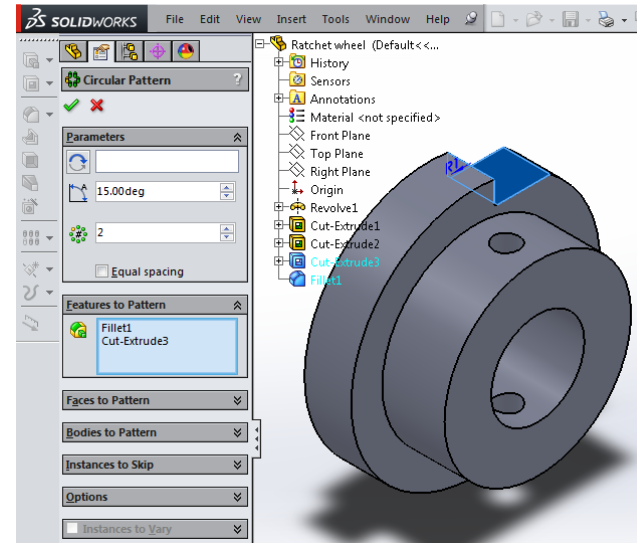
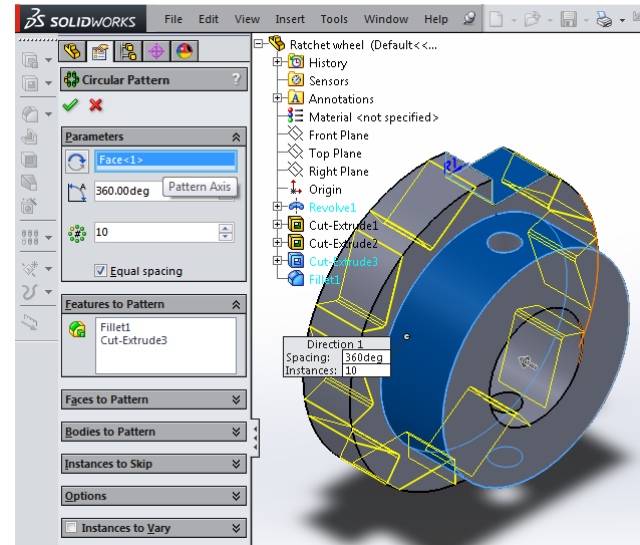


Fig24

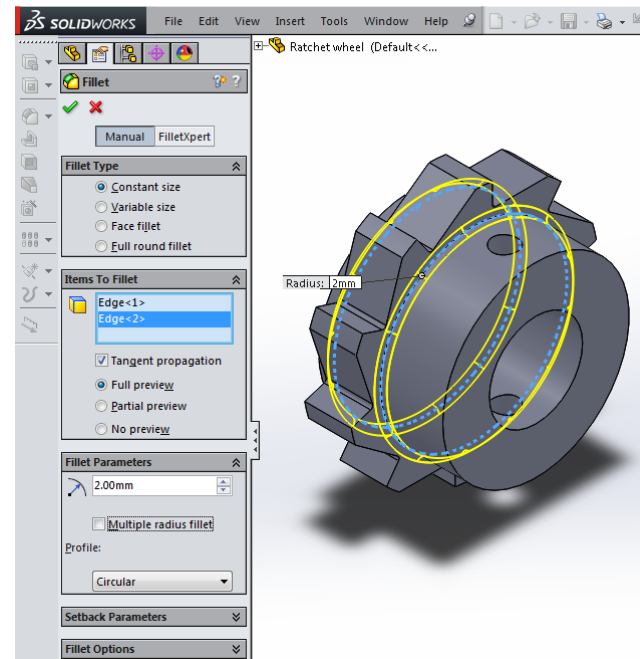
The *Circular Pattern* requires an axis which can be defined by selecting a cylindrical or a conical face of the part. Click in the first field (*Pattern Axis*) of the Parameters group and then select the cylindrical surface of the hub of the part! The selected features will be evenly spaced along a complete circle. Check the *Equal Spacing* checkbox, type 360° into the Angle field and 10 into the *Number of Instances* field (Fig25)! Accept the settings!

Fig25



Fillet the edges on both sides of the tothinging (Fig26)!

Fig26



Chamfer the outer and the inner edges on both end faces of the part (Fig27)! The part is completed (Fig28).

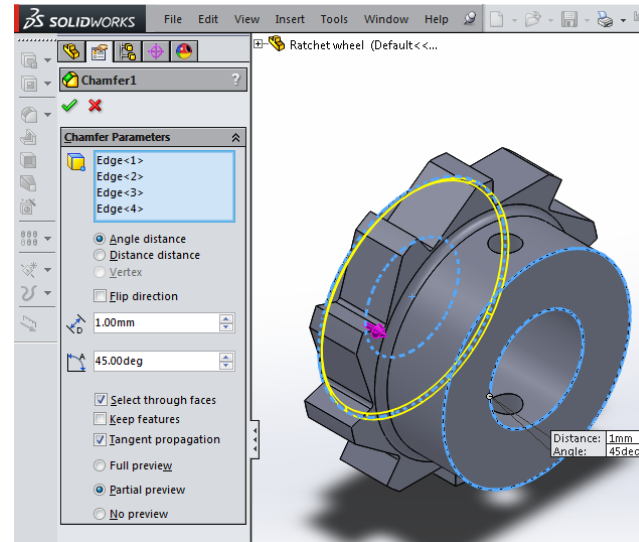


Fig27

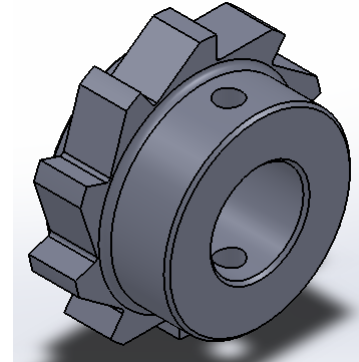


Fig28

The part document can be downloaded from the following link: [Ratchet Wheel.SLDPRT](#)

4.3 Pawl

In a real manufacturing environment the *Pawl* would be created from two separate parts:

- A hub made by turning;
- A sheet metal part which then fixed to the hub by welding.

The dimension network of the part (**Error! Reference source not found.**) reflects this information. It would be possible to model the two individual parts and then to unite them in an *Assembly* but this chapter is about part modeling so the *Pawl* will be created as a contiguous set of material.

Let the hub the first feature of the part – it is a revolved body so a *Sketch* is necessary. Select the *Front Plane* and start sketching! Sketch the to-be axis of rotation (*CenterLine*) and the half profile of the feature (*Fig1*)! Perform the dimensioning according to (*Fig2*)!

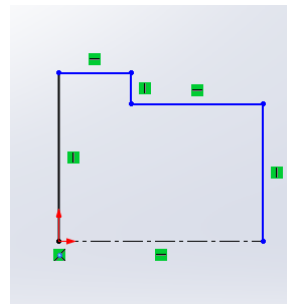


Fig1

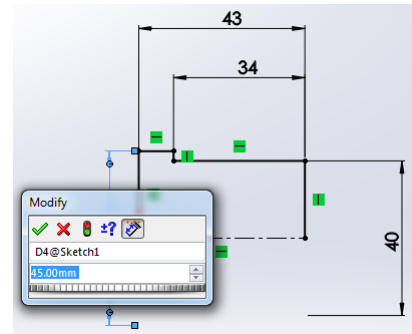


Fig2

Use the *Revolved Boss/Base* tool to transform the sketch to a feature! Answer “*Igen*” (Yes)! The default settings are just fine so accept the settings (*Fig3*);

Fig3

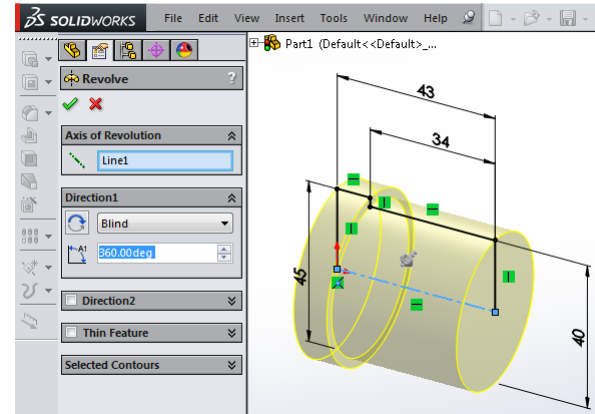


Fig3

Select the narrow circular ring face of the part (*Fig4*) and start sketching! Set the viewing direction *Normal* To the sketch plane!

Fig4

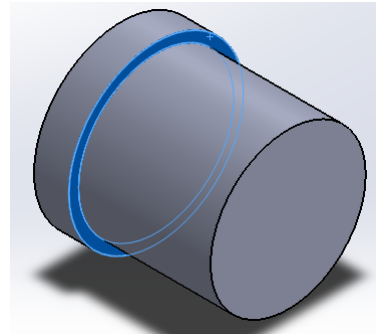


Fig4

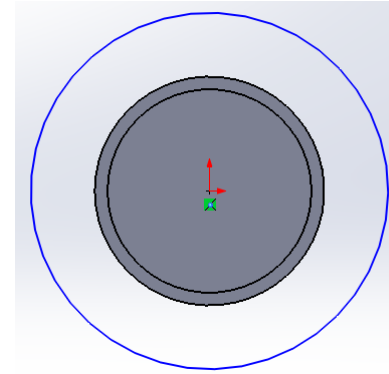
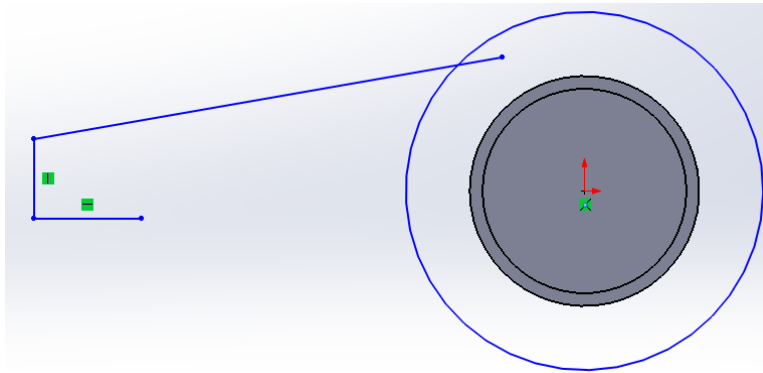


Fig5

Sketch an origin centered circle (Fig5)! Sketch the interconnected *Lines* according to Fig6!

Fig6



Select the *3 Point Arc* tool in the Sketch toolbar (*Fig7*)!

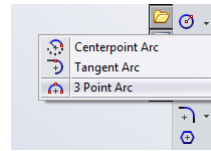


Fig7

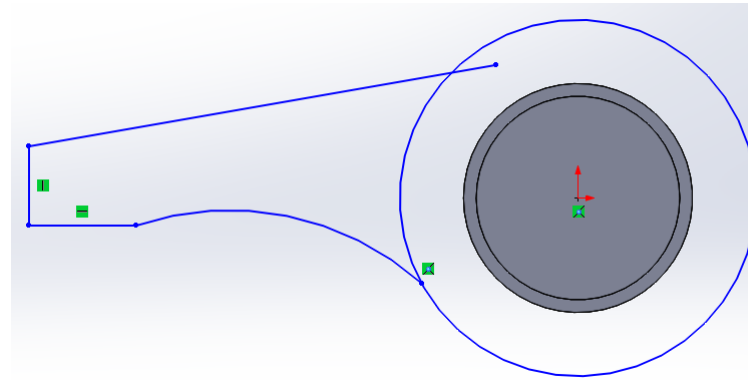


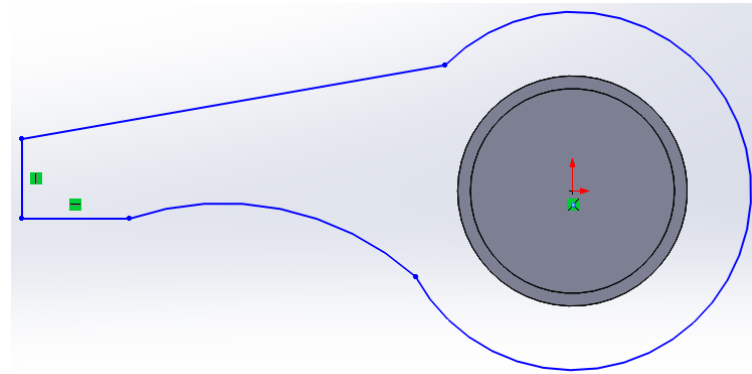
Fig8

Define a circular arc by placing its three points:

- The starting point of the arc is at the free endpoint of the horizontal sketch segment;
- The endpoint of the arc is on the perimeter of the circle;
- An intermediate point is positioned so as to define the curvature of the arc (*Fig8*);

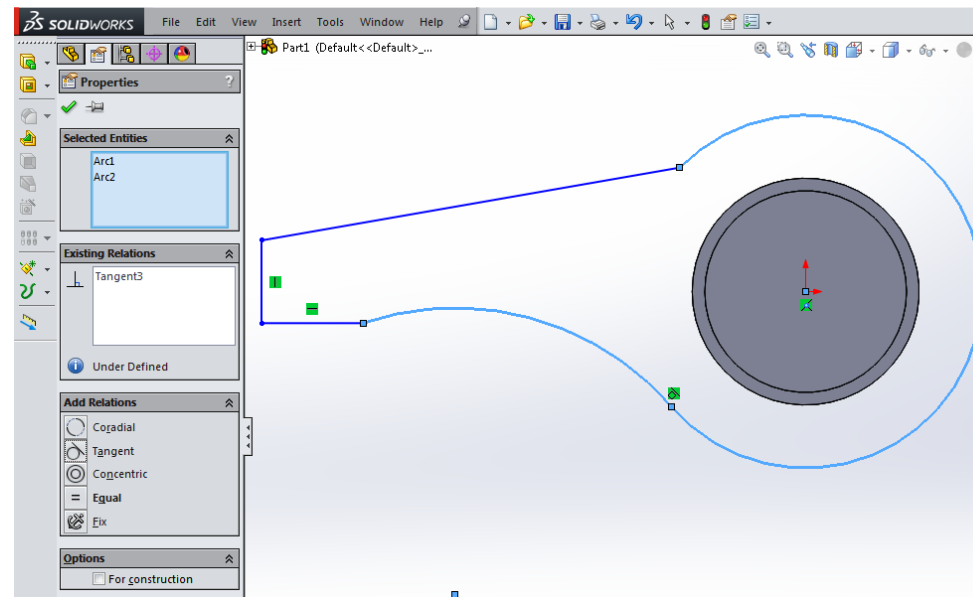
Use the Trim Entities tool to remove the unnecessary elements of the sketch (*Fig9*)!

Fig9



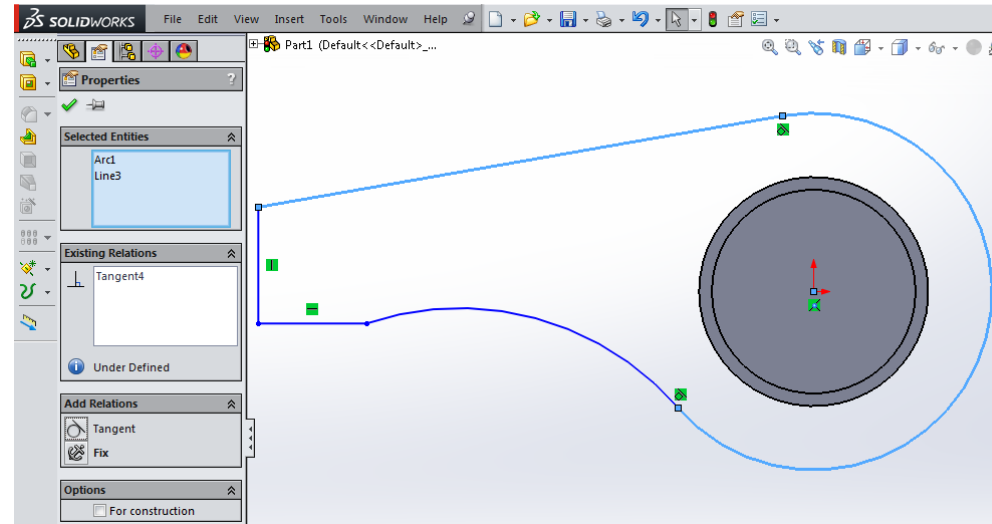
The transition of the two circular arcs is tangential. Select both of the arcs and apply the Tangent relation (Fig10)!

Fig10



Define the tangency for the *Right* arc and the *Line* connecting to it (*Fig11*);

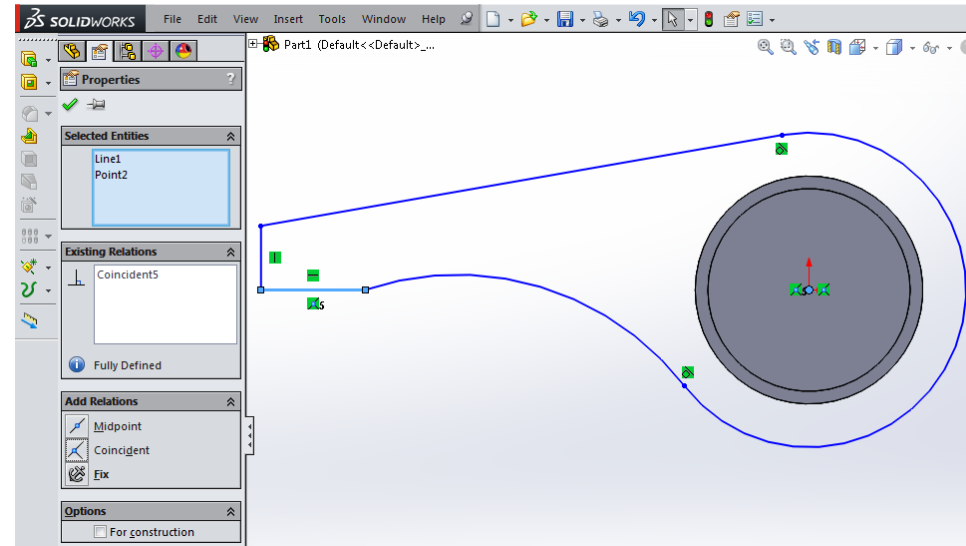
Fig11



It is hard to realize, but the elongation of the horizontal *Line* intersects the *CenterLine* of the hub of the part (**Error! Reference source not found.**).

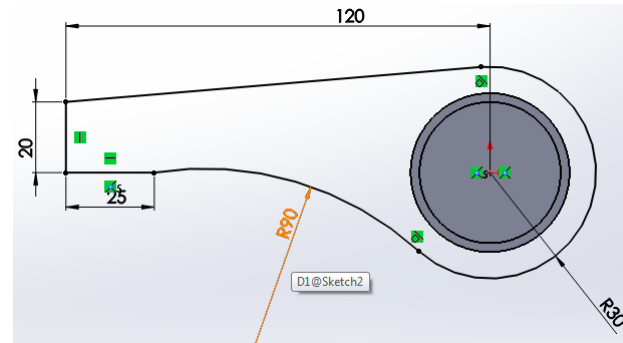
Select the horizontal *Line* and the origin of the sketch (or the center of the *Right* arc) then choose the *Coincident* relation (*Fig12*)!

Fig12



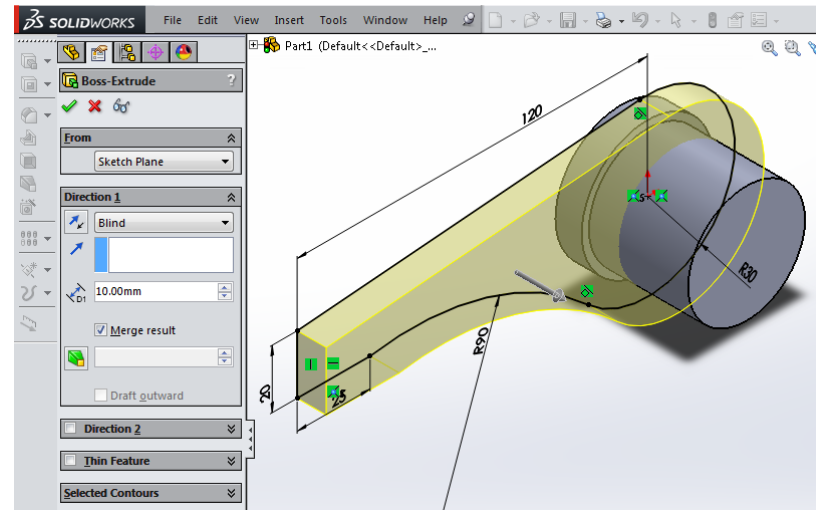
Perform the dimensioning according to Fig13!

Fig13



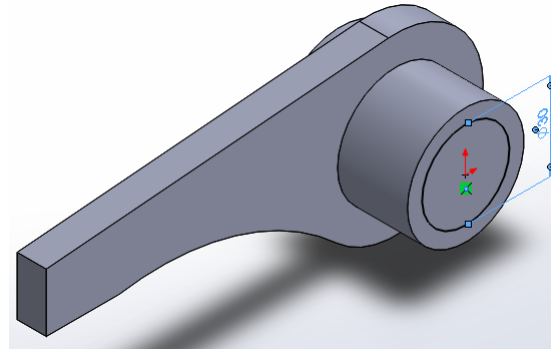
Create an Extruded Boss/Base feature (Fig14);

Fig14



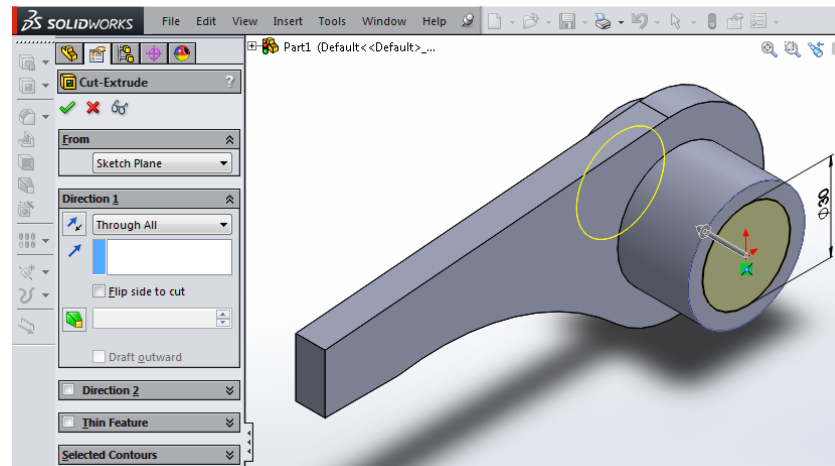
Sketch an origin centered Circle onto the circular face of the part and dimension it (Fig15)!

Fig15



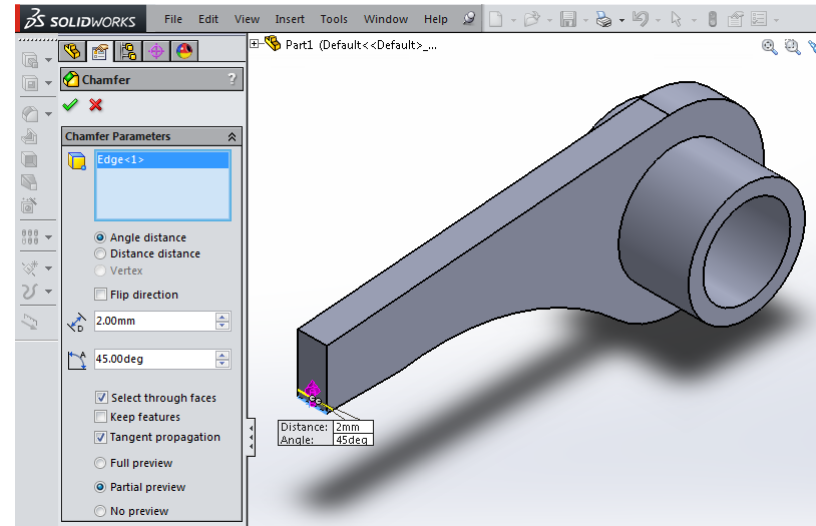
Use the Extruded Cut tool to create a through-hole (Fig16)!

Fig16



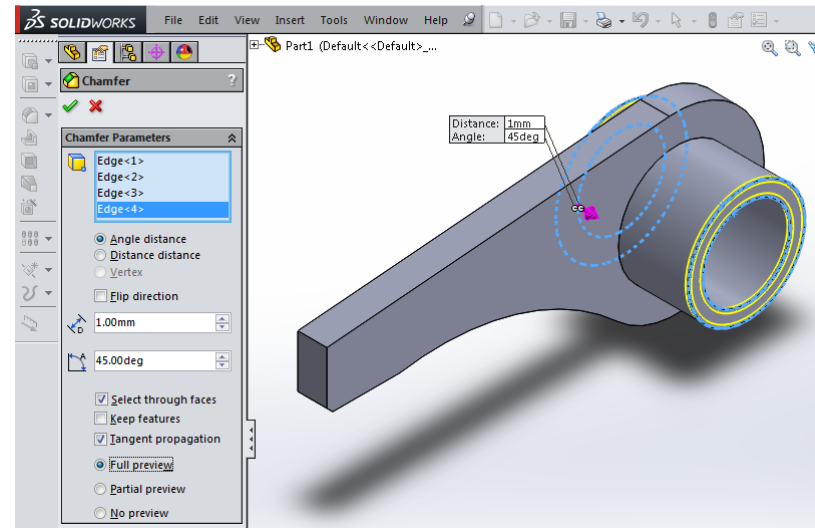
Chamfer the nose of the part (Fig17)!

Fig17



Chamfer the outer and the inner edges on both end faces of the hub (Fig18)! The part is completed.

Fig18



The part document can be downloaded from the following link: [Pawl.SLDPRT](#)

4.4 Spring

The spring is made by bending and coiling a wire. The cross-section of the wire is circular. Such a part can be modeled in *SolidWorks* by *sweeping* – moving a profile along a path.

First the path will be created. It can be divided to three characteristic portions:

- The *Right* end of the path is a planar curve;
- The middle of the path is a *Helix*;
- The left end of the path is a planar curve.

To create a *Helix* sketch a *Circle* onto the !! Set the origin of the sketch to be *Coincident* with the circle! Relate the center of the circle vertically over the origin of the sketch (*Fig1*)!

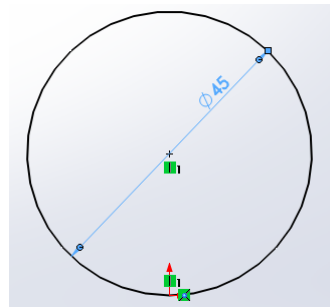


Fig1

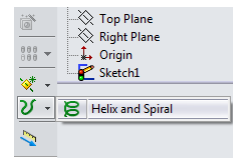
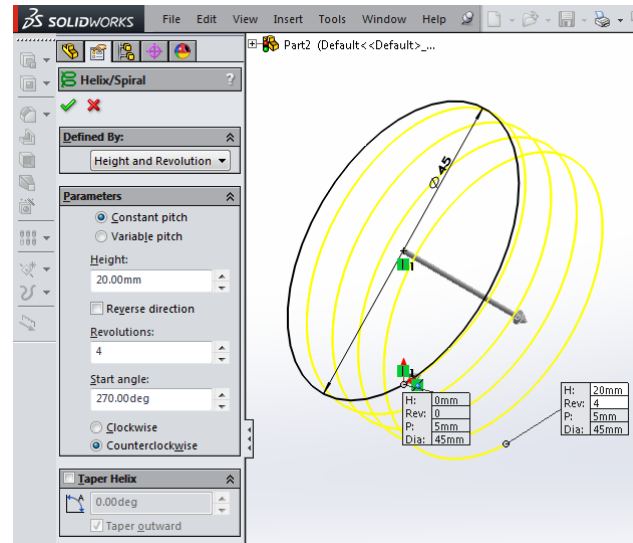


Fig2

Select the *Helix and Spiral* tool in the Curves group of the Features toolbar (*Fig2*)! The *Helix* is defined by its total height and number of revolutions – the pitch is calculated by *SolidWorks*. Set the parameters according to *Fig3*! Notice that the Start angle is 270° so the curve starts at the *Origin*. The number of the Revolutions is an integer therefore the endpoint of the curve is on the *Top Plane*.

Fig3



Click OK to finalize the *Helix*! Select the *Top Plane* and start sketching! Sketch three interconnected *Lines* according to Fig4!

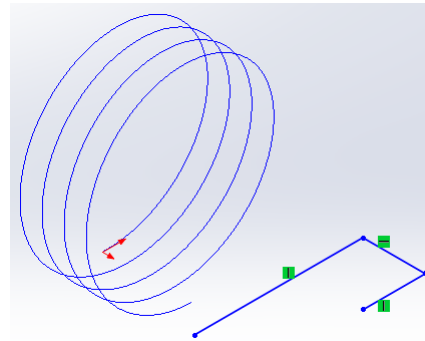


Fig4

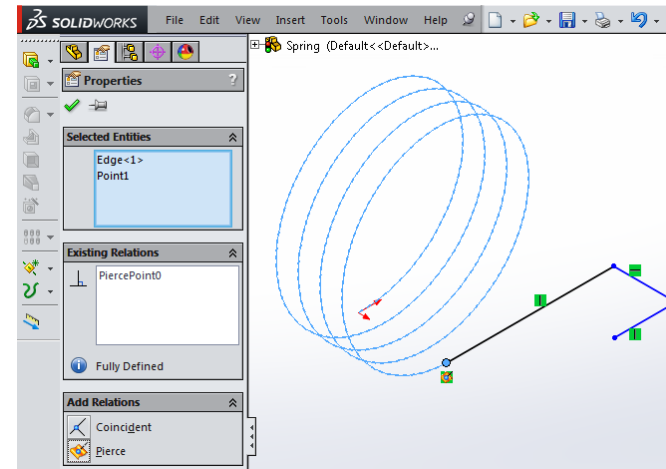
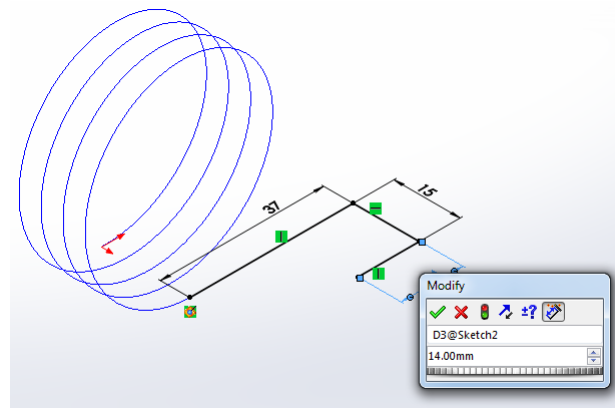


Fig5

Defining the connection between the *Helix* and the current sketch is a bit problematic. If you select the endpoints of the curves *SolidWorks* does not offer the *Coincident* relation. To solve the problem click on the *Helix*; then click on the endpoint of the longer vertical sketch segment pressing *Ctrl*. Select the *Pierce* relation (Fig5)! This way the selected point will be the intersection point (piercing point) of the helical curve and the sketch plane.

Fig6



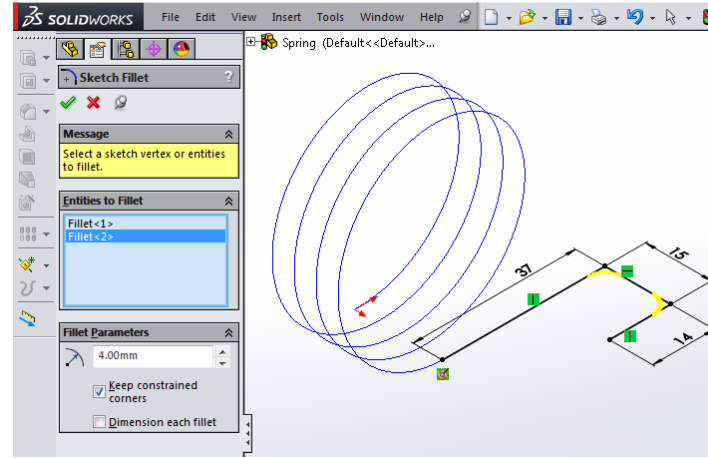
Perform the dimensioning (*Fig6*)! The wire of the spring cannot have sharp bendings so the corners of the sketch must be filleted. Select the Sketch Fillet tool in the Sketch toolbar (*Fig7*)!



Fig7

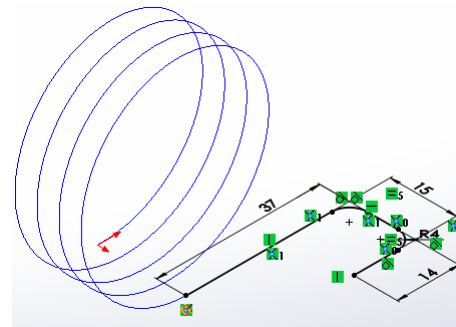
Set up the *Fillet Parameters* (*Fig8*) and click on the corners of the sketch!

Fig8



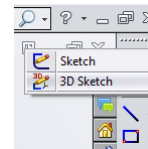
Notice that a lot of relations have appeared. The fillet arcs are tangential to the connecting *Lines*; the arcs are equal etc (Fig9).

Fig9

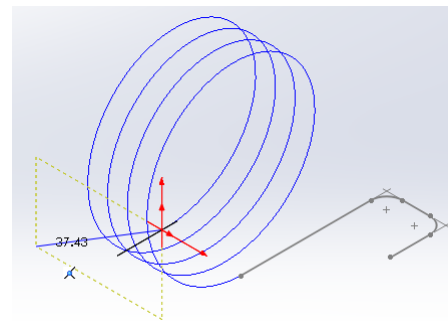
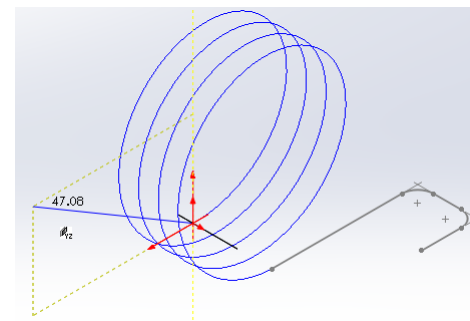


Leave the sketch keep*ing* its changes!

It is not possible to create the curve for the other end of the *Spring* as a sketch – a 3D Sketch is necessary. In a *3D Sketch* the placement of *Lines*, circles and other entities is not limited to a plane – they can be placed and orientated freely in the three dimensional space. Select the *3D Sketch* tool in the Sketch toolbar (*Fig10*)!

*Fig10*

Start sketching a *Line*! A local coordinate system appears. Click on the origin to place the starting point of the *Line*. As the red arrows indicate the currently sketched *Line* segment will lie in the (xy) plane of the local coordinate system (*Fig11*). This is not adequate because we would like to draw the segment parallel to the z axis. Press *Tab* on the keyboard the change the *Orientation* of the coordinate triad (*Fig12*).

*Fig11**Fig12*

Draw a *Line* segment in the direction of the z axis then a next one in the direction of the y axis (Fig13)!

Press *Tab* twice to reach the proper *Orientation* of the coordinate triad (Fig14)!

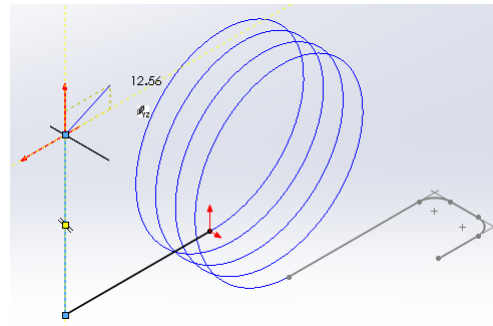


Fig13

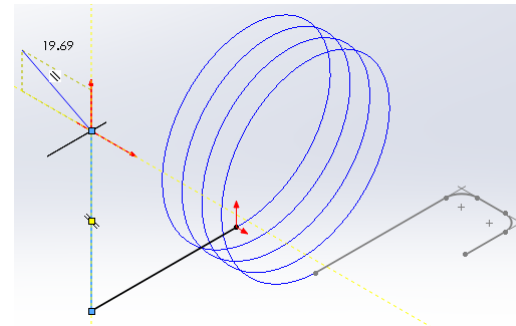


Fig14

Sketch the remaining segments – first in the x direction and then in the y direction (Fig15)! Press *Esc* to stop drawing *Lines*! It is advisable to change the viewing direction to check the proper *Orientation* of the *Lines* (Fig16).

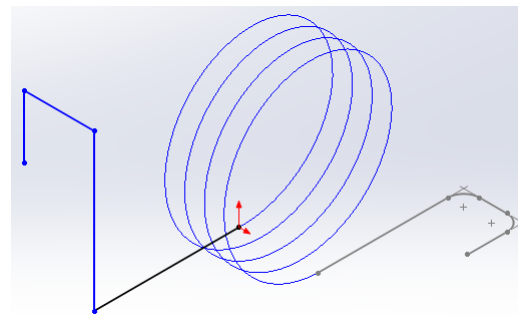


Fig15

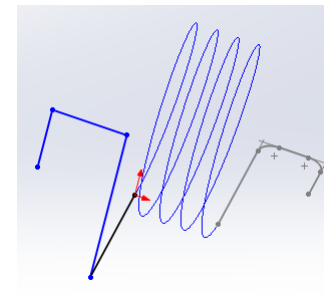
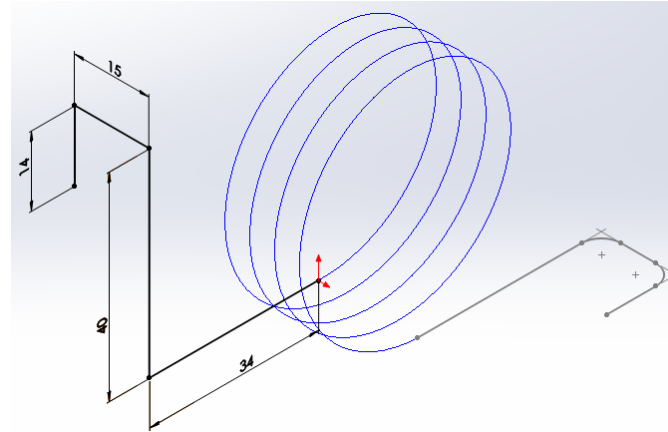


Fig16

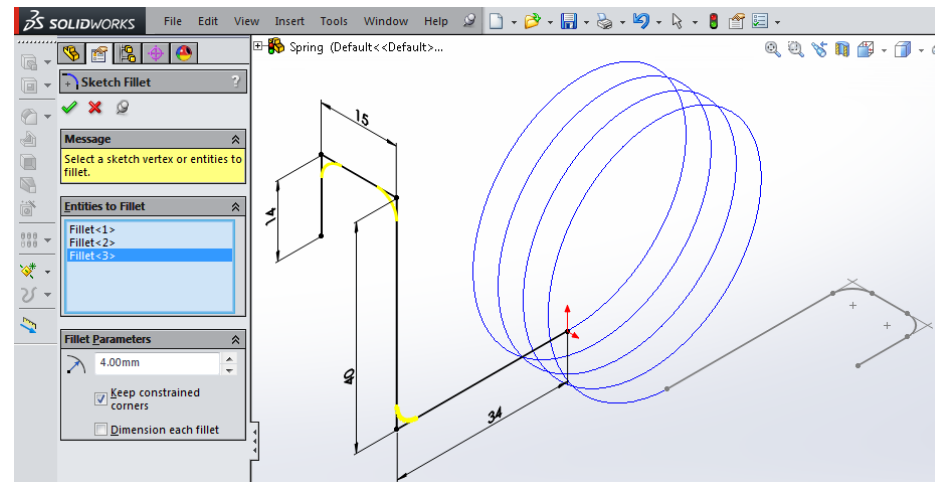
Perform the dimensioning (Fig17)!

Fig17



Fillet all of the corners of the *Line* sequence (Fig18)! Leave the sketch keep*ing* its changes!

Fig18



Now we have three individual curves. Select the *Composite Curve* tool in the *Features* toolbar to combine them (Fig19)! Select all of the three curves to join them (Fig20)! The path of the sweep is completed.

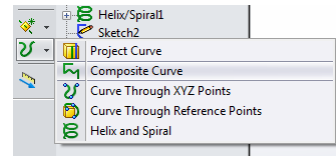


Fig19

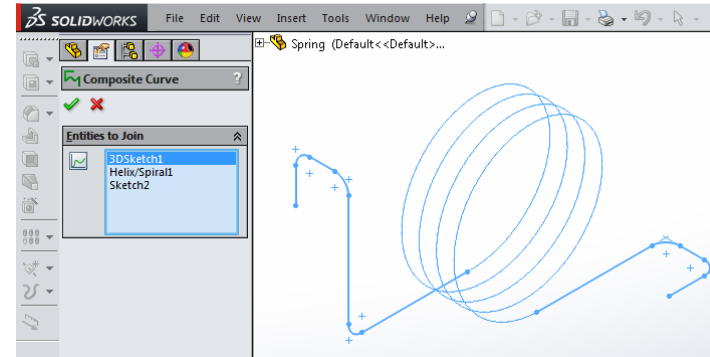


Fig20

The cross-section of the sweep shall be defined on a plane which is perpendicular to the path contains one of its endpoints. The main planes (*Top*, *Front* and *Right*) are not suitable so a new plane must be defined.

Select the *Plane* tool in the *Reference Geometry* group of the *Features* toolbar (Fig21)!

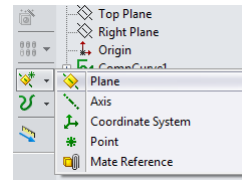


Fig21

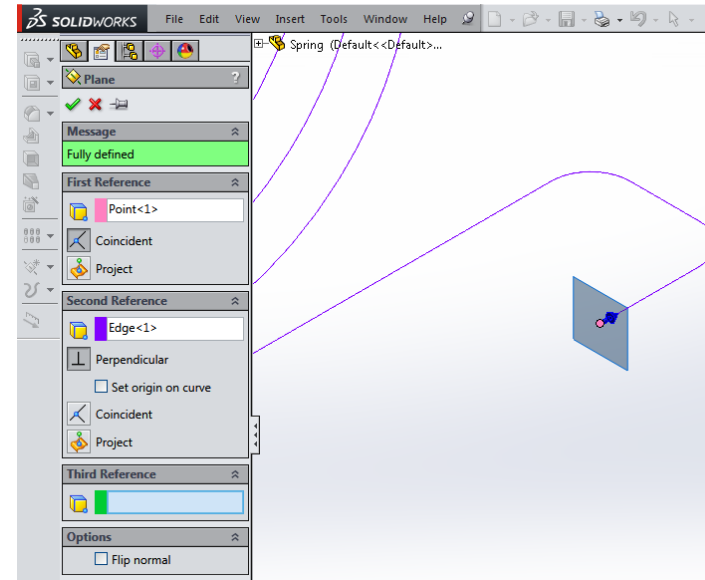


Fig22

A plane can be defined several ways. This time a point and the normal of the plane will be used. Click on the starting point of the composite curve (*First Reference*) and then on the starting segment of the curve (*Second Reference*)! These elements are enough for the definition so click on the green checkmark (*Fig22*)! A new plane (*Plane1*) has been created. Select this plane and start sketching a *Circle* on it! Unfortunately it is impossible to aim at the starting point of the path so place the center of the circle into a general position. Press Esc then select the center of the circle! Pressing *Ctrl* select the starting segment of the path and choose the *Pierce* relation (*Fig23*)!

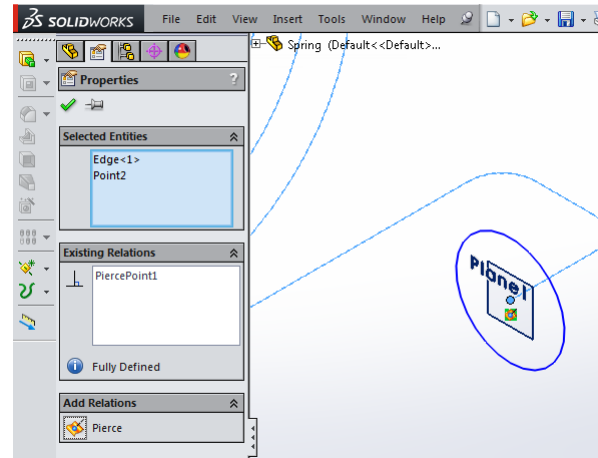


Fig23

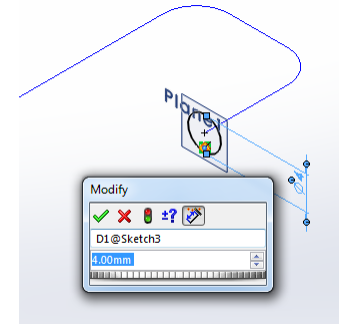


Fig24

Dimension the diameter of the circle (Fig24) and leave the sketch keeping its changes!

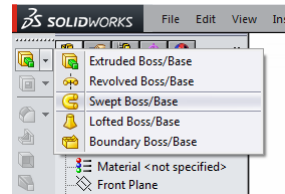


Fig25

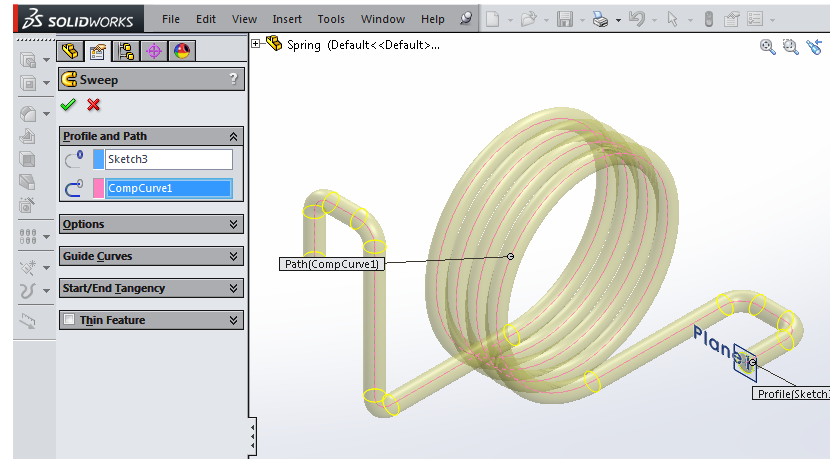


Fig26

It's time to create a swept feature so select the *Swept Boss/Base* tool in the *Features* toolbar (Fig25). Click on the circle to select it as the profile then click on the composite curve to select it as the path of the sweep (Fig26). Accept settings! The Spring is completed (Fig27)

Fig27

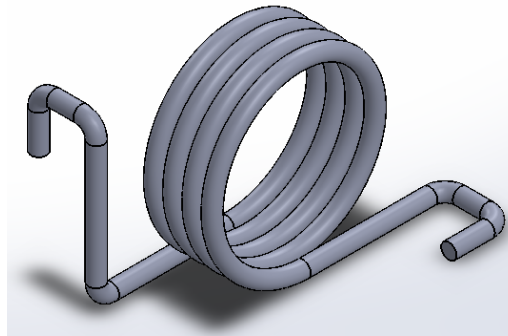


Fig27

The part document can be downloaded from the following link: [Spring.SLDPRT](#)

5. fejezet

5.1 Practices

Create the models of Fig1 – Fig15!

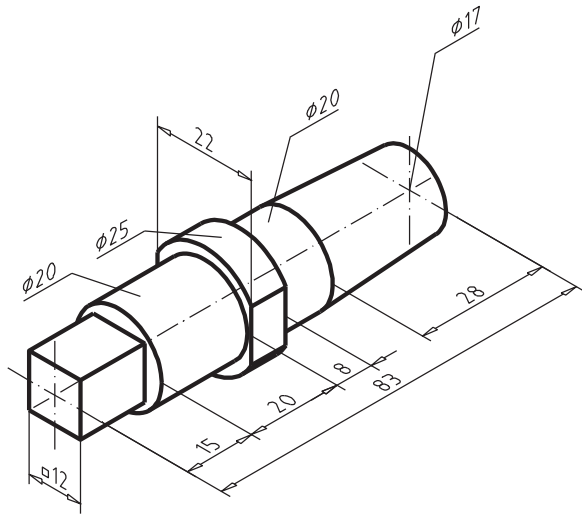


Fig1

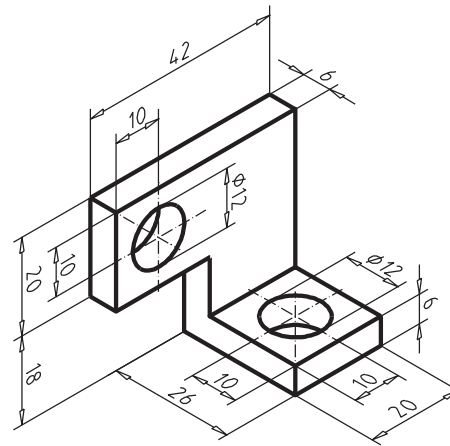


Fig2

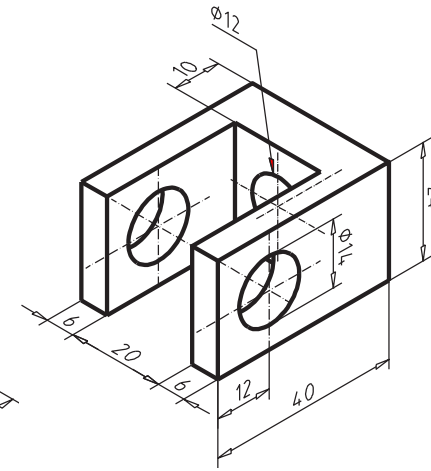


Fig3

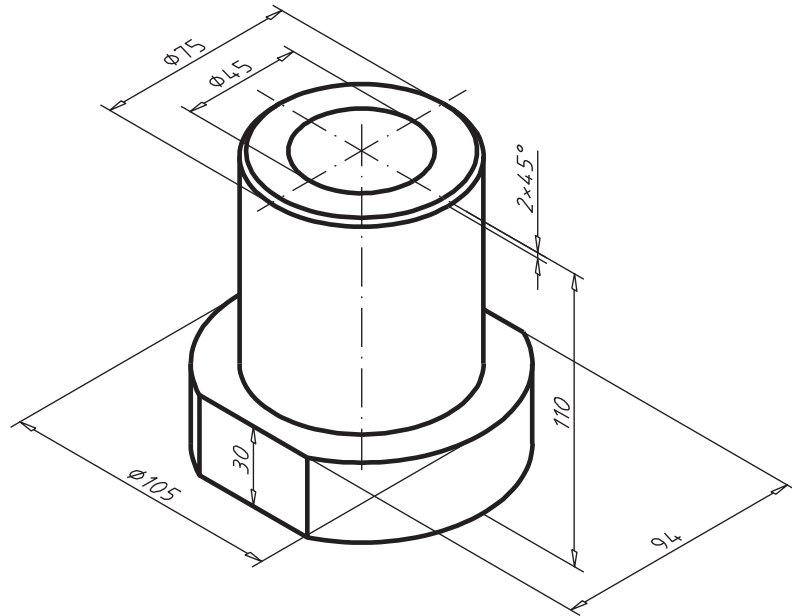


Fig6

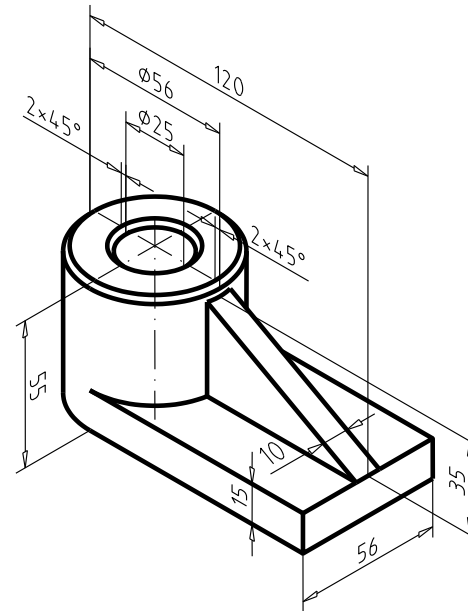


Fig7

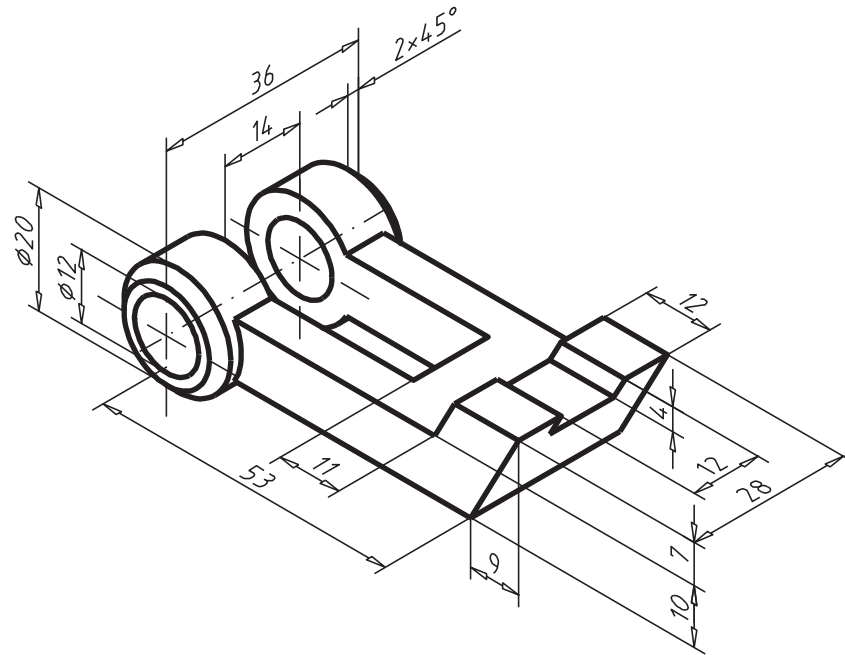


Fig8

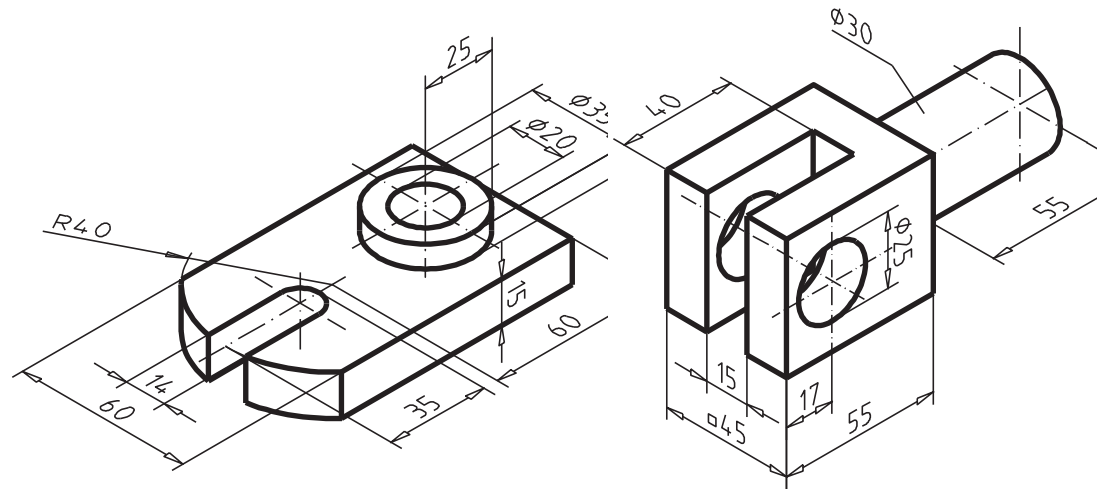


Fig9

Fig10

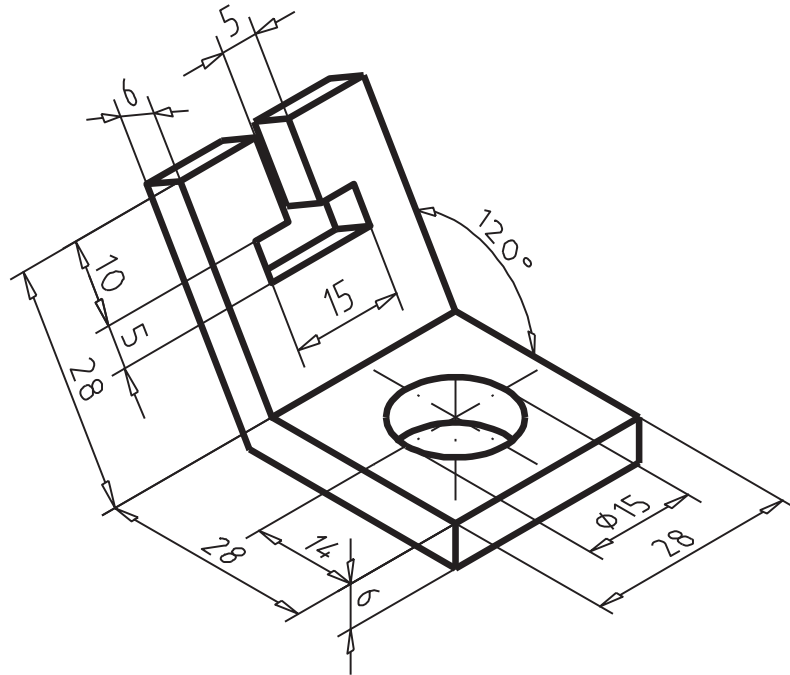


Fig11

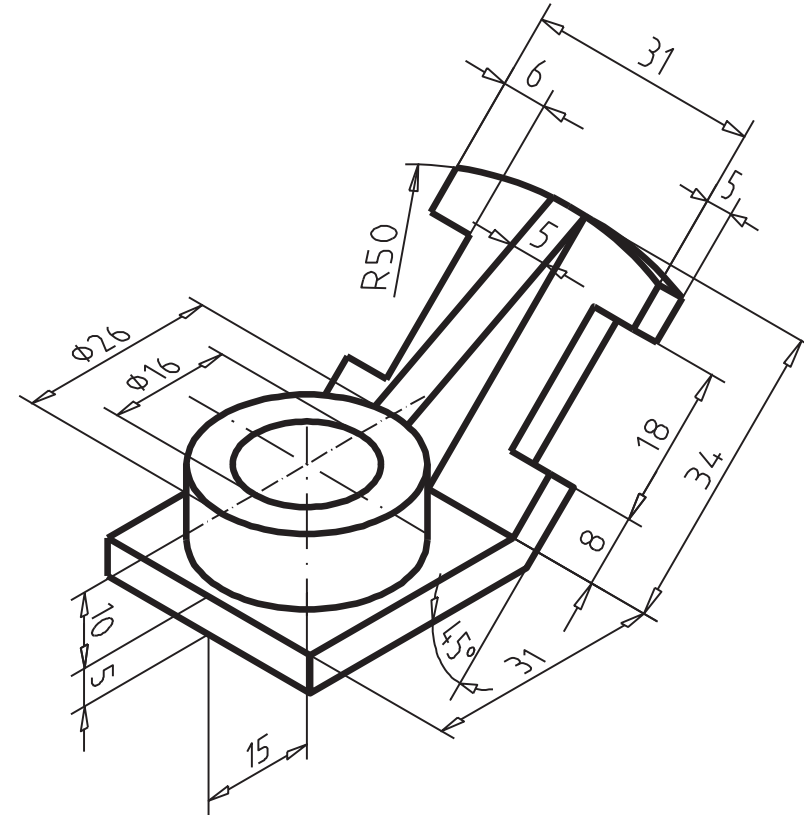


Fig12

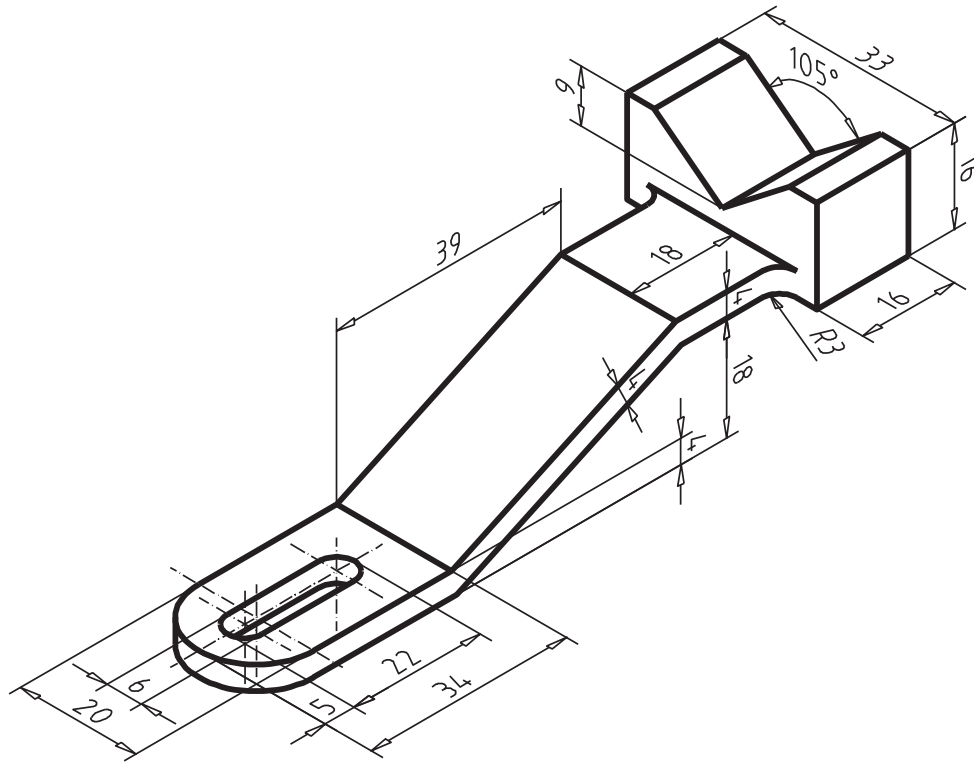


Fig13

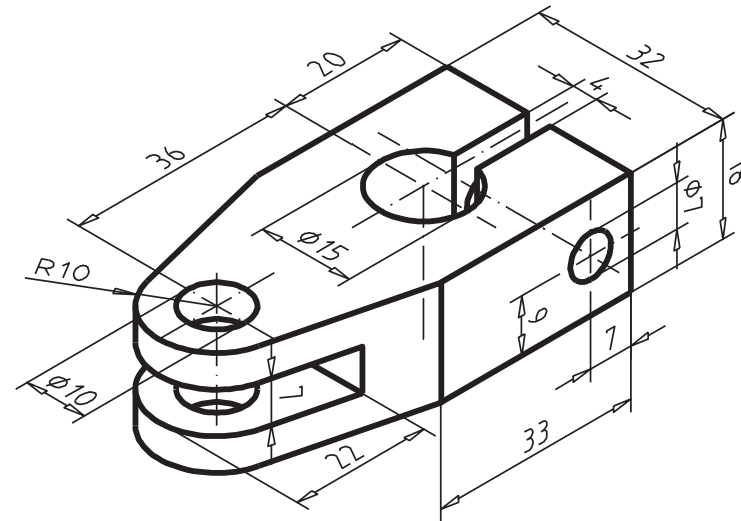
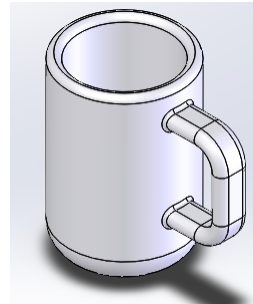


Fig14

6. fejezet

6.1 Modeling a *Mug*

Fig1



In this practice a *Mug* will be modeled (*Fig1*). The drawing of the *Mug* can be seen on **Error! Reference source not found**, and can be downloaded from the following link:

[*Mug.pdf*](#)

The dimensions of the part are not finalized. We would like to "play" with them to find the most suitable design - the model shall contain the designer's intent and it must survive the possible modifications. Predictably the outer diameter, the height and the wall thickness will be modified.

THE DESIGN INTENT

To make the cleaning of the *Mug* easy and to prevent damages during the use of the *Mug* the edges are filleted. The fillet at the upper edge must have the possible maximal radius depending on the wall thickness. The wall thickness is prescribed because it is a functional property of the product. If the wall thickness would be uneven it would cause cracking as the *Mug* getting hot - the wall thickness must be constant.

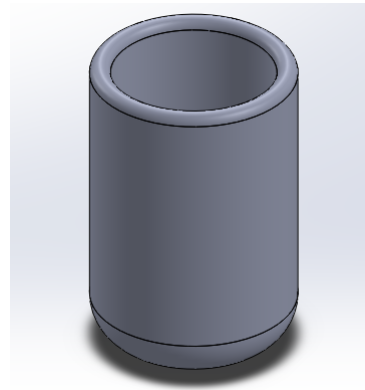
BUILDING THE MODEL

Wrong ideas

The first thought can be to create the body of the *Mug* either as a revolved feature or by extrusions then the ear can be modeled as a swept feature.

Suppose that the body was created (Fig2)!

Fig2



Where to put the sketch of the profile of the *Sweep* feature?

- The ! is not suitable because this way the Sweep feature would create unnecessary protrusions inside the body of the *Mug* (Fig3);
- It would be possible to place the sketch onto a newly defined plane which is tangential to the outer cylindrical surface of the part (Fig4). This solution is wrong because the connection between the ear and the body would be imperfect (Fig5);

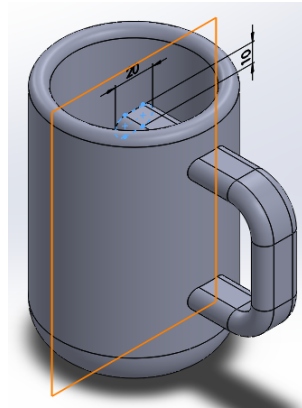


Fig3

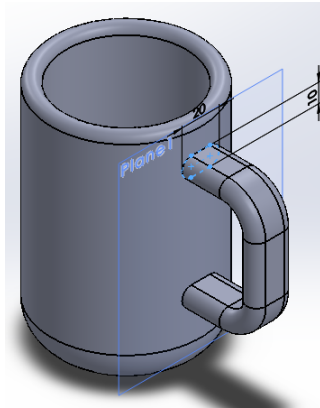


Fig4

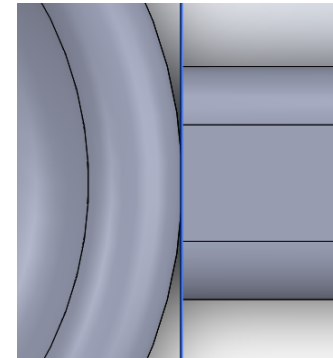


Fig5

- Other idea can be to place the sketch onto a new plane which is tangential to the *inner* cylindrical surface of the part (Fig6). This solution is wrong seems to be quite good but imagine what would happen if later the wall thickness were changed to be much smaller (Fig7).

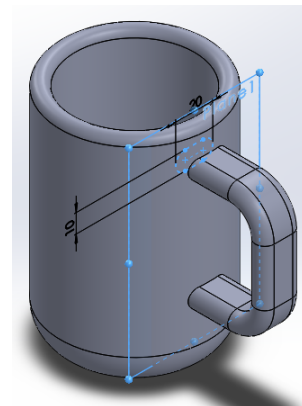


Fig6

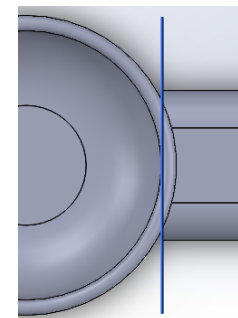


Fig7

The conclusion is that the cavity of the *Mug* must be created after the ear.

The *Right* model

Create the body of the part as a solid cylindrical protrusion! Select the *Top Plane* and sketch an origin centered *Circle* onto it! Dimension the diameter of the *Circle* and create an *Extruded Boss/Base* feature (Fig8)!

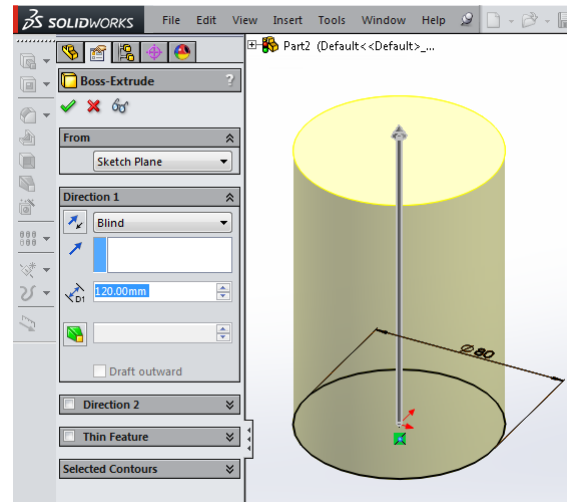


Fig8

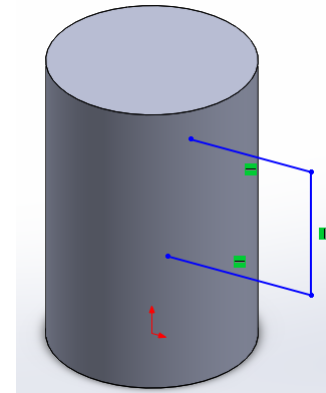


Fig9

The next feature is the ear. The path curve is defined by a Sketch which is lying in the *Front Plane*. Draw three interconnected *Line* segments according to Fig9!

The path must start and end on the *Z* axis so select the two endpoints of the *Line* sequence and the *Origin* then choose the *Vertical* relation (Fig10)!

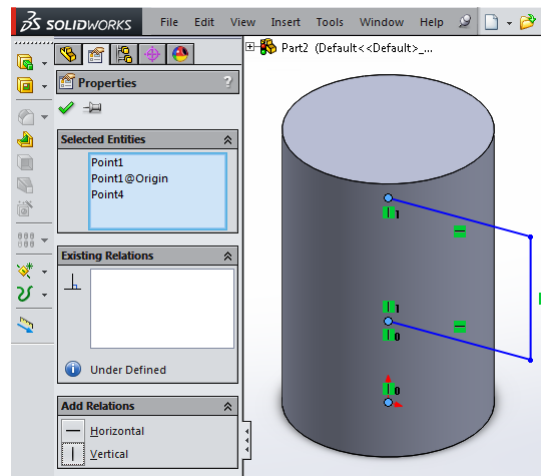


Fig10

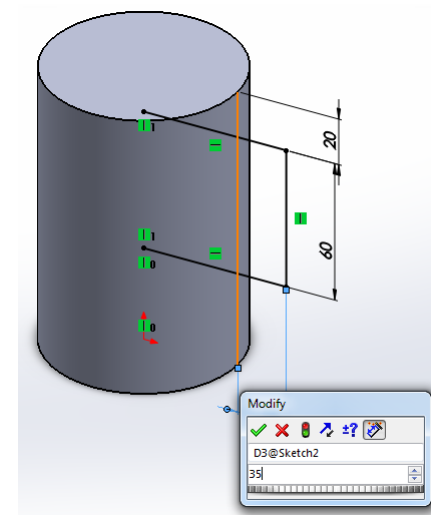


Fig11

Perform the dimensioning (Fig11)! Fillet the corners of the *Line* sequence (Fig12) and leave the sketch keep*ing* its changes!

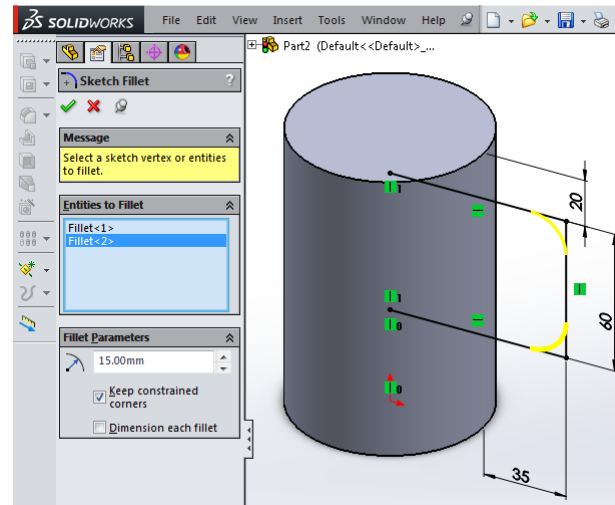


Fig12

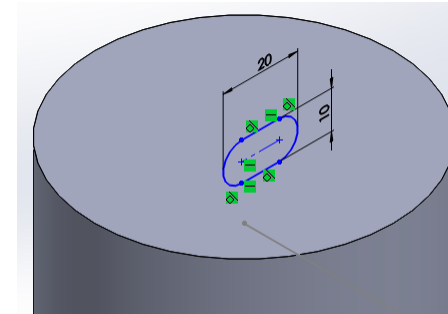


Fig13

Select the *!* and start sketching! Sketch two horizontal *Line* segments connected by tangential half circles! Connect the centers of the half circles with a *CenterLine*! Dimension the oval shape (Fig13)!

Select the *CenterLine* and the upper endpoint of the previous sketch then apply the *Midpoint* relation (Fig14)! Leave the sketch *keep*ing its changes!

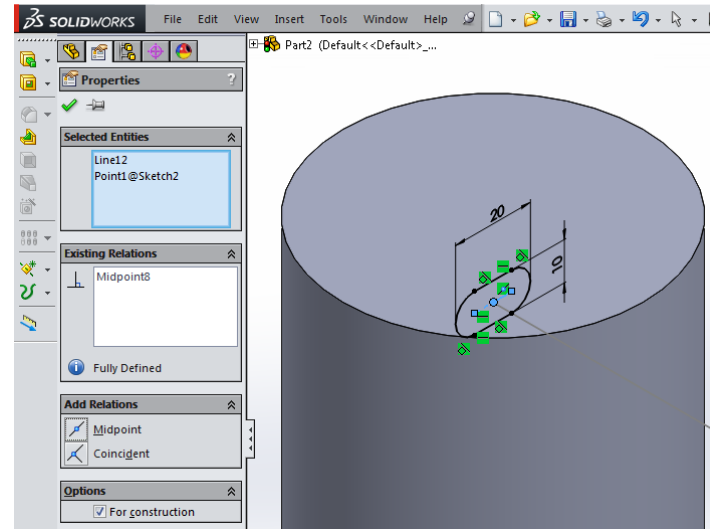


Fig14

Create a *Swept Boss/Base* feature by selecting the appropriate sketches (Fig15)!

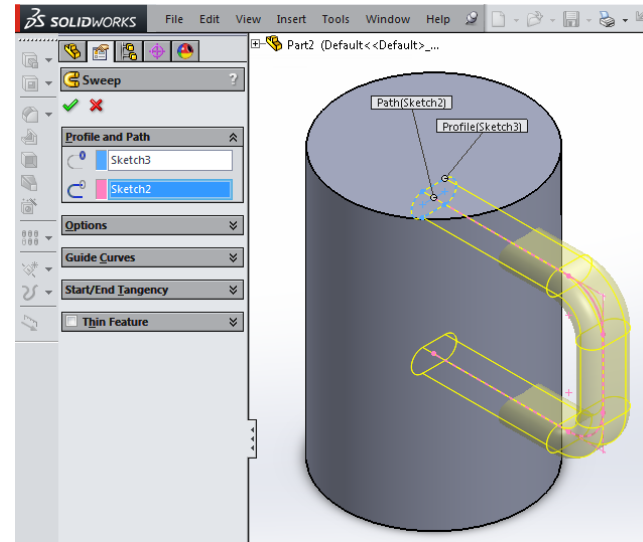


Fig15

Now the ear of the *Mug* is done so it is time to make it hollow!

The *Shell* feature is not applicable because it would make also the ear hollow - the cavity will be created by cutting.

Select the top face of the part and start sketching! Don't sketch a Circle because the wall thickness is the important information not the inner diameter. Make sure that the top circular face of the part is selected and select the Offset Entities tool in the Sketch toolbar (*Fig16*) to create an equidistant copy of the selected face's edge! Set up the proper direction of the offset and the value of it (*Fig17*) then accept settings!

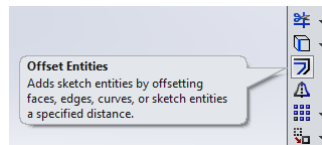


Fig16

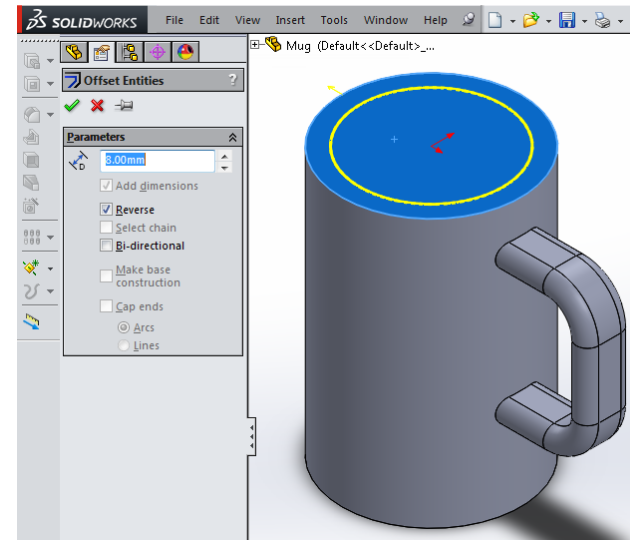


Fig17

Create an *Extruded Cut* feature! Instead of the depth of the cut we want to prescribe the thickness of the remaining material so select the *Offset From Surface* option to define the end condition of the cut (Fig18).

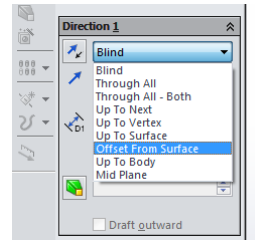
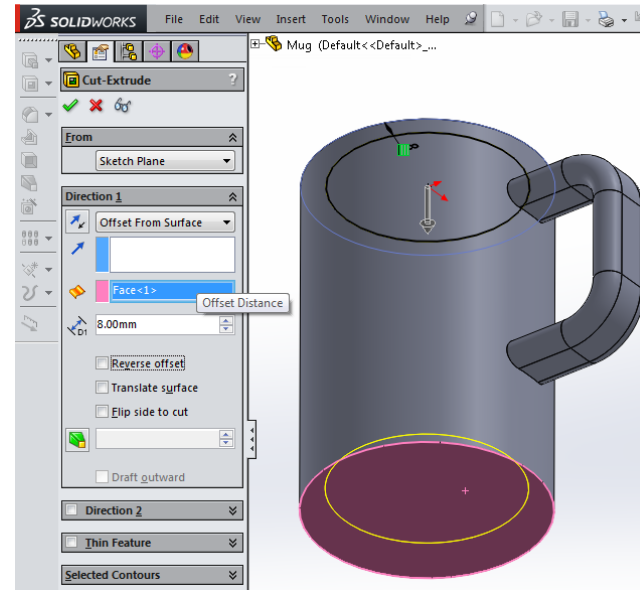


Fig18



Change the viewing direction to see the bottom face of the part the select it! Type the value of the wall thickness to the *Offset Distance* field - the cut will stop this distance away from the selected face (Fig 19). Accept settings!

LINKING DIMENSIONS

The cavity of the *Mug* was created by defining the thickness sideways and at the bottom. These values are independent yet - if one of them is changed the other remains unchanged. The solution is to link the two dimensions together.

Double-click on the inner surface of the *Mug* to display its dimensions! *Right-click* on the upper wall thickness dimension and select *Link Values* from the pop-up menu (*Fig20*)! Type a meaningful name into the *Name* field of the *Shared Values* dialog and press *OK* (*Fig21*)!

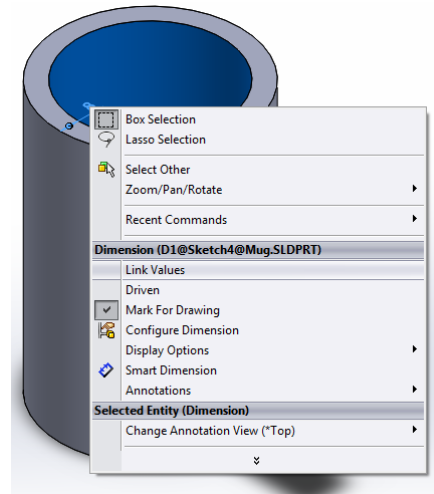


Fig20

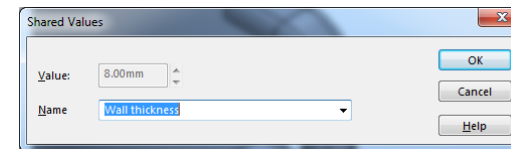


Fig21

Right-click on the bottom wall thickness dimension and select *Link Values* again! The linking is performed through sharing a common name so select the previously typed name from the drop-down list of the *Name* field (*Fig22*).

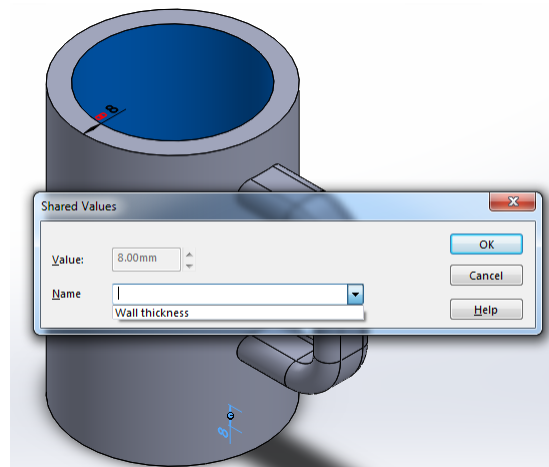


Fig22

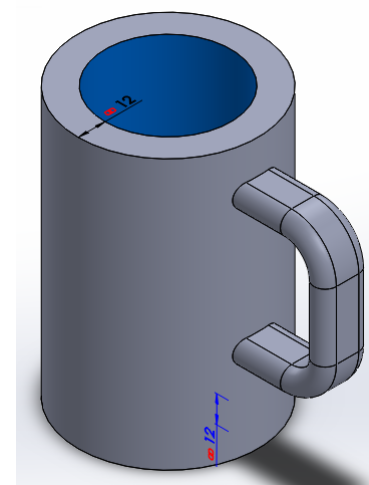


Fig23

The wall thicknesses are linked now. Try to modify one of them and ascertain that the other one changes also. Don't forget to rebuild the model. Notice the link symbol in front of the linked dimension numbers (*Fig25*).

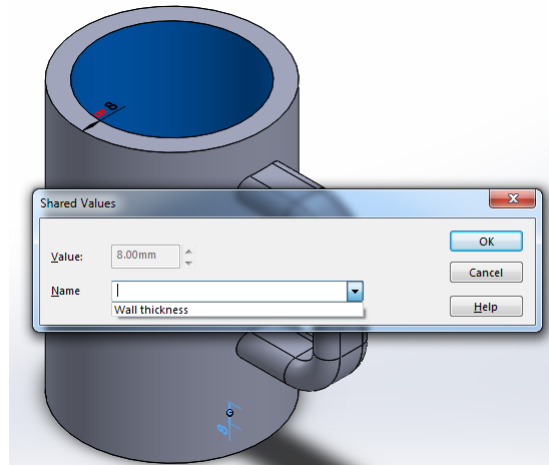


Fig24

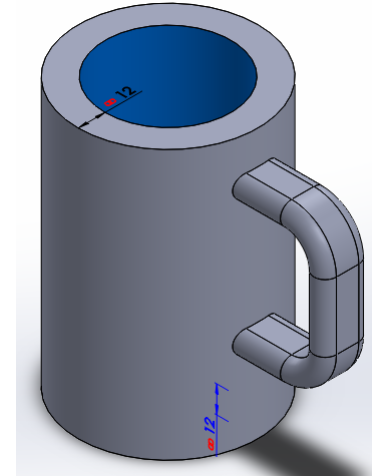


Fig25

To better see the inner details of the part select the *Section View* tool in the View (*Heads-Up*) toolbar (Fig26). Choose the *Front Plane* to use as a section plane (Fig27)!

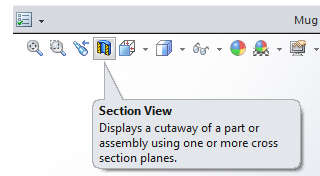


Fig26

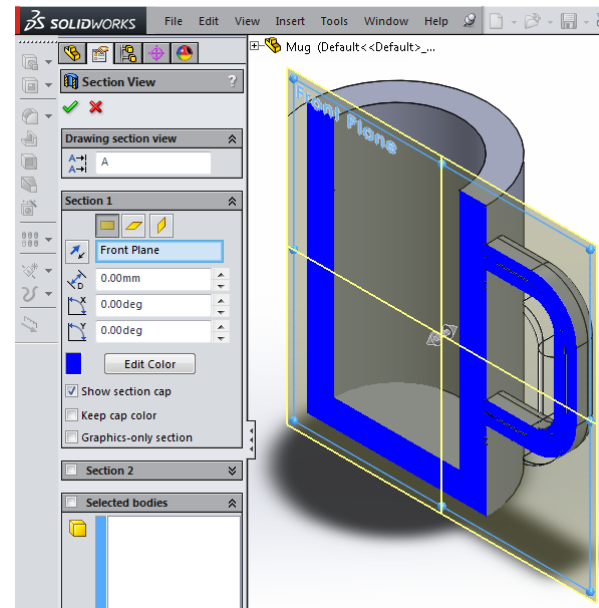


Fig27

Set the linked wall thicknesses back to their previous value!

EQUATIONS

Create the inner Fillet at the bottom of the *Mug* (Fig28)!

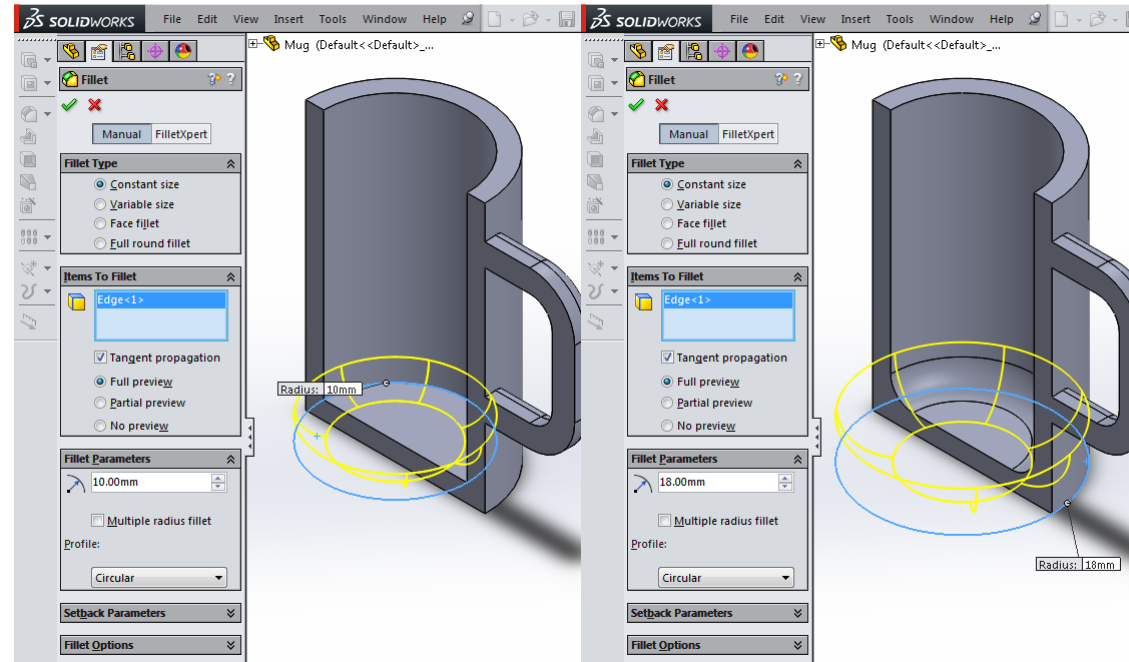


Fig28

Fig29

Create the outer Fillet at the bottom of the part (Fig29)! The value of the radius depends on the inner radius and the wall thickness.

To keep the wall thickness constant the outer radius must be recalculated every time when the inner radius or the wall thickness changes. This can be automated using an *Equation*.

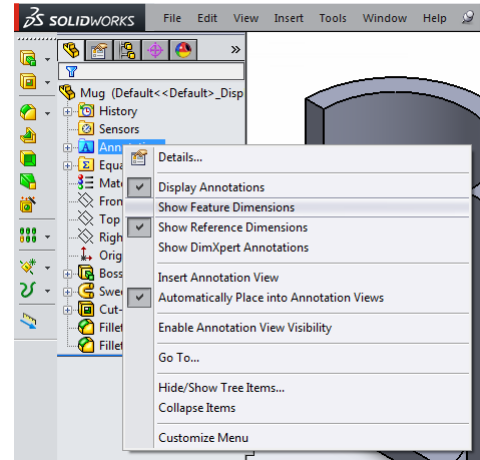


Fig30

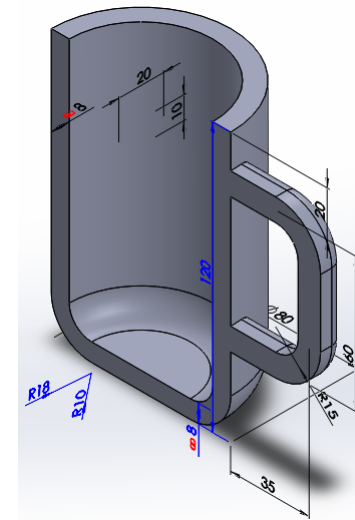


Fig31

Right-click on the *Annotations* entry of the feature list in the *Feature Manager* (Fig30) and select *Show Feature Dimensions*! Every previously defined dimension is displayed (Fig31). Select the *Tools/Equations...* entry in the Main menu! Click on the *Add Equation* entry in the Name column then select the dimension of the outer radius on the *Drawing Area*! The name of the dimension appears in the cell - this will be the result of the *Equation*.

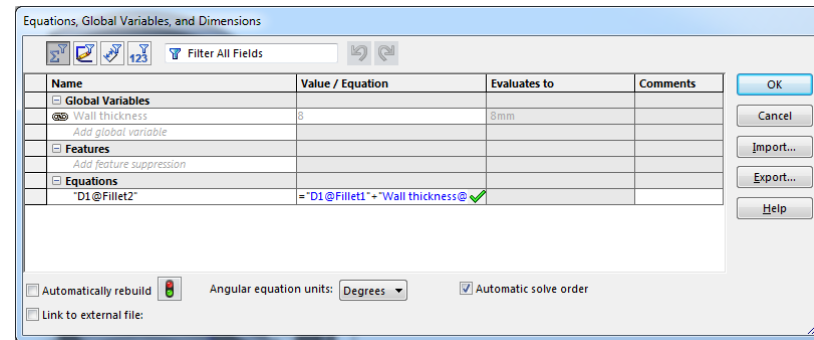


Fig32

The cursor jumps to the next cell. Select the inner radius on the *Drawing Area* type a plus sign and select the wall thickness. This way an *Equation* is defined - the outer radius is calculated as the sum of the inner radius and the wall thickness (Fig32). Press the green checkmark to accept the *Equation* then press OK!

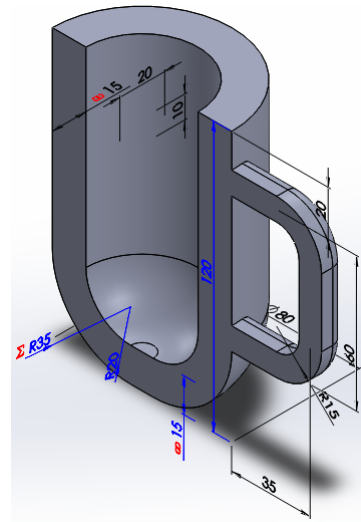


Fig33

Check the working of the *Equation*) Modify the wall thickness and the inner radius then rebuild the model! The wall thickness remains constant on every region of the part (*Fig33*). Notice that the dimension of the outer fillet is preceded by a symbol showing that this dimension is driven by an *Equation*.

The upper edges of the *Mug* are also filleted. The radius of the Fillet depends on the wall thickness – the half of it. Fillet the edges in one operation (*Fig34*)! The value of the radius is not important – choose a sufficiently small value.

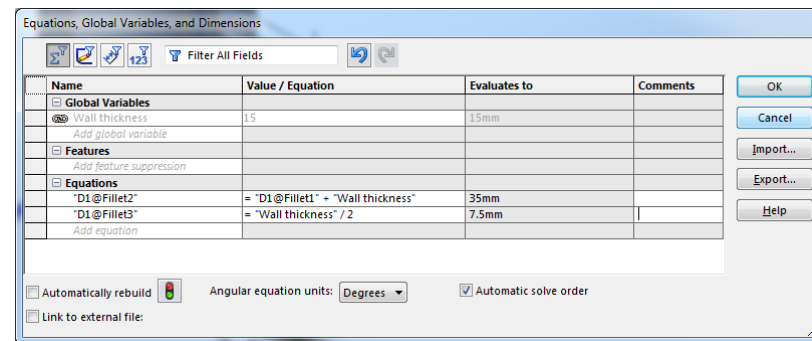


Fig35

The model refreshes and the upper fillet shows the proper shape (*Fig36*).

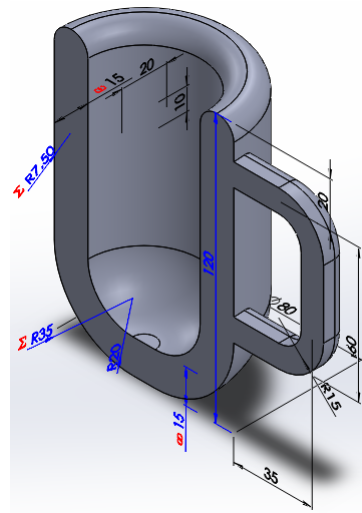


Fig36

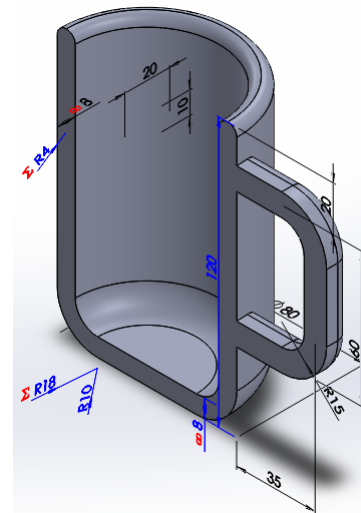


Fig37

Restore the dimension values according to the drawing on **Error! Reference source not found.** and Rebuild the model (Fig37)!

As we have seen it was possible to create a model which contains some intelligence to follow the possible modifications.

CONFIGURATIONS

Configurations in *SolidWorks* allow you to create multiple variations of a part or *Assembly* model within a single document. Configurations are a convenient way to develop and manage families of models with different dimensions, *Components*, or other parameters.

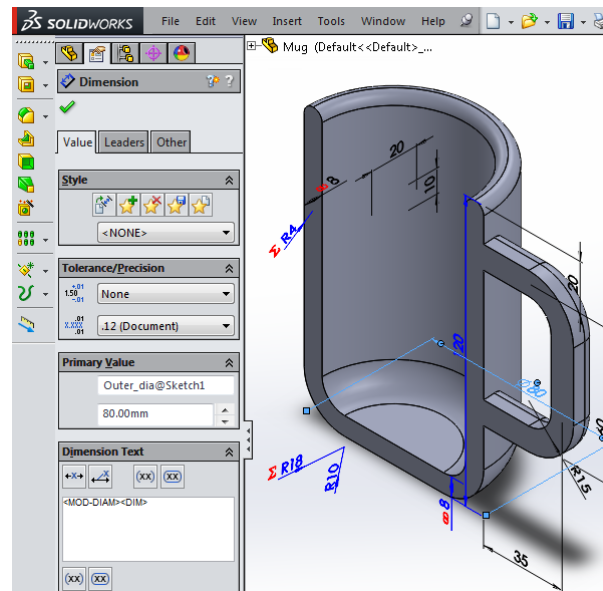
You can create configurations manually, or you can use a *Design Table* to create multiple configurations simultaneously. *Design Tables* provide a convenient way to create and manage configurations in a worksheet. You can use *Design Tables* in both part and *Assembly* documents.

In the following subchapter we will create configurations of the *Mug* using a *Design Table*.

To make the handling of the dimensions easier rename them to have more informative names.

Select the outer diameter of the *Mug* on the *Drawing Area* - the *Properties* of the dimension appear on the left side of the screen (*Fig38*).

Fig38



Overwrite the text in the *Primary Value* group to rename the dimension to *Outer_dia* – the *@Sketch1* suffix will be appended automatically.

Perform the previous step for the remaining characteristic dimensions:

- The height of the *Mug* will be renamed to *Height*;
- The name of the inner radius at the bottom will be *Inner_radius*;
- The name of the width of the ear will be *Ear_width*;
- The name of the height of the oval cross-section of the ear will be *Ear_thickness*;
- The name of the dimension for the vertical position of the ear will be *Ear_vertical*;
- The name of the height of the ear will be *Ear_height*;
- The horizontal size of the ear shall be renamed to *Ear_horizontal*;
- The fillet radius of the path of the ear will be *Ear_radius*.

Notice that the wall thicknesses have been renamed during the linking.

Check if every dimension was renamed by displaying their names. Press the *View Dimension Names* button in the *Hide/Show Items* group of the *View (Heads-Up)* toolbar (Fig39)!

DESIGN TABLE

The *Design Table* is a *Microsoft Excel* spreadsheet embedded to the *SolidWorks* document. First it is necessary to create the table. Select the *Insert/Tables/Design Table...* item of the *Main* drop-down menu. Set up the *Design Table* dialog according to *Fig41*; then click on the green checkmark!

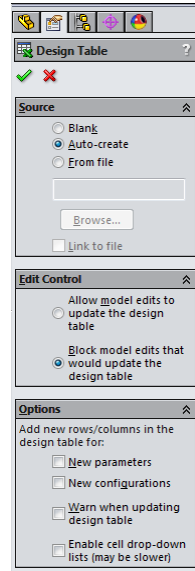


Fig41

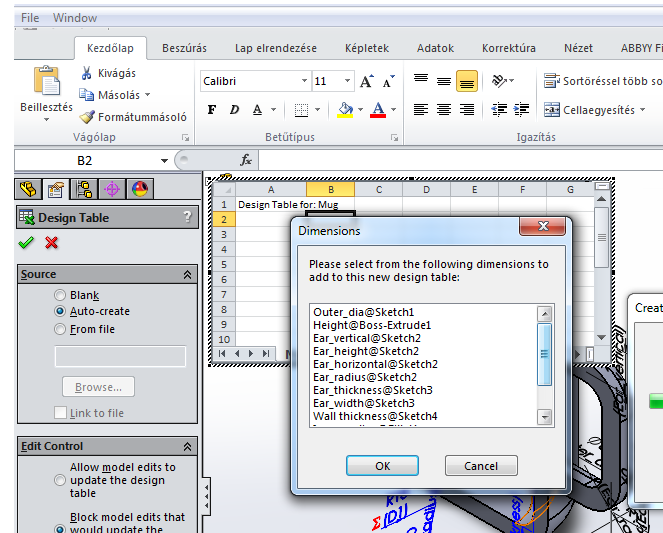


Fig42

Notice that the menu has been changed to Excel's menu and an embedded spreadsheet appeared (*Fig42*). A dialog asks which dimensions shall appear in the *Design Table* (*Fig43*). Notice that the wall thickness appeared only once in the list! Select the renamed dimensions and press *OK* (*Fig43*)!



Fig43

The A1 cell of the table contains the obligatory *Design Table* for: *Mug* text. The B2..K2 cells contain the names of the selected dimensions - these are the parameters of the configurations. The A3 cell contains the name of the only configuration of the part (*Default*) and the B3..K3 cells contain the values of the parameters (Fig44).

	A	B	C	D	E	F	G	H	I	J	K	L
1	Design Table for: Mug											
2		Outer_diam@Sketch1	Height@Boss-Extrude1	Ear_vertical@Sketch2	Ear_height@Sketch2	Ear_horizontal@Sketch2	Ear_radius@Sketch2	Ear_thickness@Sketch3	Ear_width@Sketch3	Wall_thickness@Sketch4	Inner_radius@Fillet1	
3	Default	80	120	20	60	35	15	10	20	8	10	
4												
5												
6												
7												
8												
9												
10												
11												
12												
13												
14												

Fig44

Fill the table according to Fig45! Click outside the table to leave Excel!

	A	B	C	D	E	F	G	H	I	J	K	L
1	Design Table for: Mug											
2		Outer_dia@Sketch1	Height@Boss-Extrude1	Ear_vertical@Sketch2	Ear_height@Sketch2	Ear_horizontal@Sketch2	Ear_radius@Sketch2	Ear_thickness@Sketch3	Ear_width@Sketch3	Wall thickness@Sketch4	Inner_radius@Fillet1	
3	Cocoa	80	120	20	60	35	15	10	20	8	10	
4	Tea	70	70	15	40	25	12	8	15	6	8	
5	Coffee	60	60	10	30	20	10	6	12	5	6	
6	Beer	100	120	25	80	50	20	12	20	10	15	
7												
8												
9												
10												
11												
12												
13												
14												

Fig45

A message box informs us that the *Design Table* generated four new configurations of the part (Fig46).

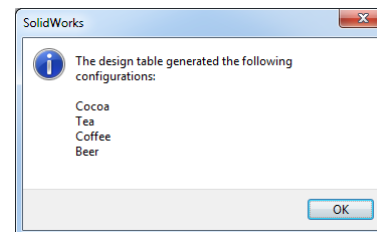


Fig46

The colour of the dimensions what are included in the *Design Table* changed to magenta marking that these values can be modified in the *Design Table* only.

Select the *Configuration Manager* tab on the left side of the screen to see the configurations of the *Mug* (Fig47). The configurations are listed here. As it can be seen the active configuration is the *Default* one.

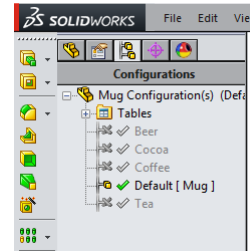


Fig47

Double-click on the configuration's name what you want to see. The *Cocoa* configuration is identical to the *Default* configuration. The *Tea* configuration can be seen on Fig48. The *Coffee* configuration appears on Fig49.

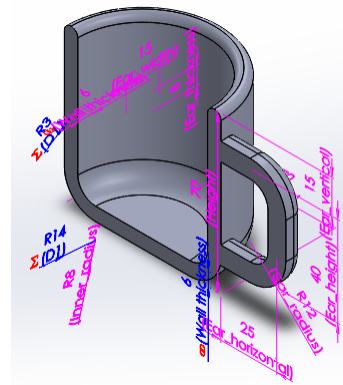


Fig48

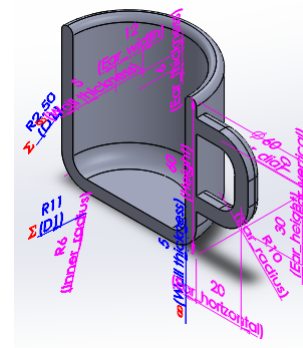


Fig49

The ear of the Beer configuration looks a bit strange (Fig50). Let's go back to the *Design Table* to modify the corresponding parameters. Expand the Tables item and *Right-click* on *Design Table* (Fig51)!

There are two possibilities to edit the table:

- In an embedded window as it was done previously (*Edit Table*);
- In a separate window (*Edit Table in New Window*).

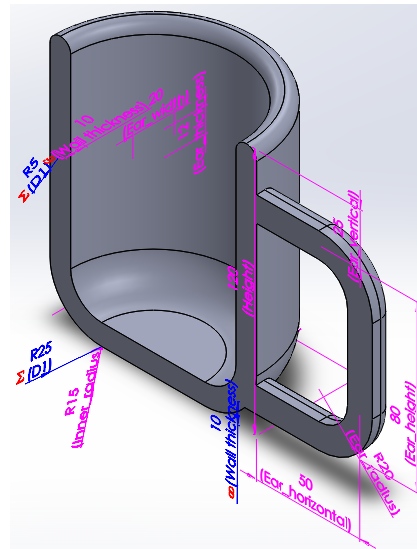


Fig50

Maybe it is more comfortable to edit the *Design Table* in a separate window.

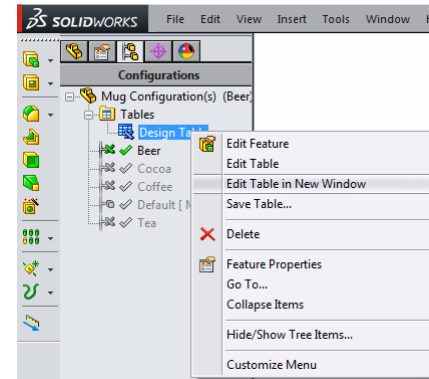


Fig51

Fig52

A 3D model of a mug with the following dimensions and coordinate systems:

- Dimensions:**
 - Top horizontal: 20 (For width)
 - Top vertical: 10 (For thickness)
 - Inner radius: R15
 - Bottom horizontal: 10 (For thickness)
 - Bottom vertical: 45 (For horizontal)
 - Handle vertical: 65 (For height)
 - Handle horizontal: 45 (For horizontal)
 - Handle radius: R20 (For radius)
- Coordinate Systems:**
 - RS Z(D1):** Located at the top left of the mug.
 - R25 Z(D1):** Located at the bottom left of the mug.

192

SUPPRESSING FEATURES

Create the Fillet at the connection of the ear and the body (*Fig54*)! Make a different configuration active! Notice that the Fillet is not present - to be more precise it is present but suppressed (*Fig55*).

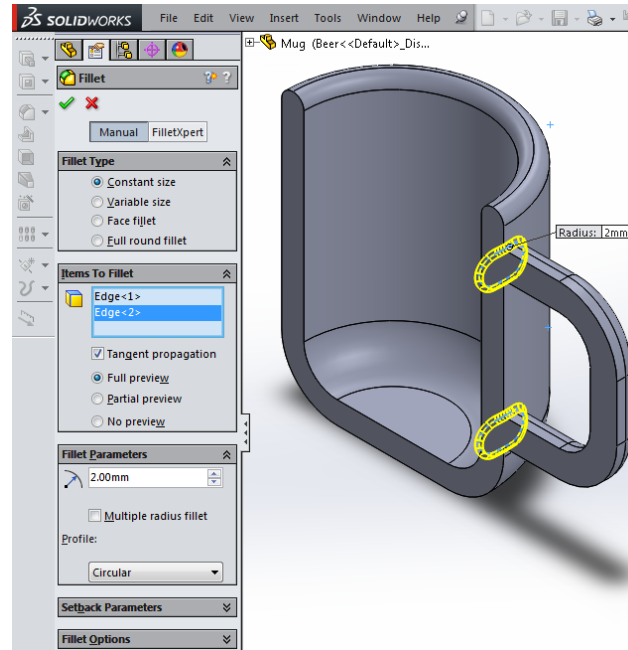


Fig54

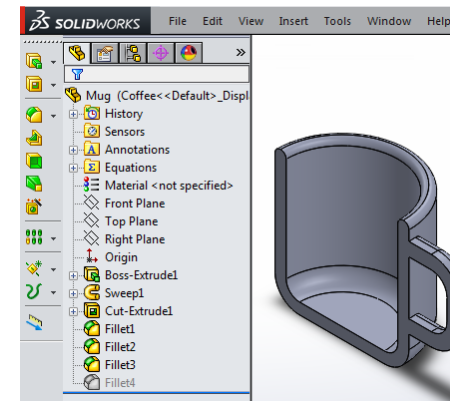


Fig55

Right-click on the last feature (*Fillet4*) and select the *Configure Feature* item from the pop-up menu (*Fig56*)! As the appearing dialog window shows; the feature is suppressed in every configuration except in which it was created (*Fig57*). Uncheck every checkbox and press OK.

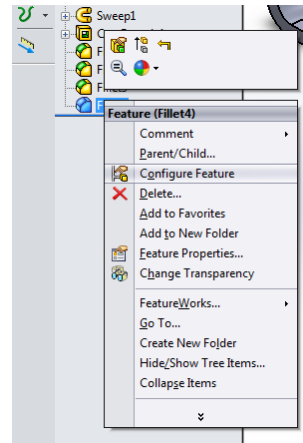


Fig56

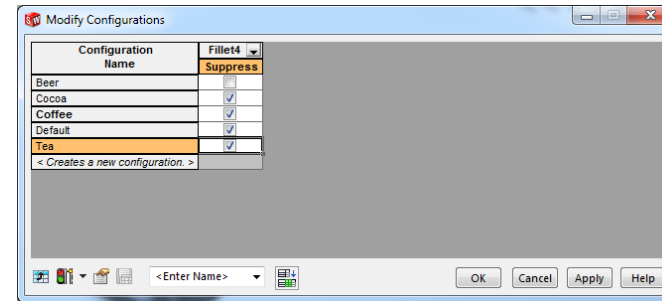


Fig57

Expand the product family of the *Mug* with a children's version. This configuration will be identical to the *Default* one except it will have two ears! Make the *Default* configuration active! Mirror the Sweep1 and the Fillet4 feature to the ! (Fig58)!

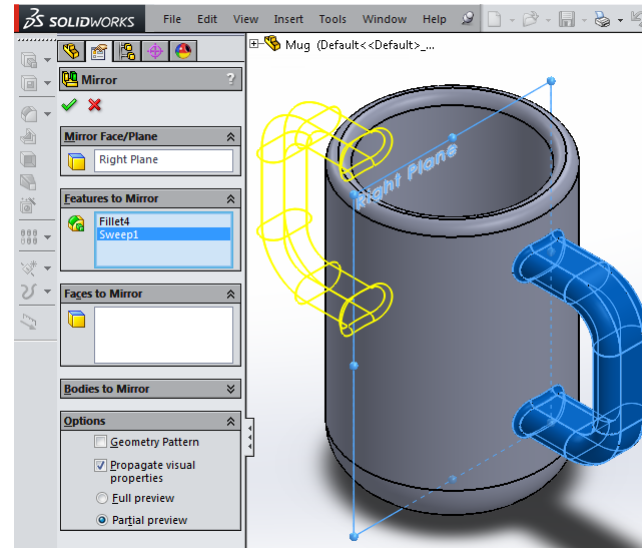


Fig58

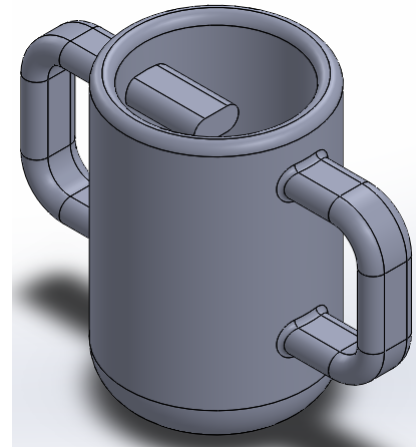


Fig59

The result is wrong (Fig59). To solve the problem modify the order of the features. The *Fillet4* and the *Mirror1* features must be between the *Sweep1* and the *Cut-Extrude1* features (Fig60).

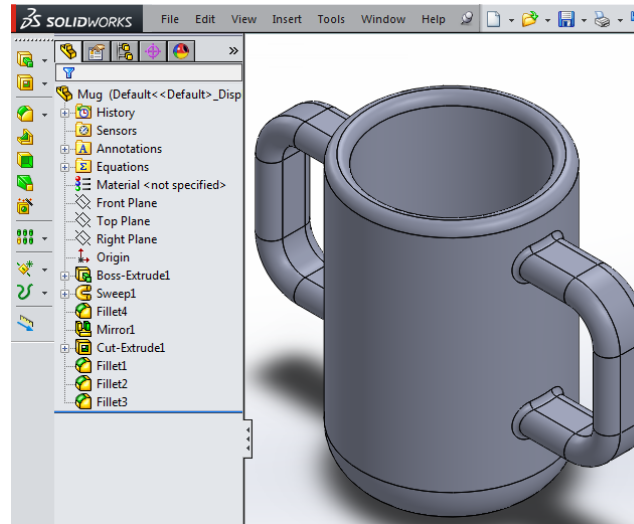


Fig60

Edit the *Design Table* in a separate window to control the status of the mirrored ear! As the table opens the active cell is the first empty cell in the row of the parameters (L2). Use *Alt-Tab* to switch to the *SolidWorks* window and double-click on the surface of the mirrored ear. Switch to the *Excel* window! As you can see a new parameter was inserted into the *Design Table* – the status of the *Mirror1* feature (Fig61).

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Design Table for: Mug												
2		Outer_dia@Sketch1	Height@Boss-Extrude1	Ear_vertical@Sketch2	Ear_height@Sketch2	Ear_horizontal@Sketch2	Ear_radius@Sketch2	Ear_thickness@Sketch3	Ear_width@Sketch3	Wall_thickness@Sketch4	Inner_radius@Fillet1		
3	Cocoa	80	120	20	60	35	15	10	20	8	10	UNSUPPRESSED	
4	Tea	70	70	15	40	25	12	8	15	6	8		
5	Coffee	60	60	10	30	20	10	6	12	5	6		
6	Beer	100	120	20	65	45	20	12	20	10	15		
7													

Fig61

Fill a new row in the table according to Fig62! The dimensions of the *Children's* configuration can be copied from the *Cocoa* row. Complete the column for the status of the mirrored ear – it will be suppressed in every configuration except in the *Children's* one.

	A	B	C	D	E	F	G	H	I	J	K	L
1	Design Table for: Mug											
2		Outer_dia@Sketch1	Height@Boss-Extrude1	Ear_vertical@Sketch2	Ear_height@Sketch2	Ear_horizontal@Sketch2	Ear_radius@Sketch2	Ear_thickness@Sketch3	Ear_width@Sketch3	Wall_thickness@Sketch4	Inner_radius@Fillet1	\$STATE@Mirror1
3	Cocoa	80	120	20	60	35	15	10	20	8	10	SUPPRESSED
4	Tea	70	70	15	40	25	12	8	15	6	8	SUPPRESSED
5	Coffee	60	60	10	30	20	10	6	12	5	6	SUPPRESSED
6	Beer	100	120	20	65	45	20	12	20	10	15	SUPPRESSED
7	Children's	80	120	20	60	35	15	10	20	8	10	UNSUPPRESSED
8												

Fig62

Close Excel to return to the *Mug*! Check if the modification is correctly performed on every configuration.

Finally create a member of the part family for short drinks! It will be a small one without any ears.

Edit the *Design Table* in a separate window! Switch to the *SolidWorks* window and double-click on the surface of the original ear to insert a new parameter to the *Design Table* for the status of the Sweep1 feature – it will be unsuppressed in every older configuration. Create a new row in the table for the Short drink configuration according to Fig63! The cells belonging to the parameters of the ear can be left empty because the Sweep1 feature is suppressed.

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Design Table for: Mug												
2		Outer_dia@Sketch1	Height@Boss-Extrude1	Ear_vertical@Sketch2	Ear_height@Sketch2	Ear_horizontal@Sketch2	Ear_radius@Sketch2	Ear_thickness@Sketch3	Ear_width@Sketch3	Wall_thickness@Sketch4	Inner_radius@Fillet1	SS1@Mirror1	SS1@Sweep1
3	Cocoa	80	120	20	60	35	15	10	20	8	10	SUPPRESSED	UNSUPPRESSED
4	Tea	70	70	15	40	25	12	8	15	6	8	SUPPRESSED	UNSUPPRESSED
5	Coffee	60	60	10	30	20	10	6	12	5	6	SUPPRESSED	UNSUPPRESSED
6	Beer	100	120	20	65	45	20	12	20	10	15	SUPPRESSED	UNSUPPRESSED
7	Children's	80	120	20	60	35	15	10	20	8	10	UNSUPPRESSED	UNSUPPRESSED
8	Short drink	35	40							3	8	SUPPRESSED	SUPPRESSED

Fig63

Close *Excel* to return to the *Mug*! A message informs us that a new configuration was created. Make the *Short drink* configuration active to see its design (Fig64)!

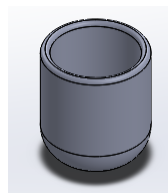


Fig64

Notice that not only the *Sweep1* and the *Mirror1* features are suppressed but also *Fillet4* because it is dependent of *Sweep1*.

The modeling process of the part family is completed.

The part document can be downloaded from the following link: [Mug.SLDPRT](#)

6.2 Practices

CLEVIS PIN WITH HEAD, ISO 2341:1986

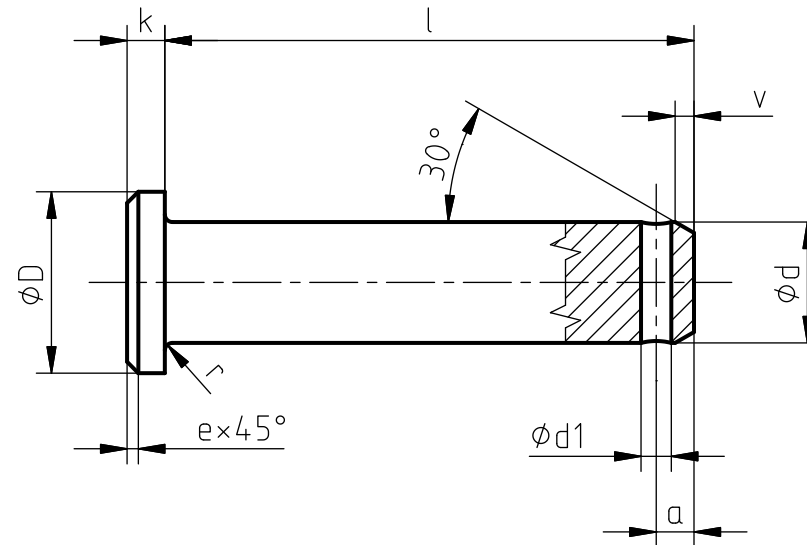


Fig1

Create the configurations of the part on Fig1. The dimensions of the part are in the table below:

d	D	k	e	r	v	d₁	a
5	10	3	0,5	0,5	1	1,2	2
6	11	3	0,5	0,5	1,5	1,6	2,2
8	14	4	1	0,5	2	2	2,5
10	16	4	1	0,5	2	2,5	3
12	20	5	1,5	1	2,5	3,2	4
14	22	5	1,5	1	2,5	3,2	4
16	24	5	1,5	1	2,5	4	5
18	26	5	1,5	1	3	4	5
20	30	6	2	1	3	4	5
25	35	7	2	1	4	5	6,5
28	38	7	2	1	4	5	6,5
30	40	7	2	1	4	6,3	8

The possible l - d pairs can be found in the table of the next page.

l	d											
	5	6	8	10	12	14	16	18	20	25	28	30
10	+	+										
12	+	+										
14	+	+	+									
16	+	+	+									
18	+	+	+	+								
20	+	+	+	+								
22	+	+	+	+	+	+	+					
25	+	+	+	+	+	+	+					
28	+	+	+	+	+	+	+					
30	+	+	+	+	+	+	+					
35	+	+	+	+	+	+	+	+	+			
40	+	+	+	+	+	+	+	+	+	+		
45	+	+	+	+	+	+	+	+	+	+		
50	+	+	+	+	+	+	+	+	+	+		
55			+	+	+	+	+	+	+	+	+	+
60			+	+	+	+	+	+	+	+	+	+
65			+	+	+	+	+	+	+	+	+	+
70				+	+	+	+	+	+	+	+	+
75				+	+	+	+	+	+	+	+	+
80				+	+	+	+	+	+	+	+	+
85					+	+	+	+	+	+	+	+
90					+	+	+	+	+	+	+	+
95					+	+	+	+	+	+	+	+
100					+	+	+	+	+	+	+	+
105										+	+	+
110										+	+	+
115										+	+	+
120										+	+	+
125											+	+
130											+	+
135											+	+
140											+	+
145											+	+
150											+	+
155											+	+
160											+	+
165											+	+
170											+	+

HEXAGON CASTLE NUT, MSZ 2264:1987

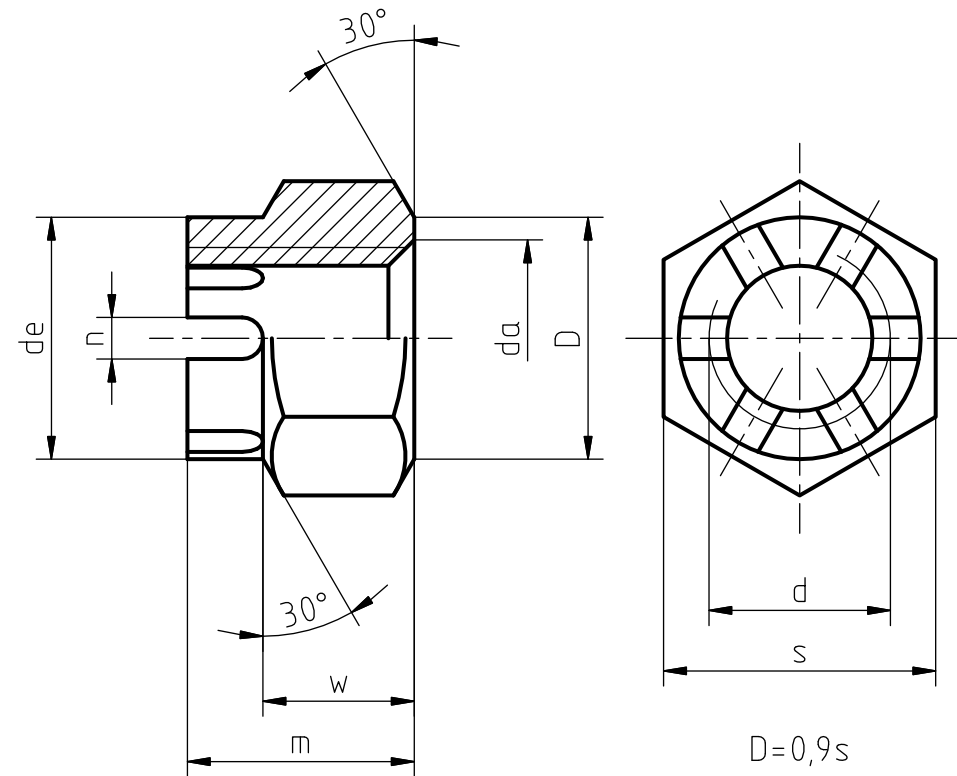


Fig2

Create the configurations of the part on Fig2. The dimensions of the part are in the table below:

d	d_a	d_e	s	m	n	w
M12	12	16	18	15	3,5	10
M14	14	18	21	16	3,5	11
M16	16	22	24	19	4,5	13
M18	18	25	27	21	4,5	15
M20	20	28	30	22	4,5	16
M22	22	32	34	26	4,5	18
M24	24	34	36	27	5,5	19
M27	27	38	41	30	5,5	22
M30	30	42	46	33	7	24
M33	33	46	50	35	7	26
M36	36	50	55	38	7	29
M39	39	55	60	40	7	31
M42	42	58	65	46	9	34
M45	45	62	70	48	9	36
M48	48	65	75	50	9	38

7. fejezet

HEXAGON CASTLE NUT MSZ 2264:1987

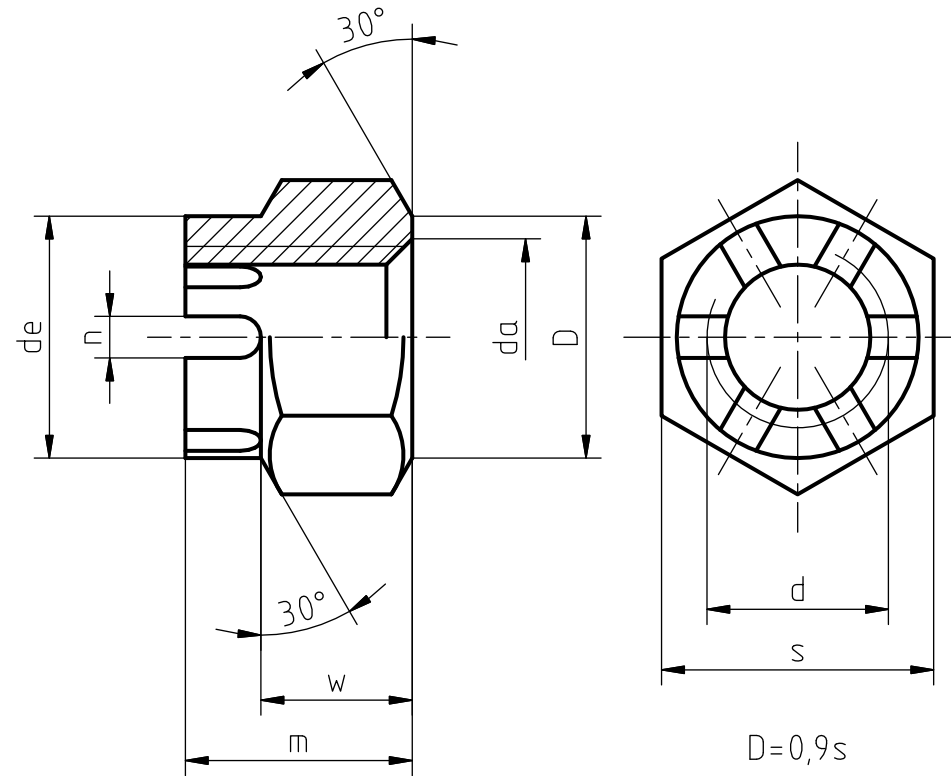


Fig1

Create the configurations of the part on Fig1. The dimensions of the part are in the table below:

d	d_a	d_e	s	m	n	w
M12	12	16	18	15	3,5	10
M14	14	18	21	16	3,5	11
M16	16	22	24	19	4,5	13
M18	18	25	27	21	4,5	15
M20	20	28	30	22	4,5	16
M22	22	32	34	26	4,5	18
M24	24	34	36	27	5,5	19
M27	27	38	41	30	5,5	22
M30	30	42	46	33	7	24
M33	33	46	50	35	7	26
M36	36	50	55	38	7	29
M39	39	55	60	40	7	31
M42	42	58	65	46	9	34
M45	45	62	70	48	9	36
M48	48	65	75	50	9	38

8. fejezet

8.1 Assemblies

An *Assembly* is a document in which parts, features, and other assemblies (subassemblies) are mated together. The parts and subassemblies are the *Components* of the *Assembly*. *Components* exist in documents separate from the *Assembly*. For example, in an *Assembly*, a piston can be mated to other parts, such as a connecting rod or cylinder. This new *Assembly* can then be used as a sub*Assembly* in an *Assembly* of an engine. The extension for a *SolidWorks Assembly* file name is *.SLDASM*.

CREATING THE ASSEMBLY OF THE RATCHET MECHANISM

In the following subchapters the *Assembly* of a ratchet mechanism will be created from the previously modeled parts. The parts can be downloaded from the following link as a compressed archive: Ratchet mechanism parts.zip

Create a new document. Select the *Assembly* type then press *OK*(Fig1)!

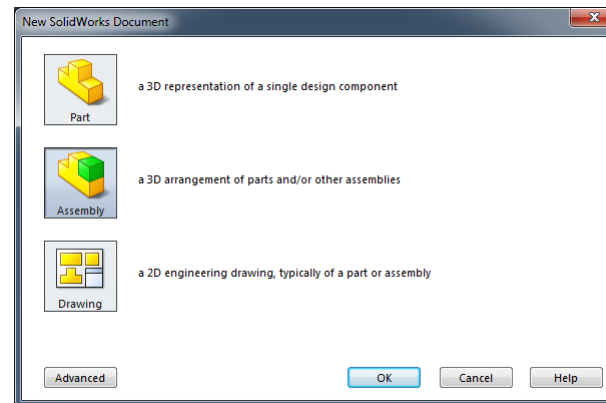


Fig1

An empty *Assembly* document window appears and a command starts to run. Press *Cancel* to take a look on the user surface (Fig2). The user surface is almost the same as it was in the case of a *Part* document. We can find the *Drawing Area*, the Sketch toolbar, the View (*Heads-Up*) toolbar, the *Status Line*, the *Feature Manager* with the main planes and the origin etc.

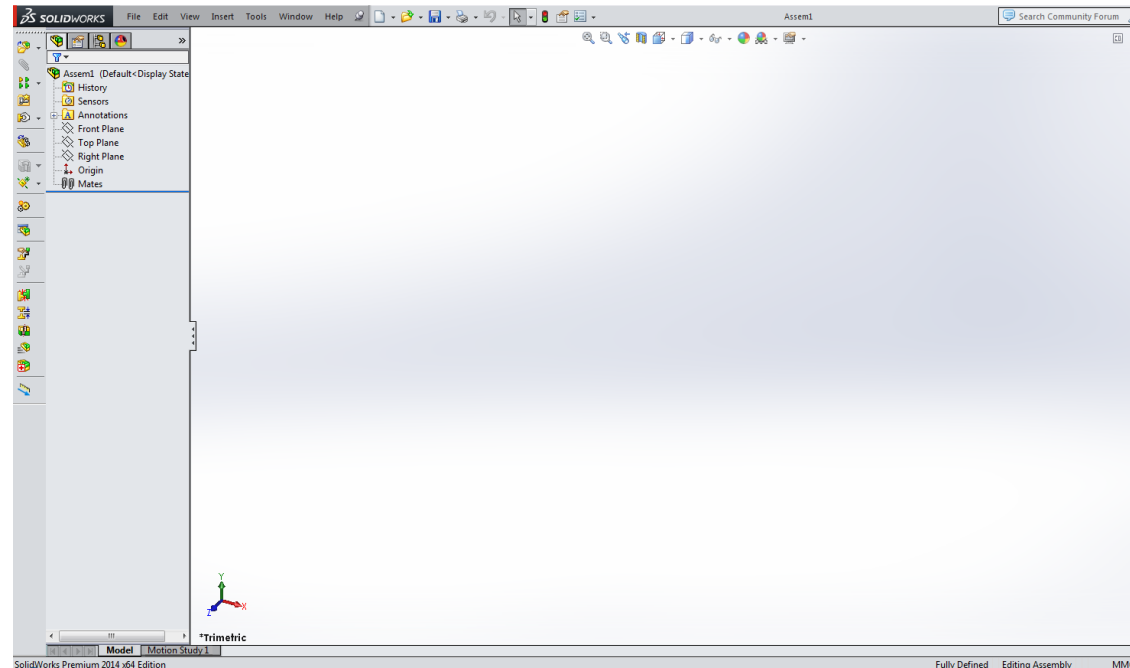


Fig2

The only difference is that on the left side of the screen the *Assembly* toolbar can be found.

INSERTING THE FIRST COMPONENT

Select the first tool (*Insert Components*) in the *Assembly* toolbar (*Fig3*) to insert *Components* into the *Assembly*!

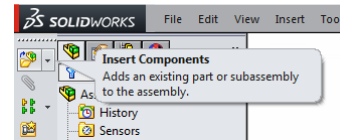


Fig3

As you can realize this was the command what was canceled previously. We have to select the first component of the *Assembly* to which the other parts or subassemblies will be connected. In the case of the ratchet mechanism this component will be the *Housing*. If the part document of the *Housing* is opened in other window then it can be selected in the list of the open documents otherwise it is necessary to *Browse* for it. After the *Housing* was opened it starts to hover over the *Drawing Area*. Do not click in the *Drawing Area* just simply click *OK* in the *Insert Components* dialog to place the *Housing* (*Fig4*). The *Housing* is inserted into the *Assembly* – its *Origin* is *Coincident* with the *Origin* of the *Assembly*. Also the main planes of the part are *Coincident* with the corresponding planes of the *Assembly*.

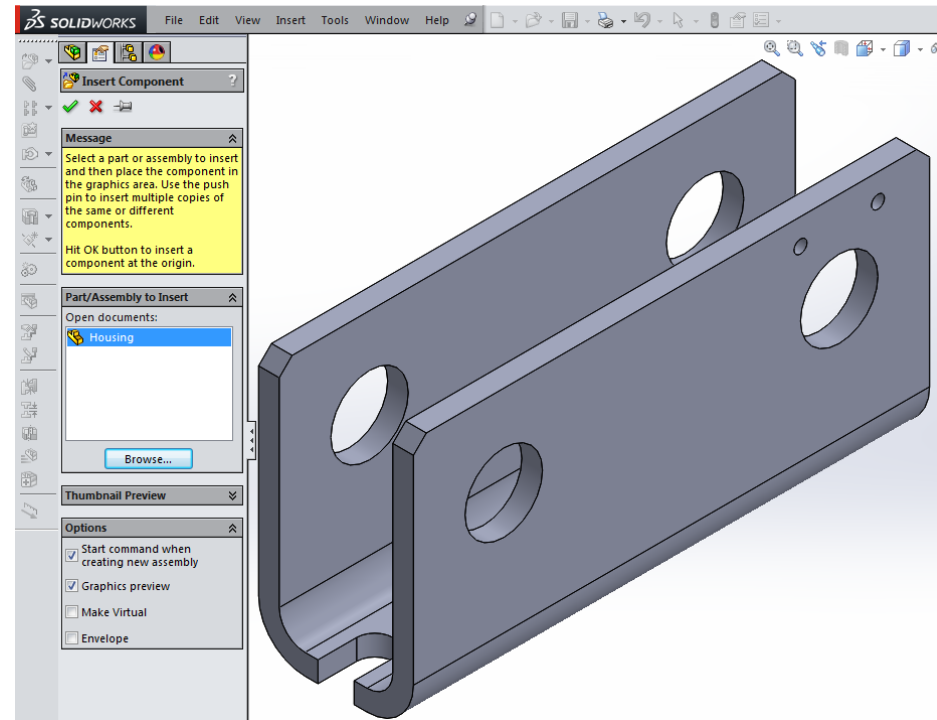


Fig4

Check the *Feature Manager* (Fig5)! It shows that the *Assembly* contains the 1st instance of the *Housing*. This instance of the part is in the *Default* configuration. The name of the part is preceded by (f) which means that the part is fixed – it cannot be moved. Usually the first component of an *Assembly* is fixed - the positions and the *Orientations* of the remaining *Components* are defined by mating. Mates are geometric relationships, such as coincidence, perpendicularity, tangency, and so on, between parts in an *Assembly*.

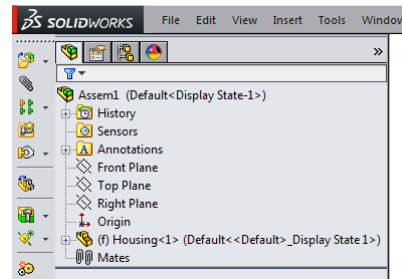


Fig5

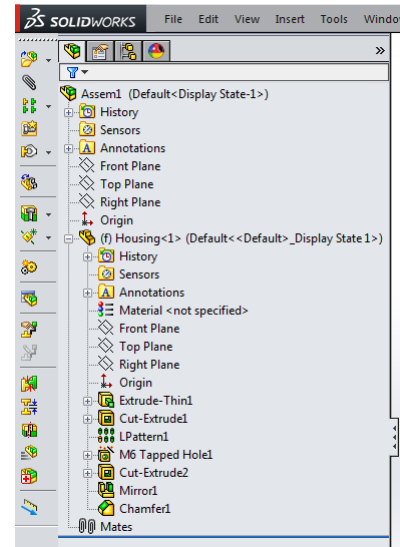


Fig6

Expand the *Housing* item of the list by clicking on the small plus sign (Fig6)! As you can see every feature of the part can be reached from the *Assembly*.

8.2 Inserting the second component

Use the *Insert Components* tool again and select *Shaft I* to insert! Click anywhere in the *Drawing Area* to place the part (*Fig1*)!

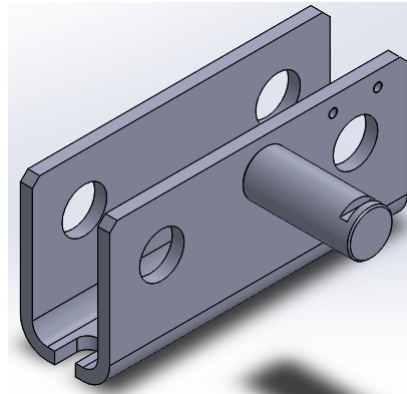


Fig1

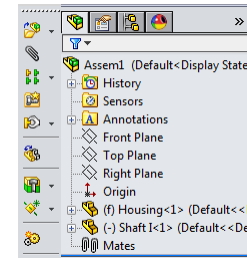


Fig2

Check the *Feature Manager* again (FigZ)\ The freshly inserted part appeared in it. The name of the part is preceded by (-) which means that the position of the part is not defined completely – the part can be dragged around in the *Drawing Area*.

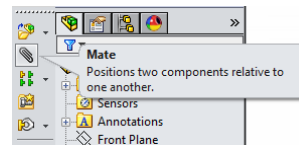


Fig3

Select the Mate tool in the *Assembly* toolbar to start defining mates (Fig3)!

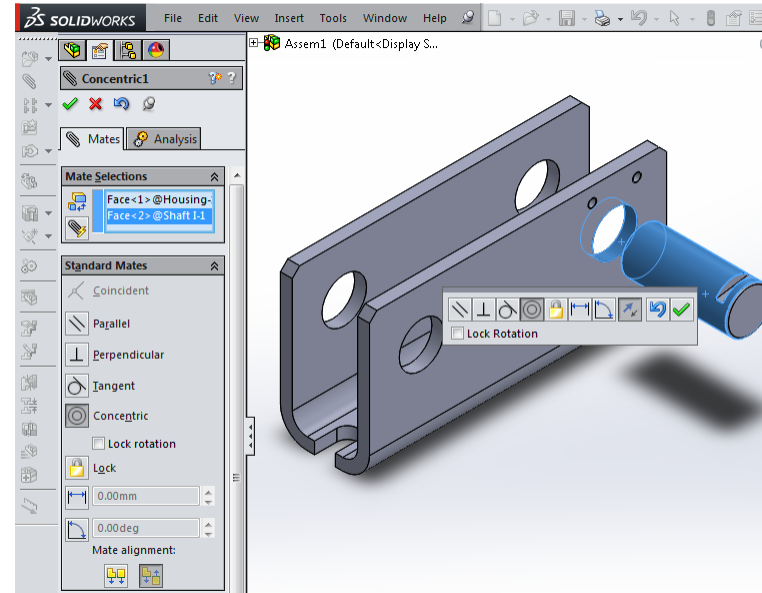


Fig4

Select the cylindrical surface of *Shaft I* and the cylindrical surface of the hole on *Housing* according to Fig4\ *SolidWorks* presumes that the most probable relation between these elements is their *Concentricity*. That's *Right* but before accepting the mate play with the

Mate alignment buttons in the *Standard Mates* group or with the *Flip Mate Alignment* button on the pop-up toolbar. The proper *Orientation* of the *Shaft* in the hole can be set up. Press *OK* to accept the mate.

If you click on the *Shaft* and try to drag it you can realize that its movement is limited.

- The *Shaft* can be moved along the hole;
- The *Shaft* can be rotated in the hole.

Define the axial position by a second mate! Select the outer side face of *Housing* and the side face of the slot on *Shaft I* (Fig5)! Accept the *Coincident* mate - the *Shaft* slides into the hole.

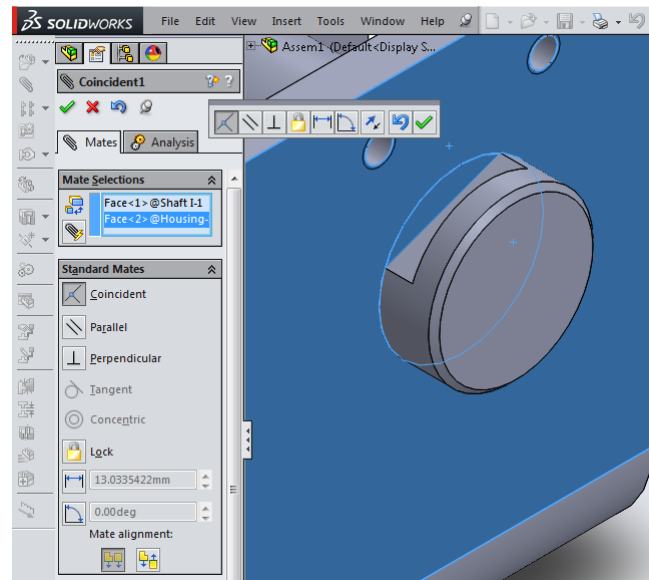


Fig5

Now the rotation is the only possible movement of *Shaft I*.

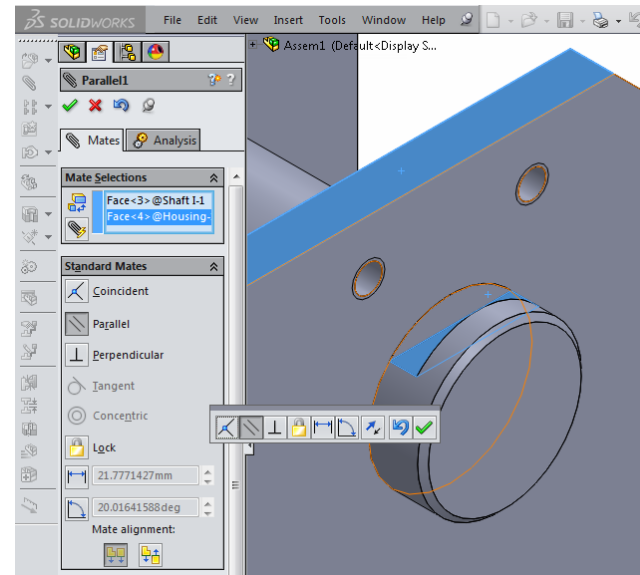


Fig6

Click on the bottom face of the slot of *Shaft I* and on the thin, horizontal upper face of *Housing* (Fig6)\ *SolidWorks* supposes that we want do define the parallelity of the selected faces – we just have to accept the *Parallel* mate. Press *OK* to end defining mates.

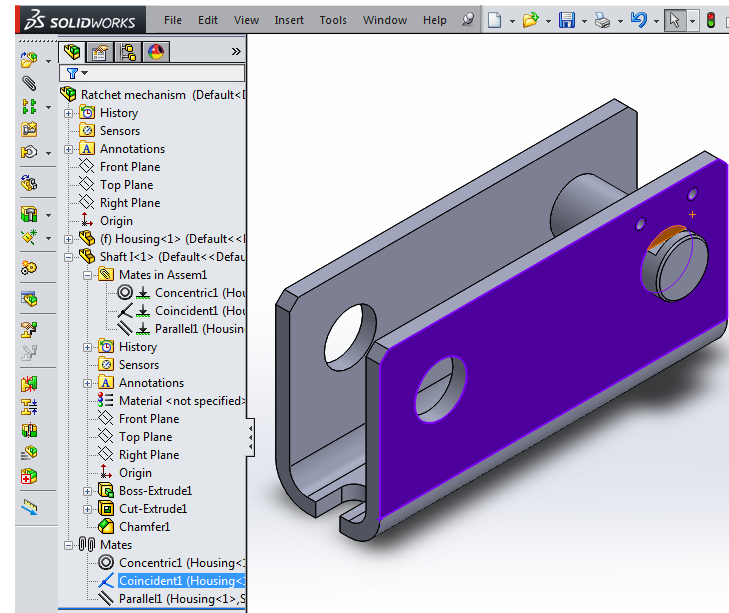


Fig7

Check the *Feature Manager* (Fig7)\ The minus sign preceding the *Shaft 1* item disappeared - the position of the part is fully defined. The defined mates are listed at the end of the feature list - they can be selected here to edit; exactly the same way as features. In a complex *Assembly* hundreds of mates can be listed which makes hard to find a mate which defines the position of a given component. To make the search easier the referenced mates are collected in the sublists of the *Components*.

For example the same Coincident! mate can be found on three places:

- At the end of the list among all of the mates;
- In the sublist of *Shaft I*;
- In the sublist of *Housing*.

Of course if you define more mates than it is necessary the position of the component becomes overdefined.

8.3 Inserting more Components

Insert the *Fixture* into the *Assembly* (Fig1)!

The insertion can be performed several ways:

- Using the *Insert Components* tool in the *Assembly* toolbar;
- Using the *Drag&Drop* method the component can be carried from an open *SolidWorks* document window or from *Windows Explorer*;
- Dragging from the *Components'* list and drop*Ping* to the *Drawing Area* while pressing *Ctrl* on the keyboard a new instance of a previously inserted component can be placed into the *Assembly*.

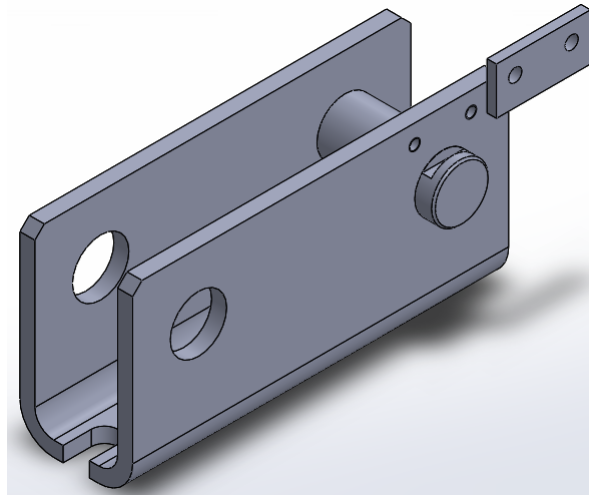


Fig1

Define mates! Select the side face of *Housing*! Change the viewing direction to see the opposite side of *Fixture* and select its back face (Fig2)! Accept the *Coincident* mate!

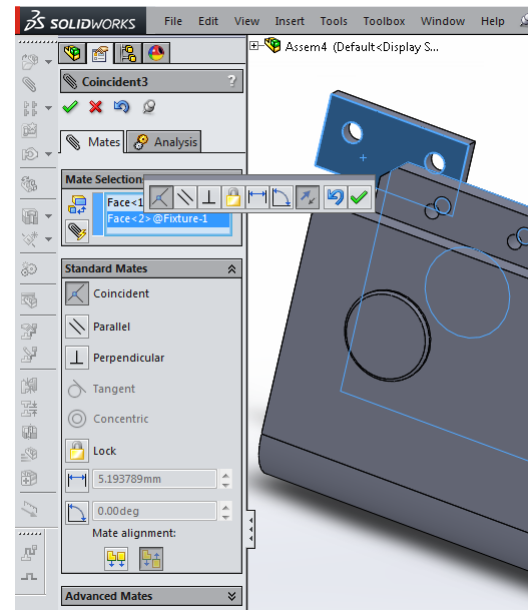


Fig2

Make the *Right* hole of the *Fixture* and the *Right* threaded hole of the *Housing* *Concentric* (Fig3)!

Finally prevent the possible rotation of the *Fixture*] Select the upper face of the *Fixture* and the upper narrow face of the *Housing*. *SolidWorks* suggests the *Coincident* relation but it is enough if the selected faces are *Parallel* (Fig4). The position of the *Fixture* is fully defined.

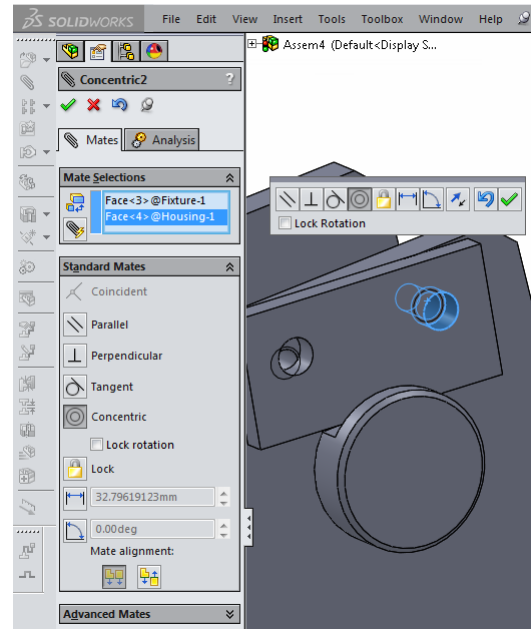


Fig3

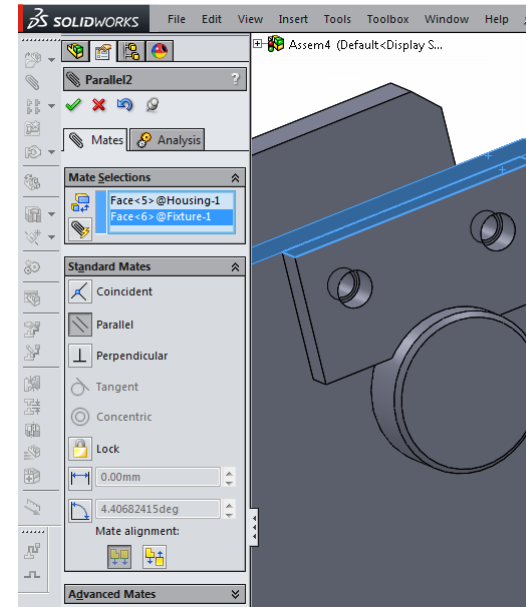


Fig4

Not only faces but also edges, vertices and other elements can be selected for defining mates. The experience is that *SolidWorks* solves assemblies easier if faces are used for mating.

In the real world the installation of the *Pawl* would be impossible. The assembled mechanism is simple, so it is possible to overview if the *Components* can be mounted or not. In case of a more complicated *Assembly* it is advisable to define mates following the mounting order strictly. This way we can avoid designing unmountable constructions.

Insert the *Pawl* into the *Assembly* and start mating! Define the *Concentric* mate between the hole of the *Pawl* and the cylindrical surface of *Shaft I* (Fig5)!

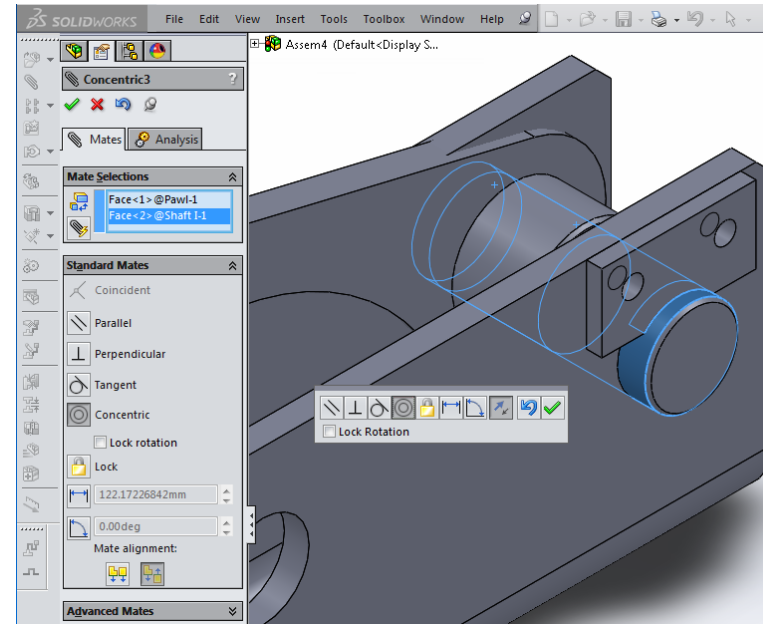


Fig5

Select the planar end face of the *Pawl's* hub! This face will be *Coincident* with the inner surface of the *Housing*. This face of the *Housing* is not visible from the current viewing direction. Of course the viewing direction could be changed dynamically but sometimes happen that a hardly visible face must be selected. *Right-click* on the visible outer face of the *Housing* and choose *Select Other* from the pop-up menu (*Fig6*). The *Select Other* dialog lists every hidden face behind the point on which you *Right-clicked* so the inner face of *Housing* can be selected (*Fig7*).

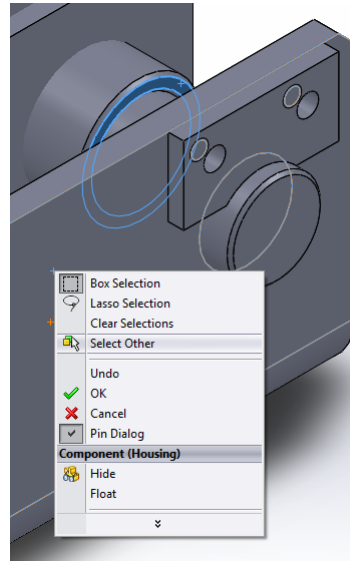


Fig6

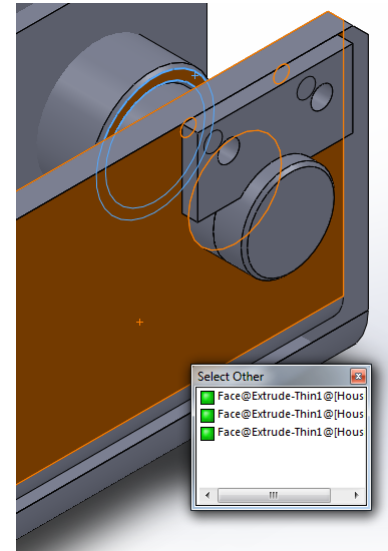


Fig7

Accept the suggested *Coincident* mate (Fig8). The *Pawl* can be rotated but the necessary mate can be defined after the positioning of the *Ratchet Wheel*.

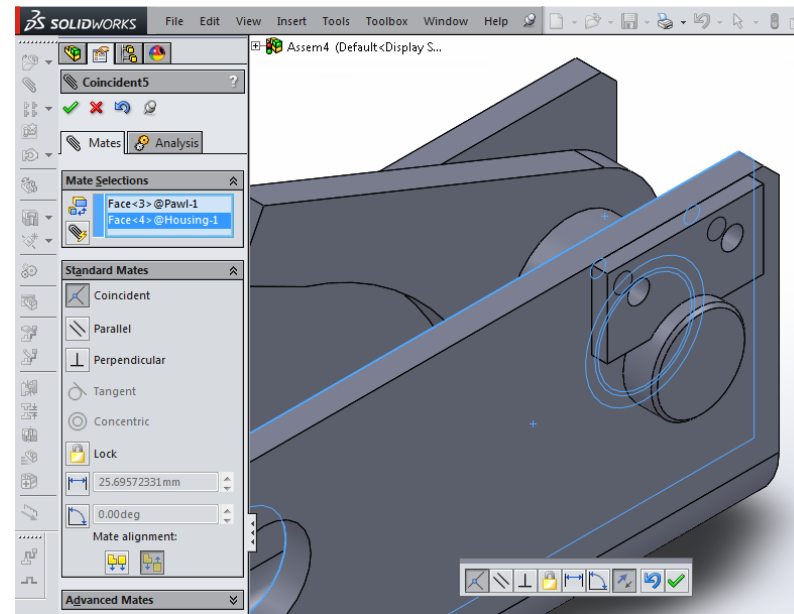


Fig8

Although it would be unmountable but create a sub*Assembly* for the other shaft of the mechanism.

Start a new *Assembly* document and insert *Shaft II* into it. Simply click OK to make the part fixed at the origin of the *Assembly*. Insert the *Ratchet Wheel* into the *Assembly* and place it in an obligatory position on the *Drawing Area* (Fig9).

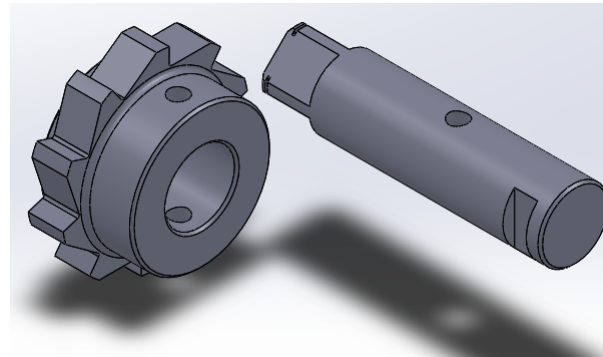


Fig9

Make the axial hole of the *Ratchet Wheel* and the cylindrical surface of *Shaft II* Concentric (Fig10).

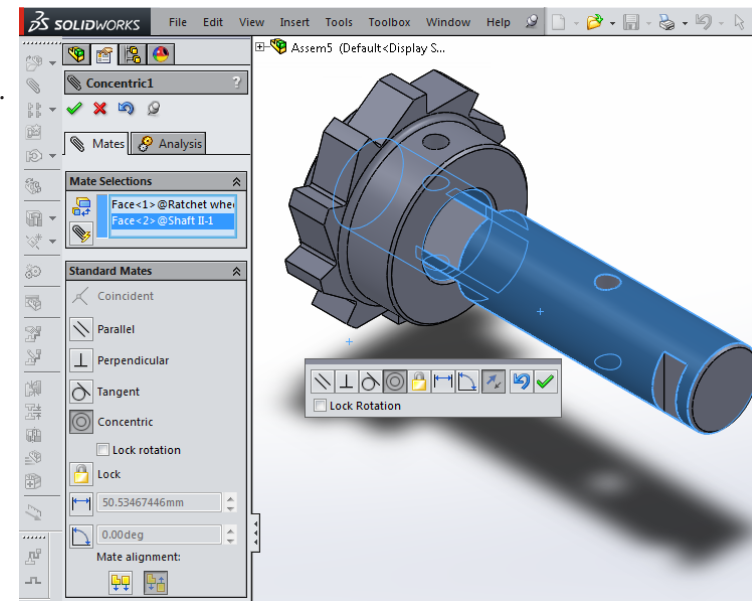


Fig10

Define the *Concentricity* of the radial holes of the parts (*Fig11*).

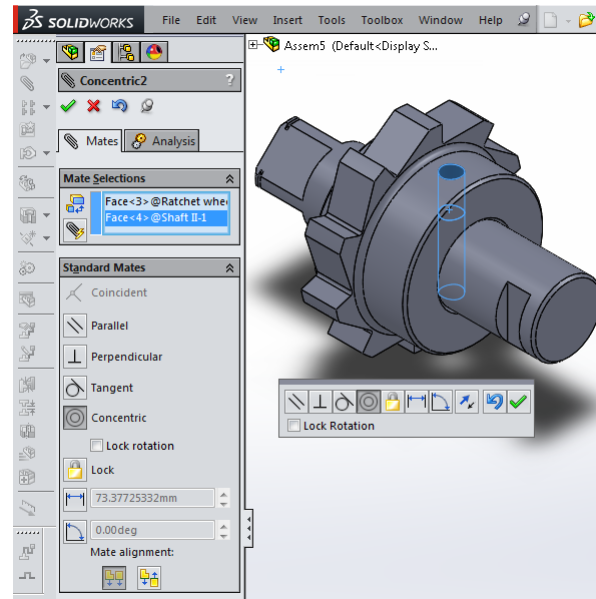


Fig11

The two *Concentricities* made the position of the *Ratchet Wheel* fully defined.

During the operation of the mechanism the torque is transferred from the shaft to the wheel by the *Pin*. Insert the *Pin* into the *Assembly* (*Fig12*) and make it *Concentric* with the radial hole of one of the *Components* (*Fig13*).

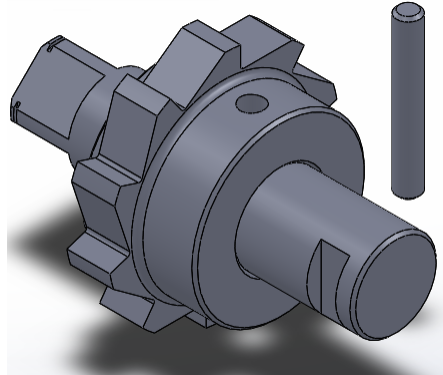


Fig12

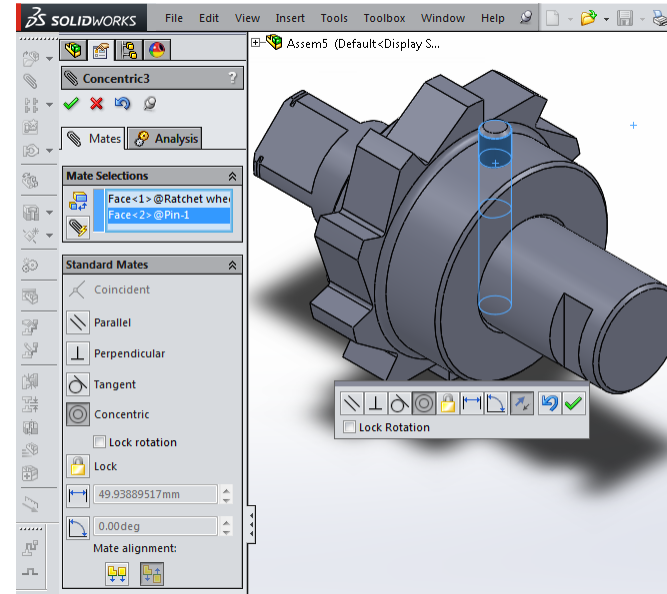


Fig13

How to push the *Pin* into the hole to be centered? Remember that the *Pin* was created using the Mid Plane option so its *Top Plane* is a symmetry plane. The *Components'* planes are available in the *Assembly* and they can be used for defining mates. Expand the *Components'* list then find the *Top Plane* of the *Pin* and the *Top Plane* of the *Ratchet Wheel*. Select both planes and apply the *Coincident* mating relation to them (Fig14). The *Pin* slides into the desired position.

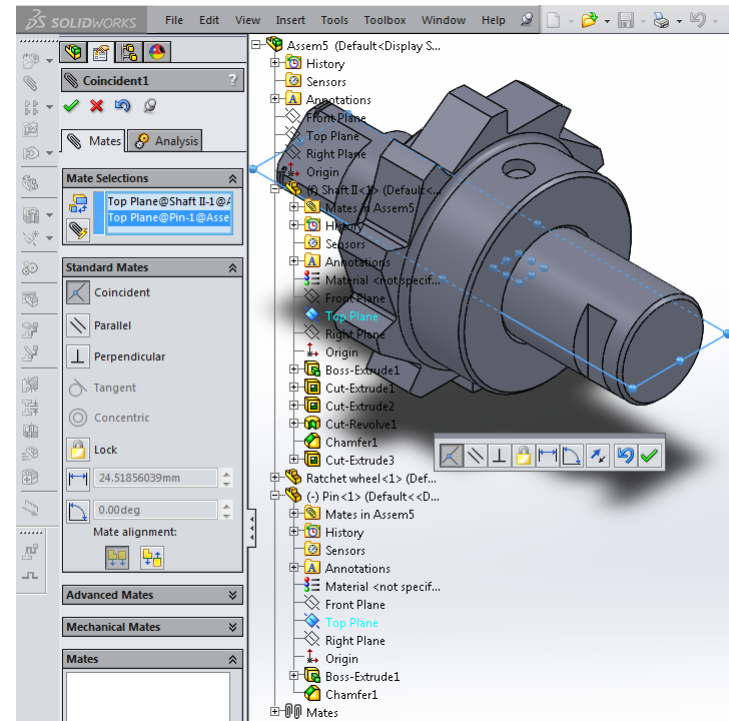


Fig14

It is hard to realize but the *Pin* can rotate in the hole. Use the planes of the *Components* again to stop the rotation! Select the Front Plane of the *Pin* and *Shaft II*! It is enough if these planes are Parallel – it will prevent the rotation (Fig15). Finish mating and save the document as *Ratchet Wheel Assembly.SLDASM*!

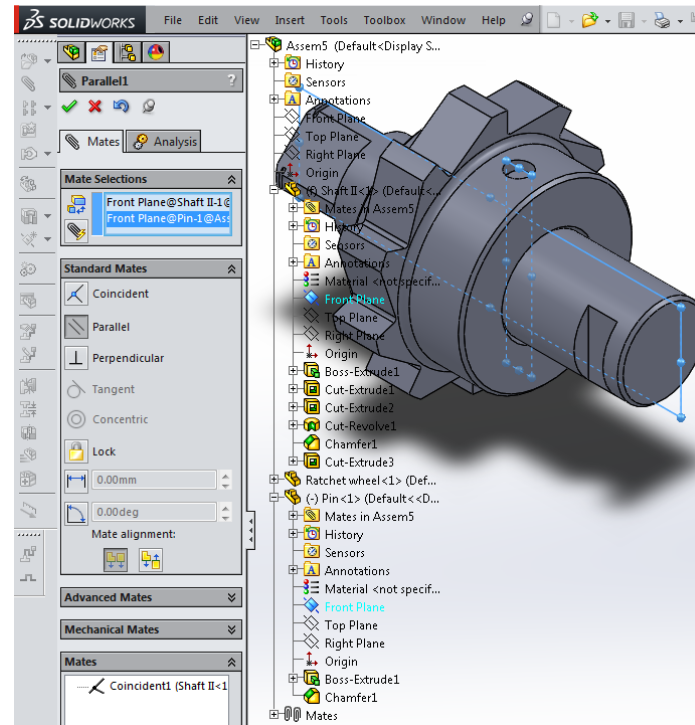


Fig15

The *Assembly* document can be downloaded from the following link:

[Ratchet Wheel Assembly.SLDASM](#)

Return to the window of the main assembly and insert the previously saved subassembly (*Fig16*)!

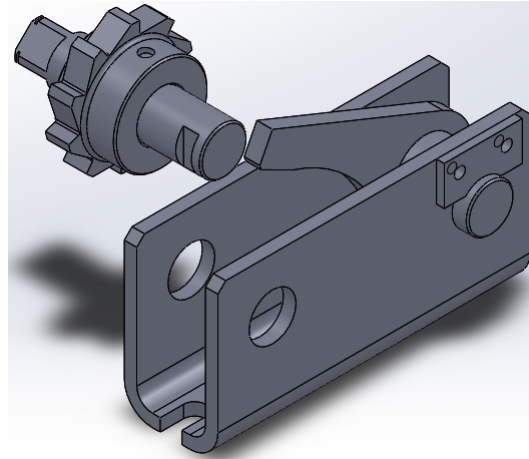


Fig16

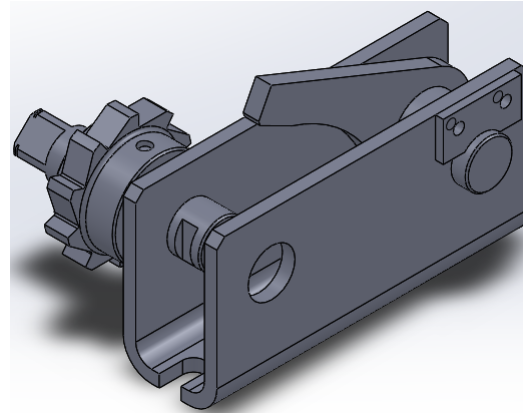


Fig17

Make the cylindrical surface of Shaft II and the free hole of the *Housing Concentric* (Fig17)!

The inner width of the *Housing* is 44 mm; the width of the *Ratchet Wheel* is 40 mm. We would like to divide the difference symmetrically on both sides of the wheel. This can be performed by an advanced mate. Expand the *Advanced Mates* group in the *Mate* dialog.

The necessary mating relation is the *Width* (Fig18). Select the two parallel inner faces of the *Housing* to define the width of a groove then select the two endfaces of the Ratchet Wheel to define the width of a tab. The width mate centers the tab within the width of the groove.

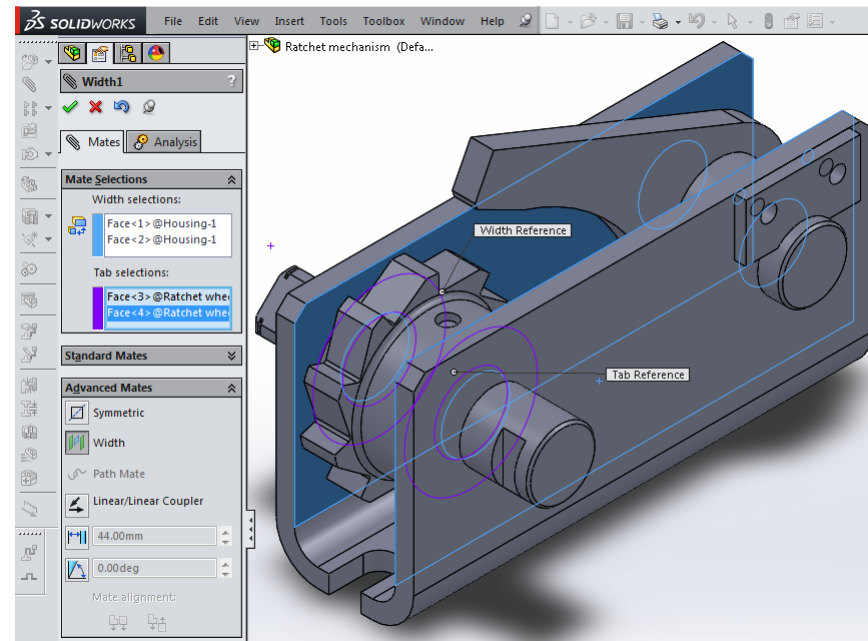


Fig18

Finally stop the rotation of the *Pawl* and the *Ratchet Wheel* by making their two faces *Coincident* according to Fig19!

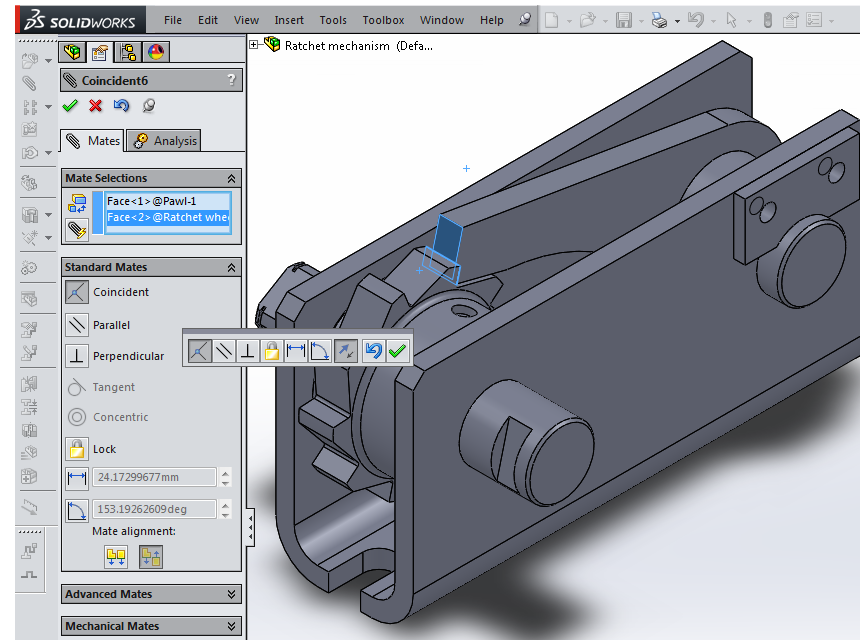


Fig19

The next component could be the *Spring*. Unfortunately it is impossible to mount into the *Assembly* because it was modeled in its unstressed form. During the *Assembly* the *Spring* must be deformed in order to act with a preloading force (torque) on the *Pawl*. What's more as the mechanism operates the *Spring* deforms constantly as it follows the movement of the *Pawl*. The usual solution of this problem is to model the *Spring* in two halves – one half will be fixed to the *Housing*; the other one will move together with the *Pawl* (Fig20).

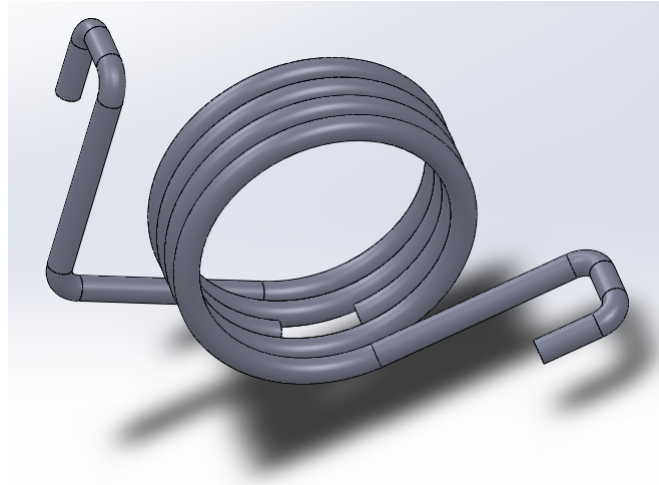


Fig20

The two spring halves can be downloaded from the following links:

[Half Spring I.SLDPRT](#)

[Half Spring H.SLDPRT](#)

Insert *Half Spring I* into the *Assembly*! Dig into the depth of the features of the part in the *Feature Manager* to find the sketch of the helical curve (*Half Spring I – Sweep1 – CompCurvel – Helix/Spirall – Sketch1*). Right-click on this sketch and find *Show* among the pop-up tools (*Fig21*). The circle of the sketch becomes visible in the *Drawing Area*.

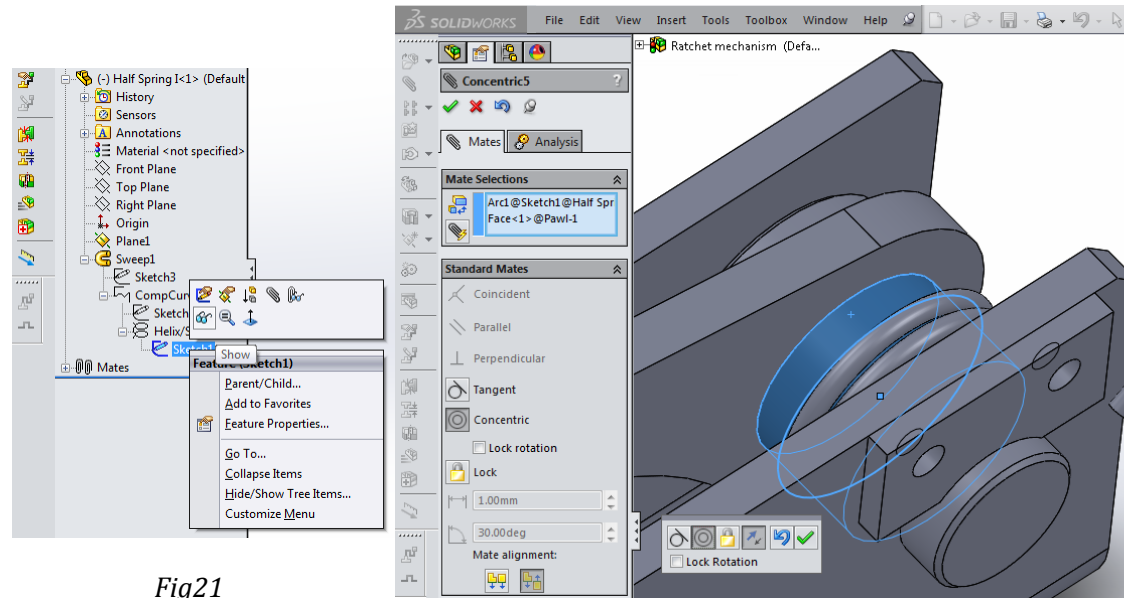


Fig21

Fig22

Start defining mates! Select the already displayed circle and the cylindrical outer surface of the hub of the *Pawl* then apply the *Concentric* mate (Fig22).

Find an appropriate viewing direction to see the inner side of the mechanism! Select the inner face of the *Housing* and the cylindrical surface of the spring wire then apply the *Tangent* mate – take care to the proper alignment (Fig23).

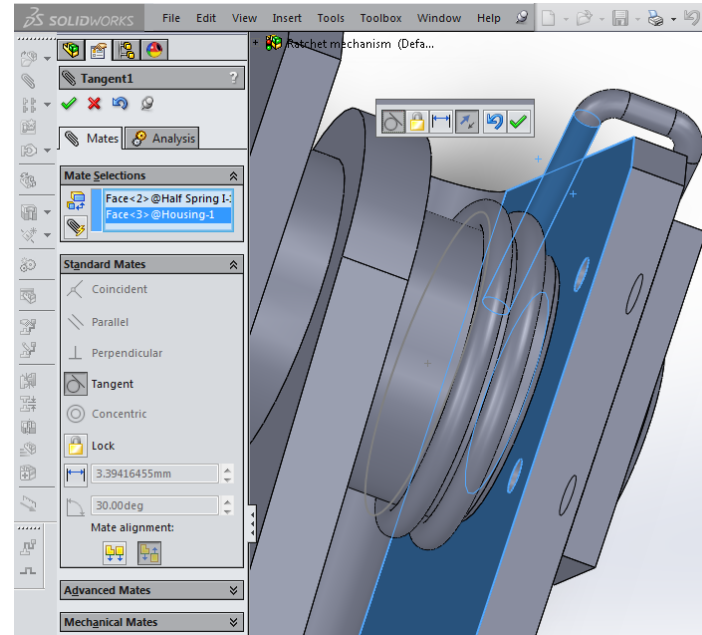


Fig23

Define another *Tangent* mate according to Fig24. If *SolidWorks* finds an improper solution the press the Back button, drag the component closer to the desired position and define the mate again.

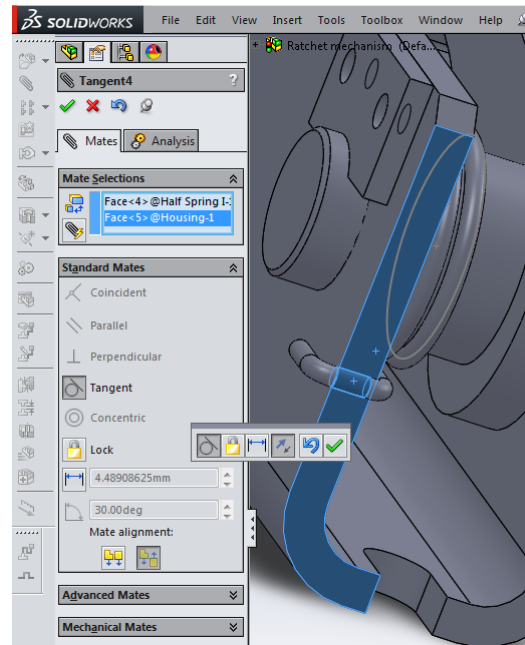


Fig24

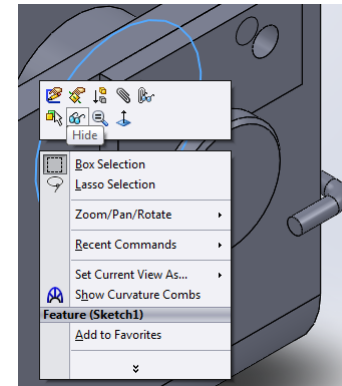


Fig25

Right-click on the circle of the *Helix* and select *Hide* from the pop-up tools (Fig25).

Insert *Half Spring II* into the *Assembly* and perform its positioning analogously to *Half Spring I*. Work independently! The result can be seen on Fig26.

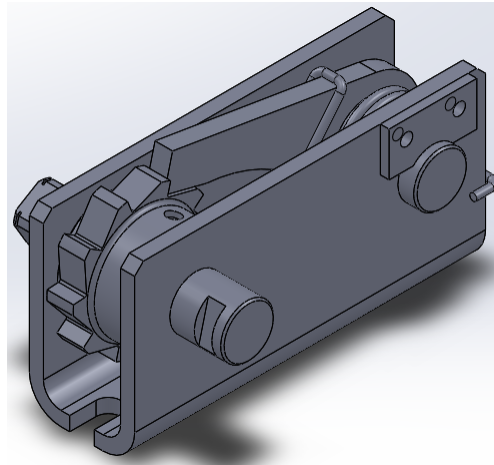


Fig26

The only missing *Components* are the screws. These elements will be inserted from the *SolidWorks Toolbox*.

SolidWorks Toolbox includes a library of standard parts. It is fully integrated with *SolidWorks*. The user can select from several standards and the desired component can be dragged into the *Assembly*. Toolbox supports international standards, including: ANSI, AS, GB, BSI, CISC, DIN, GB, ISO, IS, JIS, and KS. Toolbox includes various hardware parts: bearings, bolts, nuts, washers, keys, O-rings etc.

Click on the Design Library tab at the *Right* edge of the *Drawing Area* (Fig27)! Open the Toolbox and select the *ISO* folder then the *Bolts and Screws\Slotted Head Screws* subfolder (Fig28).



Fig27

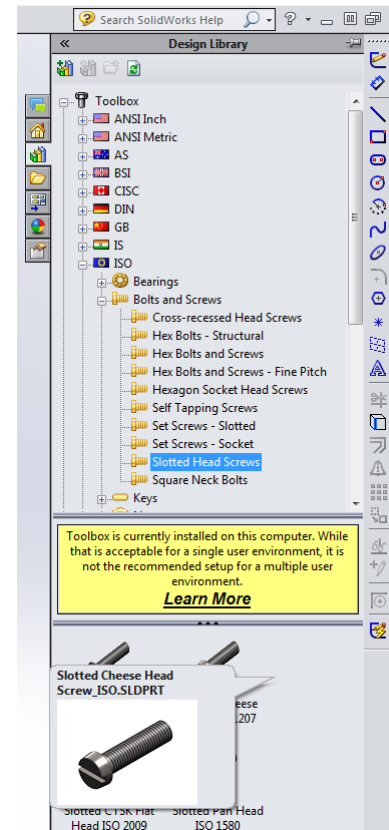


Fig28

Zoom on the *Fixture* then click on the *Slotted Cheese Head ISO 1207* in the Toolbox then drag it to the *Drawing Area*. Hover the screw over the receiving hole. *SolidWorks* analyses the geometry and finds out that an M6 screw is necessary. After a short while the screw finds its place - blinking symbols show what kind of mate will be created.

Release the mouse button to let the screw inserted (Fig29).

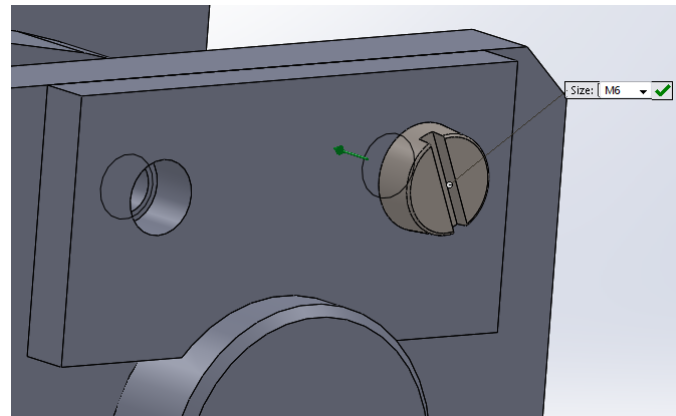


Fig29

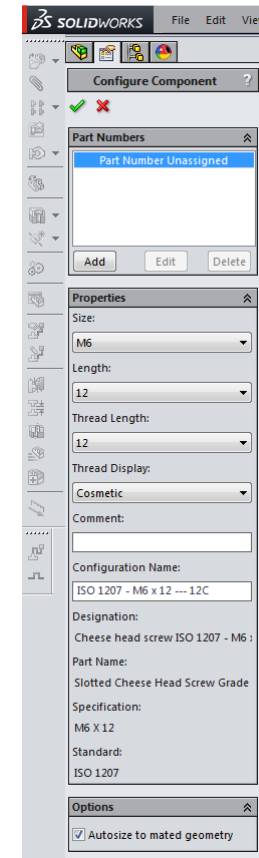
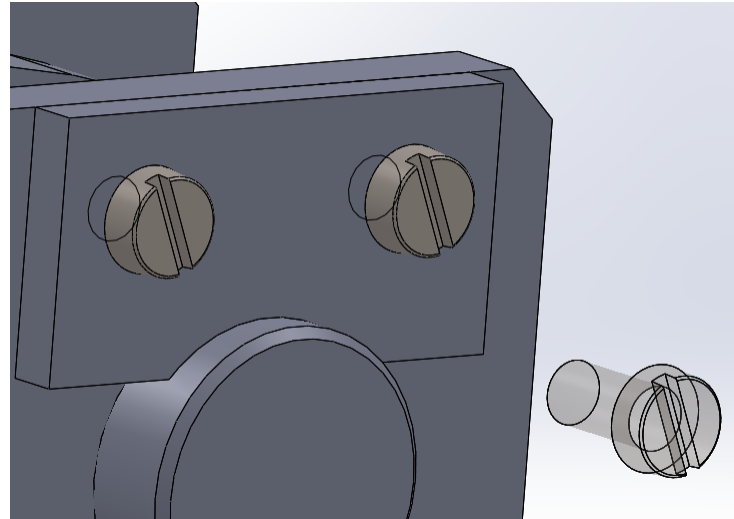


Fig30

Complete the part definition by filling the *Configure Component* dialog according to Fig30. After accepting the settings an additional instance of the selected screw can be inserted into the remaining hole (Fig31).

*Fig31*

Press Esc to stop inserting further screws!

The screws can rotate in their holes. To prevent the rotation define a *Coincident* mate for the *Top Planes* of the screws (Fig32) - this case these planes are called *Plane1*.

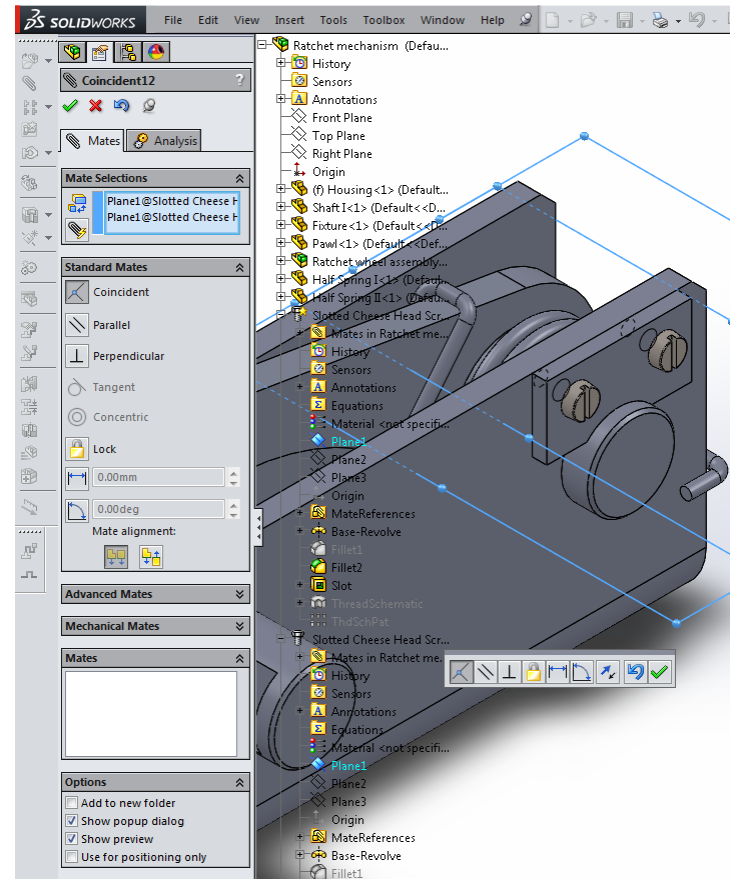


Fig32

Every component is inserted and their positions are fully defined so the *Assembly* model is completed (*Fig33*). The document can be downloaded from the following link:

[Ratchet mechanism.zip](#)

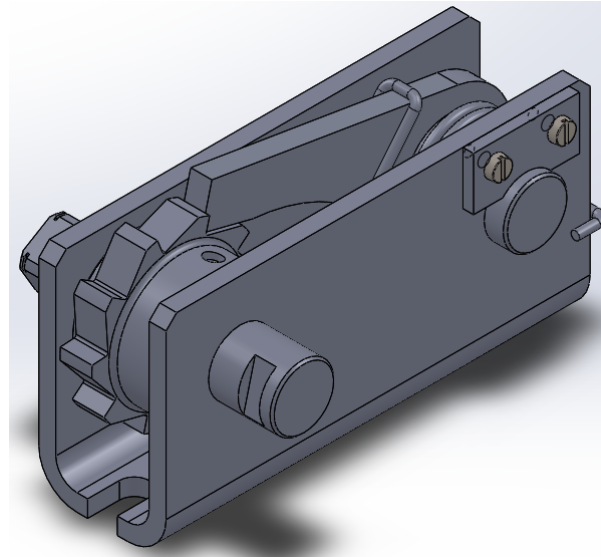


Fig33

9. fejezet

9 Practices

9.1 ASSEMBLY OF A MECHANISM I.

Download the compressed file from the following link and unzip its contents: [1396.zip](#)

Build the assembly on Fig1 from the unzipped parts!

- Use subassemblies to make the structure of the assembly clear;
- Make every component's position fully defined;
- Suppress one mate and demonstrate the operation of the mechanism.

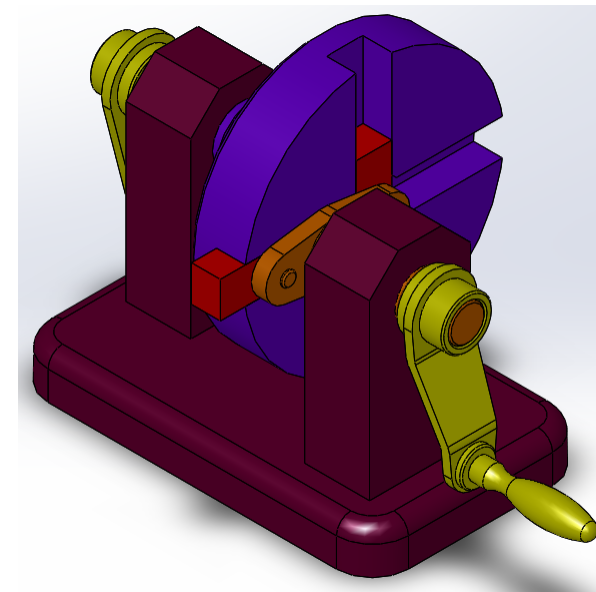


Fig1

9.2 ASSEMBLY OF A MECHANISM II.

Download the compressed file from the following link and unzip its contents:

[Oldham.zip](#)

Build the assembly on Fig1 from the unzipped parts!

- Use subassemblies to make the structure of the assembly clear;
- Make every component's position fully defined;
- Suppress one mate and demonstrate the operation of the mechanism.

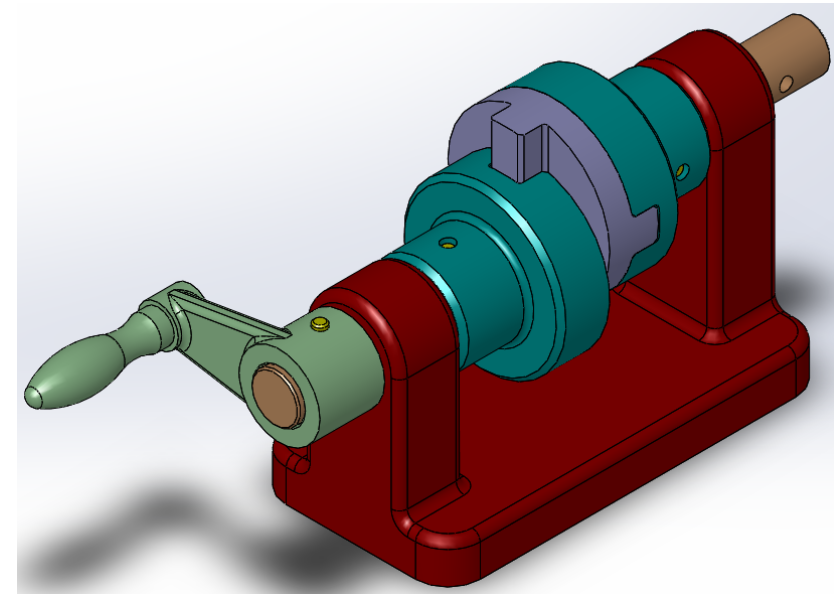


Fig1

10. fejezet

10.1 Test (Assemblies)

To check the students' assembly modeling skills a test is performed. A typical test can be seen below.

Download the compressed file from the following link and unzip its contents: [Saw.zip](#)

Build the assembly on Fig1 from the unzipped parts!

- Choose an appropriate fixed component;
- Make every component's position fully defined.

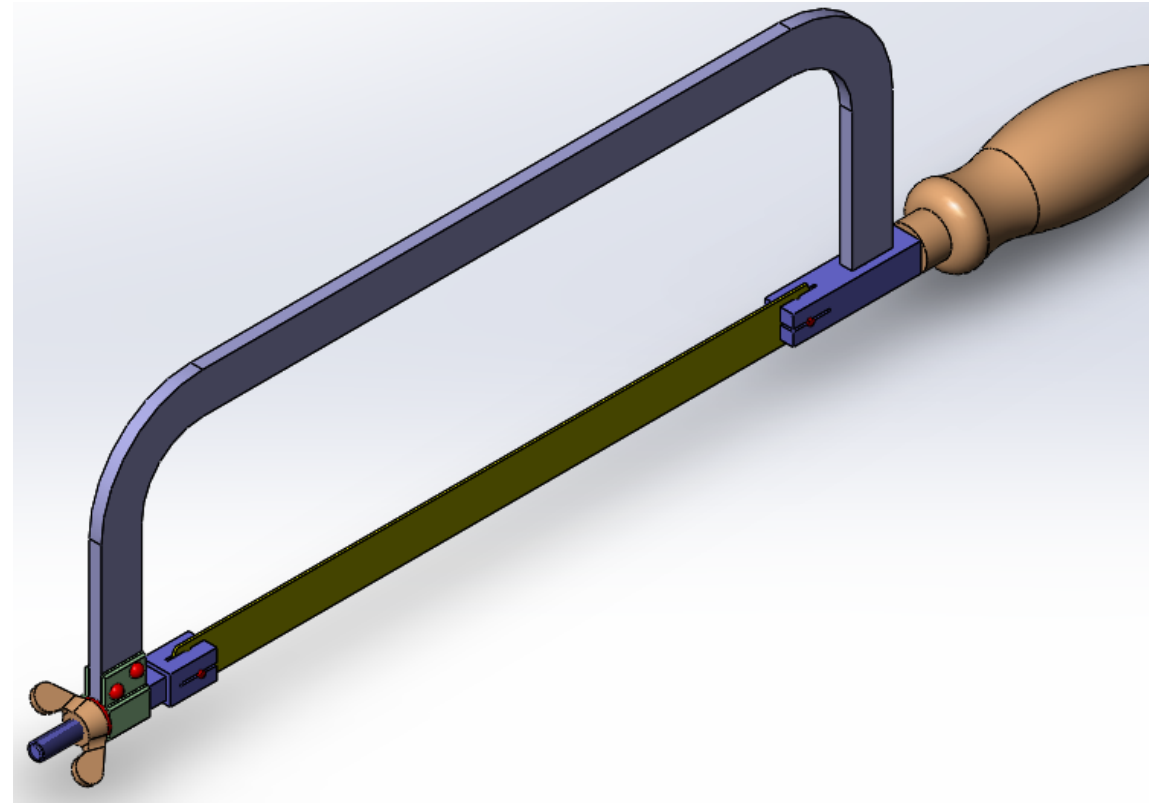


Fig1

11. fejezet

11.1 Drawings

In *SolidWorks* the user can create 2D drawings of the previously designed 3D solid parts and assemblies. Parts, assemblies, and drawings are linked documents; any changes of the part or assembly change the drawing document.

Generally, a drawing consists of several views generated from the model. Views can also be created from existing views. For example, a section view is created from an existing drawing view.

Dimensions in a *SolidWorks* drawing are associated with the model, and changes in the model are reflected in the drawing.

CREATING DRAWINGS OF PARTS

In this chapter we will see how to create and edit a part drawing. Part drawings contain all of the information which is necessary to reconstruct the exact shape of the described part. The drawing must contain every dimension to make the manufacture of the part possible.

SETTING UP THE DRAWING ENVIRONMENT

The regulations of engineering drawings are governed by standards. These standards are more or less different in the various countries. *SolidWorks* offers many options to set up the drawing environment according to the actual standard. The detailed review of these options is beyond the limitations of this workbook.

You can download drawing templates which contain all of the necessary settings to create drawings following the regulations of the corresponding Hungarian Standards. Download the compressed archive from the following link:

[SolidWorks Drawing Templates Eng.zip](#)

The archive contains various drawing sheets which contain the official title block of the College of Dunaújváros, Institute of Engineering.

You have to extract the archive into the folder where *SolidWorks* stores the document templates. Select the *Tools/Options...* item of the *Main* drop-down menu. Click on *File Locations* and select *Document Templates* in the *Show Folders for:* drop-down list (*Fig1*).

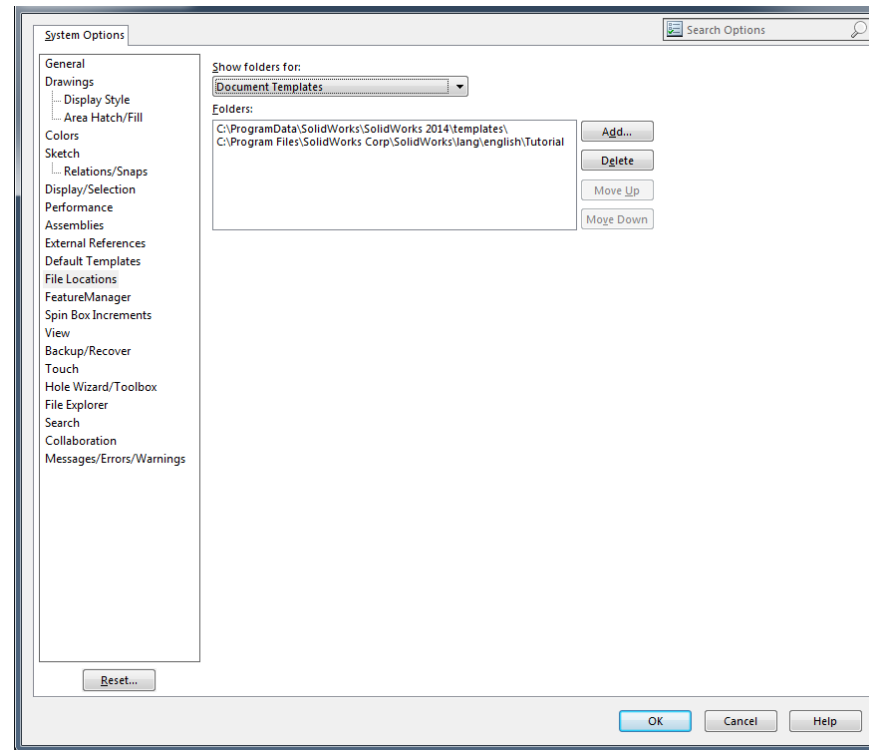


Fig1

Record the complete path of the first listed folder (*C:\ProgramData\SolidWorks\SolidWorks 2014\templates*) – this is the folder where the downloaded archive should be extracted. Press *Cancel* and perform the extraction.

Start a new document! Press the *Advanced* button on the *New SolidWorks Document* dialog (Fig2)!

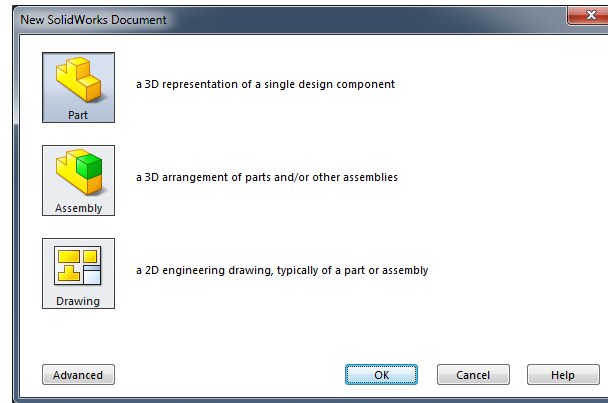


Fig2

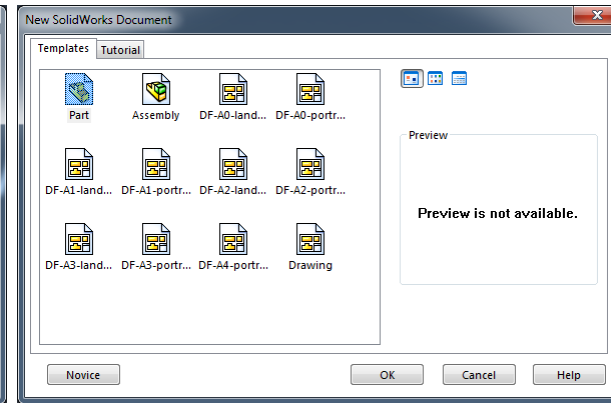


Fig3

The dialog lists the previously extracted templates together with the default *SolidWorks* templates (*Part*, *Assembly* and *Drawing*) (Fig3). Select the *DF-A3-landscape* template and press *OK*!

An empty, landscape oriented A3 sheet appears. A command has been started to run. Press *Cancel* to overview the user surface (Fig4).

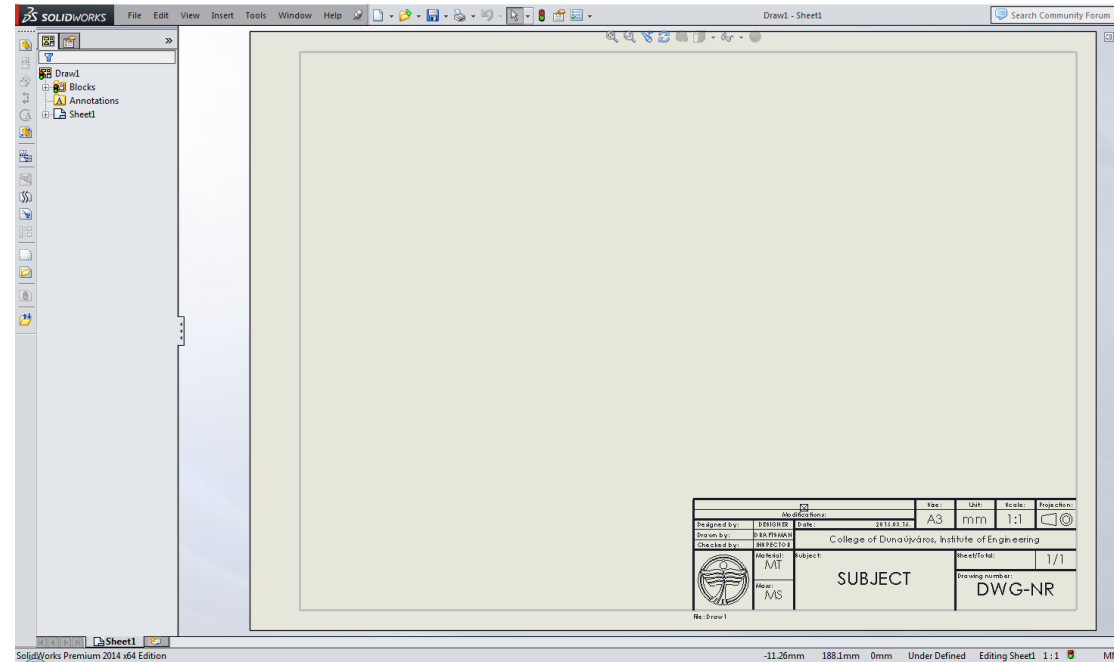


Fig4

- The *Drawing Area* shows the empty drawing sheet which is bordered and contains a title block. The border and the title block belong to the format of the sheet – it is impossible to modify them yet. The sheet will contain projections of parts, but also sketching is possible on it. Moreover also the projections can be supplemented by sketched entities;
- On the left edge of the screen the *Drawing* toolbar can be found now;
- The *Feature Manager* looks slightly different – the drawing contains no planes and no origin;
- The other toolbars around the *Drawing Area* are the same as previously.

11.2 Creating a projection

Create the drawing of the previously modeled Housing! The part document can be downloaded from the following link:

[Housing.SLDPRT](#)

Select the first tool (*Model View*) in the Drawing toolbar (*Fig1*)!

The *Model View* command starts to run. You may notice that this command was interrupted just after the creation of the drawing document. If the part document of the Housing is open in another window then it can be selected in the *Open documents*: list; otherwise it is necessary to *Browse* for it (*Fig2*). After the part document was opened click on the *Next* button. The outline of the new projection appears on the *Drawing Area*. Check the *Preview* checkbox in the *Orientation* group of the next page of the dialog to see the detailed preview of the projection (*Fig3*).

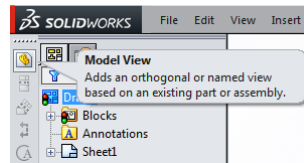


Fig1

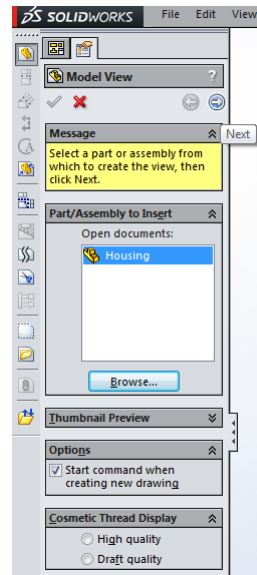


Fig2

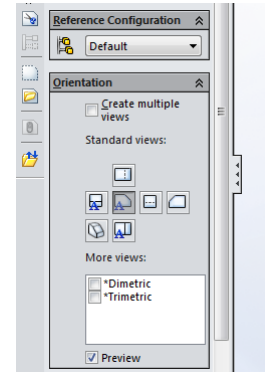


Fig3

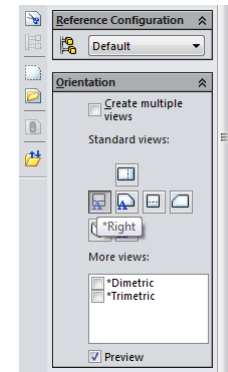


Fig4

The main view of the drawing will be the *Right* view. Select the *Right* view from the *Standard views*: group (Fig4) then put the projection onto a convenient place on the drawing sheet (Fig5)!

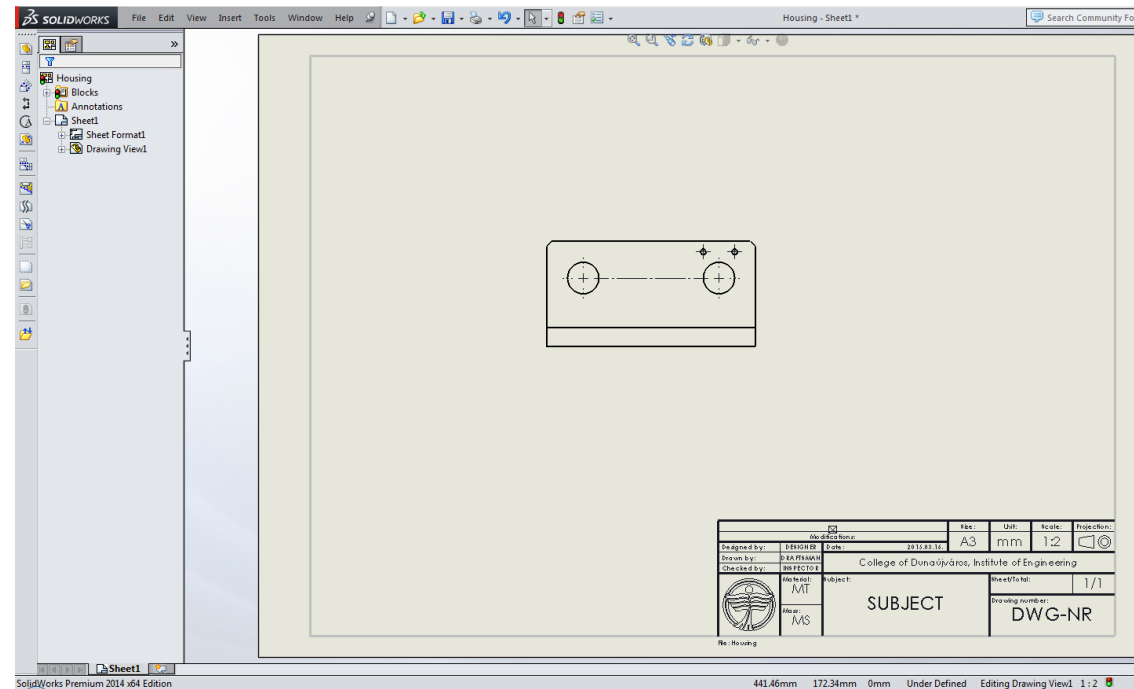
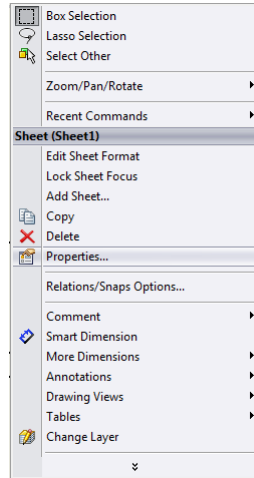
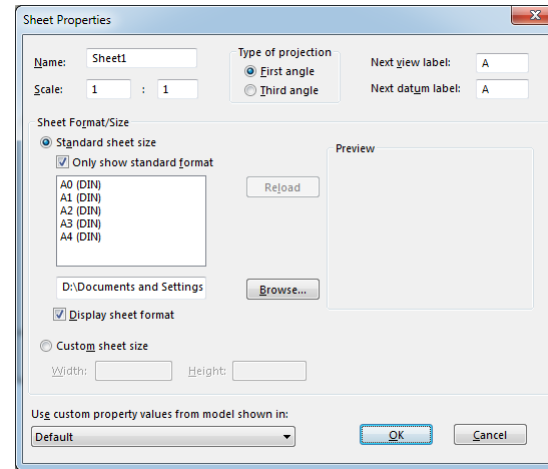


Fig5

SolidWorks starts to create a new arranged view – press *Esc* to stop.

Notice that *SolidWorks* has chosen a 1:2 scale for the drawing – the value appears automatically in the title block. This scale is determined to be the default scale of the drawing. It is too small so we will override it. *Right-click* on an empty area of the sheet and select *Properties...* in the pop-up menu (*Fig6*).

The Sheet *Properties* dialog appears (*Fig7*). It contains the default Scale – overwrite it to 1:1 and press *OK*!

*Fig6**Fig7*

Notice that not only the scale of the main view changed – the modification is reflected in the corresponding cell of the title block.

The created view (*Drawing View1*) appears at the end of the *Feature List* (*Fig8*). It contains the *Housing* together with all of its features.

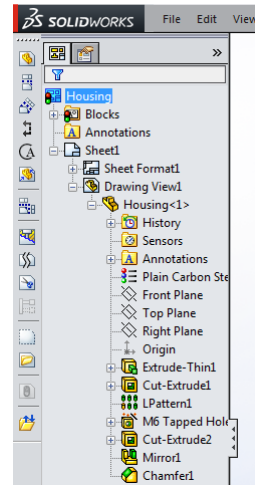


Fig8

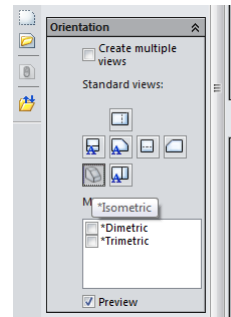


Fig9

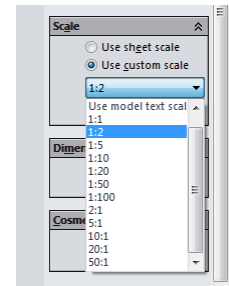


Fig10

It is customary to place a small, pictorial overview of the described part above the title block. Create this overview!

Select the *Model View* tool in the *Drawing* toolbar! Choose the *Housing* and press *Next*! Select the *Isometric* viewing direction (Fig9)! The new view will have different scale than the sheet's default; so click on the *Use custom* scale radio button and select 1:2 from the drop-down list (Fig10). Place the new view above the title block (Fig11).

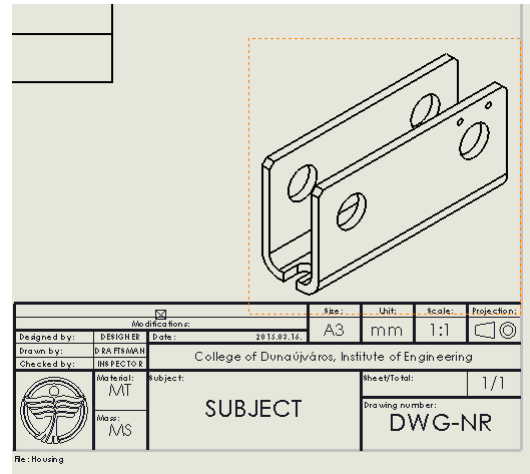


Fig11

It is conspicuous on the isometric view that the separating lines between the tangentially connected faces are described as edges. Such lines are called Tangent Edges in SolidWorks. These lines are not disturbing on the isometric overview but the description of them is not necessary on the main view.

Right-click on the main view then select *Tangent Edge/Tangent Edges Removed* from the pop-up menu to hide the unnecessary lines.

The engineering views contain dash-dot lines to inform the reader about the symmetrical *Properties* of the described part. The necessary dash-dot lines were placed automatically onto the main view. They are almost fine just a small modification is necessary.

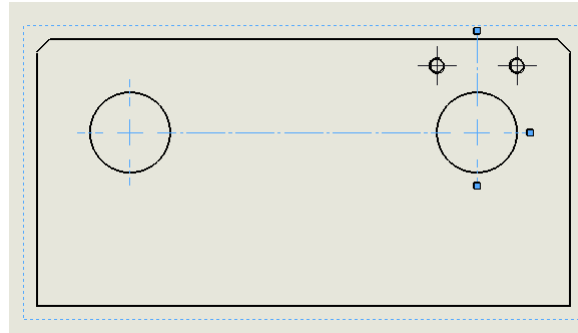


Fig12

Click on the endpoint of the vertical centerline of the *Right* large hole. Drag it upward to mark the symmetrical placement of the threaded holes (Fig12).

Finally click on the main view to select it. The selected state is marked by a thin, blue border drawn by dotted lines around the view (Fig13). The *Properties* of the view appear on the left side of the screen (Fig14).

- The view is the *Right* view of the model;
- It is displayed by removing the hidden lines;
- It uses the sheet's default scale;
- The thread symbols are drawn in draft quality.

These *Properties* can be modified at any time.

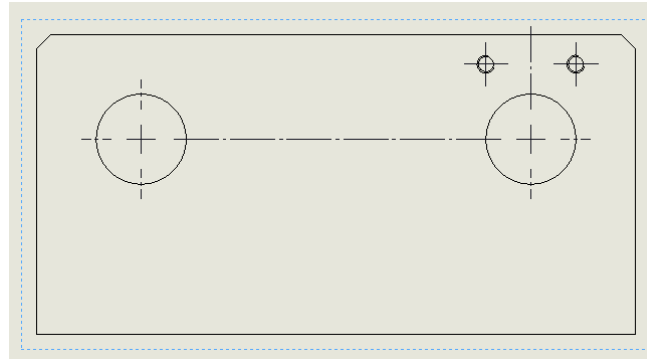


Fig13

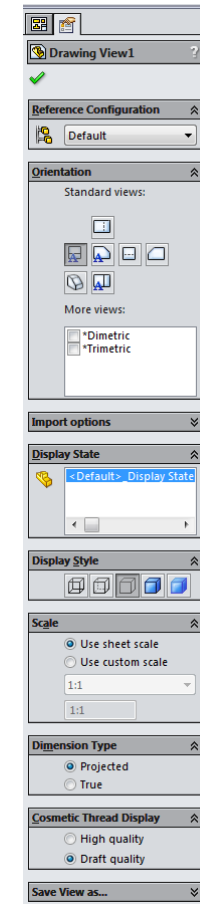


Fig14

11.3 Creating a projected view

The main view is not enough to describe the shape of the *Housing*. A top view is necessary to show the profile of the two cutouts at the bottom of the part. The top view is arranged to the main view – it will be created as a *Projected View*.

Select the Projected View tool in the *Drawing* toolbar (*Fig1*)! Click on the main view and drag the pointer downward – the preview of the new arranged view appears (*Fig2*). Place the projected view below the main view then press *Esc* to stop creating more new views.

Notice that the projected view follows the European arrangement according to the appropriate setting in the *Sheet Properties* dialog (**Error! Reference source not found.**). If the *Type of Projection* radio button is set to *First angle* the placement of the arranged views follows the European method – *Third angle* means the American method.

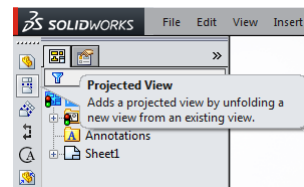


Fig1

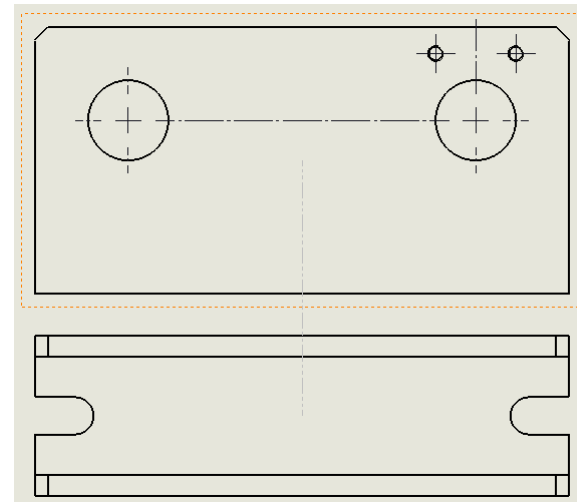


Fig2

The projected view inherited the properties of the main view:

- The display style is the same (*Hidden Lines Removed*);
- The scale of the view is the default scale of the sheet;
- The *Tangential Edges* are removed.

The projected view follows the modification of the parent view:

- If the main view is dragged to a different position the projected view moves too to keep the arrangement;
- If the viewing direction of the main view is modified also the projected is changing correspondingly.

The projected view contains no centerlines. The centerlines can be drawn by sketching or they can be inserted automatically.

To create an automatically placed centerline right-click on the projected view then select *Annotations/Centerline...* from the pop-up menu. Centerlines can be created either by clicking on a cylindrical/conical/toroidal face or by selecting two edges of the described part. Follow this latter method!

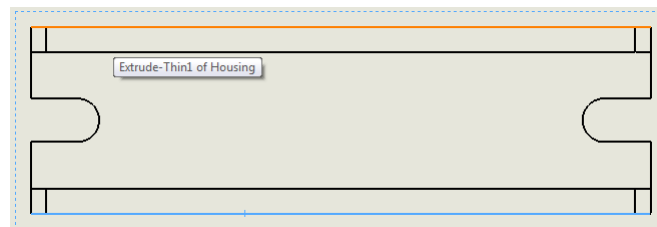


Fig3

Click on the lower horizontal outline of the part's view the click on the upper one (*Fig3*)!

SolidWorks draws the symmetry axis of the two selected element (*Fig4*). Press *Esc*! If unnecessary symmetry lines has been drawn then delete them.

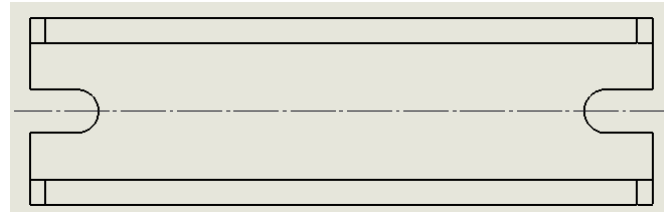


Fig4

Supplement the projected view with a sketched centerline! Use the *Centerline* sketch tool to draw a vertical centerline onto the projected view (*Fig5*). Take care to sketch onto the projection; not onto the drawing sheet. Notice the border around the view! This indicates that the sketched entity will belong to the projection. Press *Esc* to stop sketching more centerlines! Right-click on the sketched segment and choose Select *Midpoint* from the pop-up menu! Press *Ctrl* and click on the half circle of the left cutout (*Fig6*)! Make the selected midpoint and the half circle *Concentric* (*Fig7*)! Set up the correct length of the centerline by dragging its endpoint (*Fig8*)!

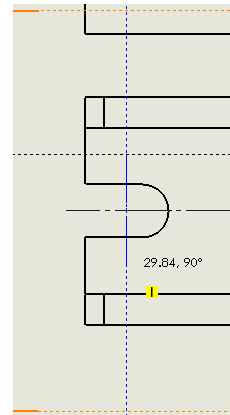


Fig5

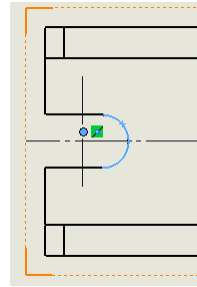


Fig6

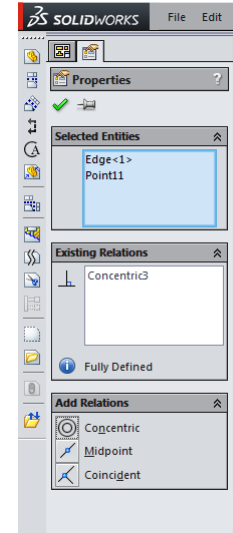


Fig7

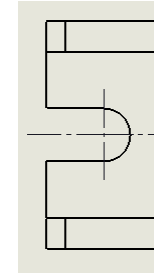


Fig8

Complete the projected view by inserting the vertical symmetry line of the part and by creating the vertical centerline of the right cutout (*Fig9*)!

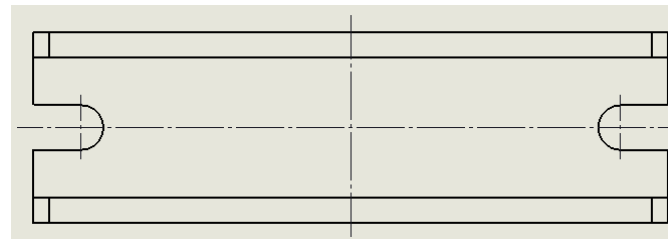


Fig9

CREATING A SECTIONAL VIEW

The exact shape of the part can not be recognized knowing only the main view and the projected view. The reader of the drawing is not able to see the U-shape of the part. Also impossible to realize that the threaded holes are located only on one wall of the part. At least one new projection is necessary. Because we would like to describe inner details this projection must be a sectional view. To show the right large holes and also one of the threaded holes a staged sectional view will be created.

Draw the tracing lines of the sectional planes' sequence first! Sketch three interconnected line segments onto the main view according to *Fig10*! Place a Point sketch entity (*Fig11*) onto each of the vertical segments (*Fig12*)!

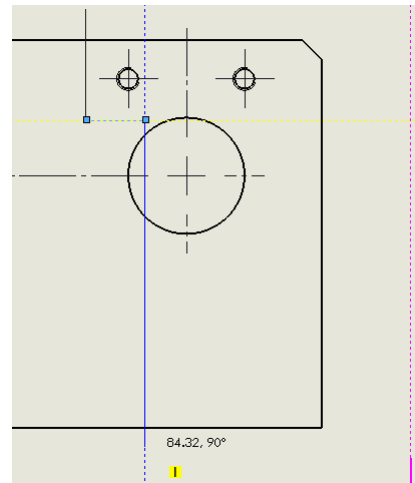


Fig10

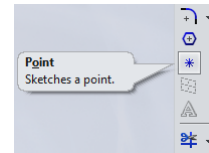


Fig11

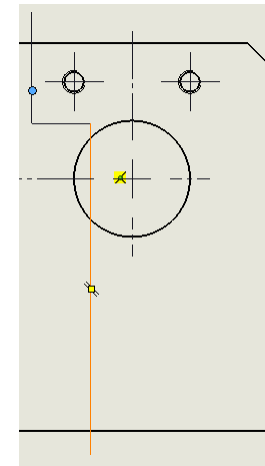


Fig12

Select the Point on the lower vertical sketch segment and the edge of the right large hole! Make them *Concentric* (*Fig13*)! Do the same with the Point on the upper vertical segment and the edge of left threaded hole! Drag the horizontal sketch segment to be vertically halfway between the large hole and the threaded hole!

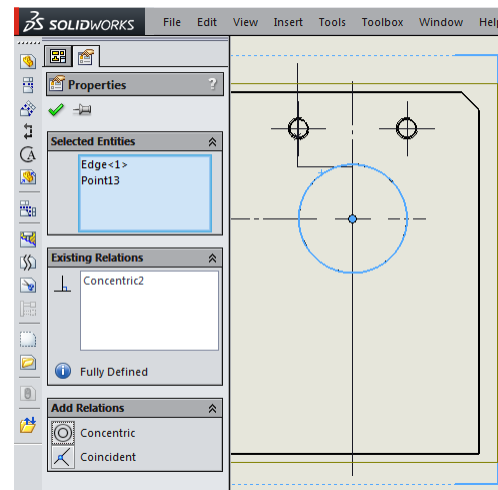


Fig13

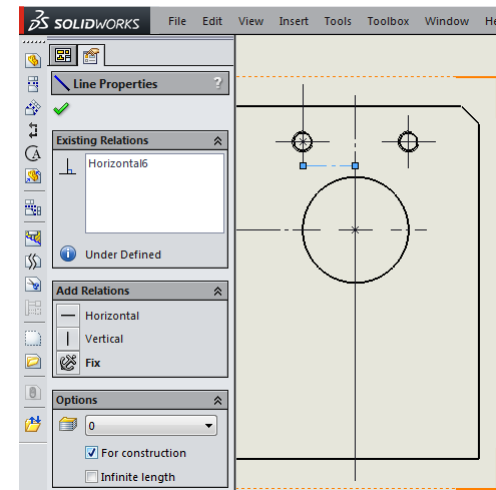


Fig14

Select the horizontal segment and check the *For construction* checkbox in the *Options* group of the *Line Properties* dialog (Fig14)!

Select one of the vertical segments and click on the *Section View* tool in the *Drawing* toolbar (Fig15)! The creation of the *Section View* starts – the preview of the sectional view appears (Fig16).

SolidWorks realizes that a staged sectional view is necessary:

- The sectioned portions of the part according to the continuous segments of the tracing line sequence are projected perpendicularly;
- The construction lines are ignored and the segment does not appear in the section view.

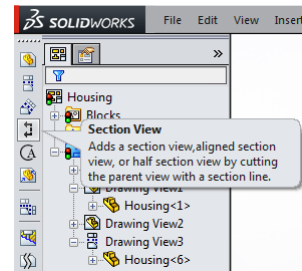


Fig15

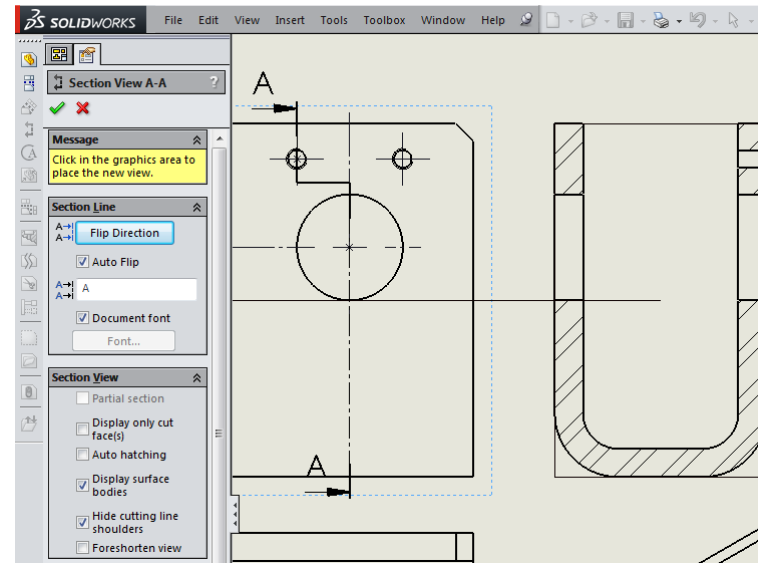
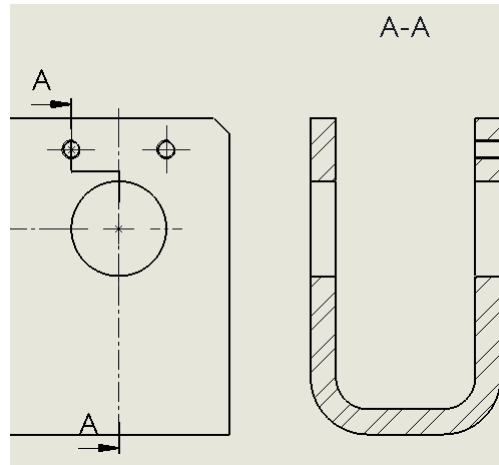


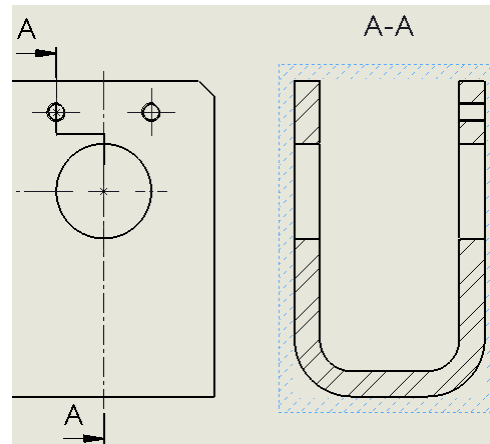
Fig16

Check the *Auto Flip* option; the identifier of the segment (A) and the *Hide cutting line* shoulders option according to (Fig16) then place the sectional view!

The sectional view has been created but some small aesthetical modifications are necessary (Fig17).

*Fig17*

- Drag the main view a bit upward to make some space; then drag the endings of the tracing line sequence away from the main view;
- Drag the identifier of the sectional view (A-A) downward to place closer to the sectional view (*Fig18*).

*Fig18*

Notice that a hatched border around the sectional view indicates that the projection is outdated. This has been happened because the section line was modified. Use the *Rebuild* tool to refresh model and all of its projections.

The hatching of the sectional view does not follow the Hungarian standard. Click on the hatched area to edit the hatch!

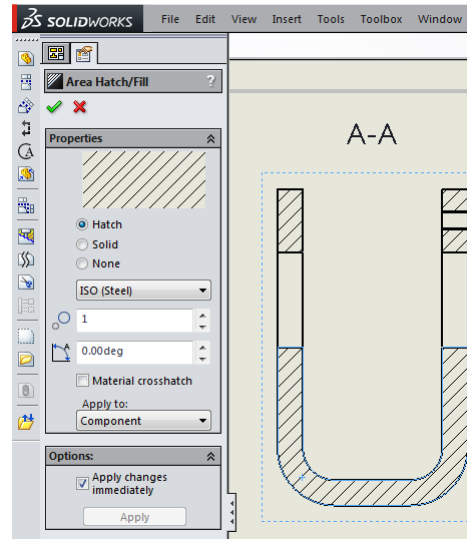


Fig19

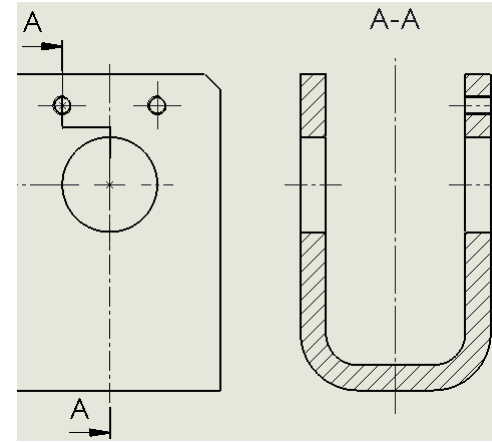
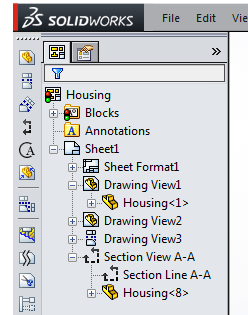


Fig20

Uncheck the *Material crosshatch* checkbox on the Area Hatch/Fill dialog to define the hatch properties manually (Fig19). Select *ISO (Steel)* from the drop-down list – this hatching pattern means the usual evenly spaced 45° stripping. Notice that the Hatch *Pattern Angle* is set to 0° – a left-slanted 45° stripping would require 90° pattern angle. Apply the hatch options for the complete component!

Finally insert the necessary centerlines into the sectional view (Fig20)!

Have a look at the *Feature Manager* (Fig21)! Not only *Section View A-A* can be found here but also its tracing line (*Section Line A-A*) – both the section view and the sketch of the section line can be edited later.

*Fig21*

Remark:

The *Point* entities of the sketch of the tracing line are visible. These elements will disappear when the drawing document will be printed.

11.4 Creating a sectional view

The exact shape of the part can not be recognized knowing only the main view and the projected view. The reader of the drawing is not able to see the U-shape of the part. Also impossible to realize that the threaded holes are located only on one wall of the part. At least one new projection is necessary. Because we would like to describe inner details this projection must be a sectional view. To show the right large holes and also one of the threaded holes a staged sectional view will be created.

Draw the tracing lines of the sectional planes' sequence first! Sketch three interconnected line segments onto the main view according to *Fig1*! Place a *Point* sketch entity (*Fig2*) onto each of the vertical segments (*Fig3*)!

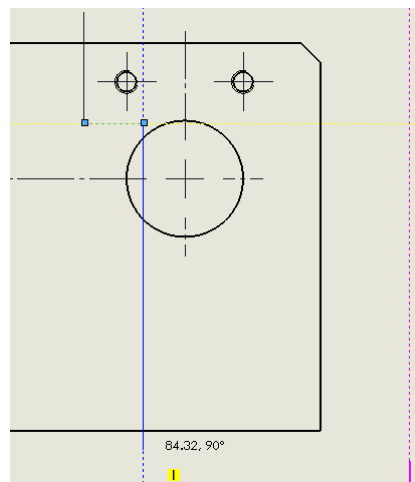


Fig1

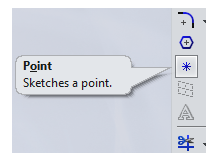


Fig2

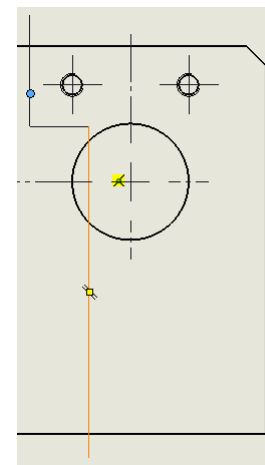


Fig3

Select the *Point* on the lower vertical sketch segment and the edge of the right large hole! Make them *Concentric* (*Fig4*)! Do the same with the *Point* on the upper vertical segment and the edge of left threaded hole! Drag the horizontal sketch segment to be vertically halfway between the large hole and the threaded hole!

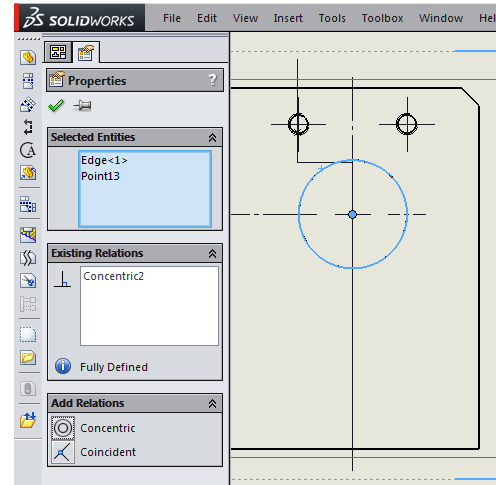


Fig4

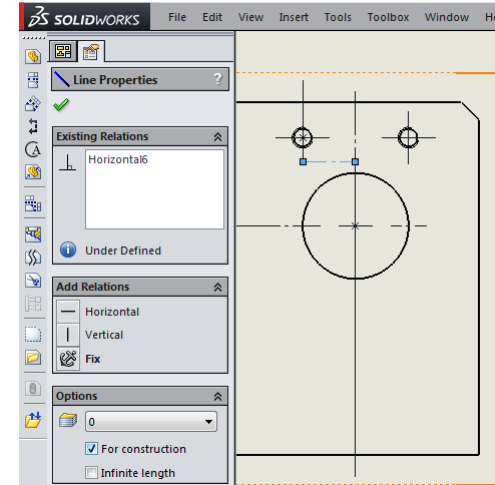


Fig5

Select the horizontal segment and check the *For construction* checkbox in the *Options* group of the *Line Properties* dialog (Fig5)!

Select one of the vertical segments and click on the Section View tool in the *Drawing* toolbar (Fig6)! The creation of the Section View starts – the preview of the sectional view appears (Fig7).

SolidWorks realizes that a staged sectional view is necessary:

- The sectioned portions of the part according to the continuous segments of the tracing line sequence are projected perpendicularly;
- The construction lines are ignored and the segment does not appear in the section view.

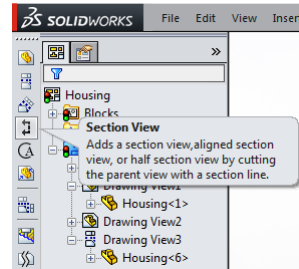


Fig6

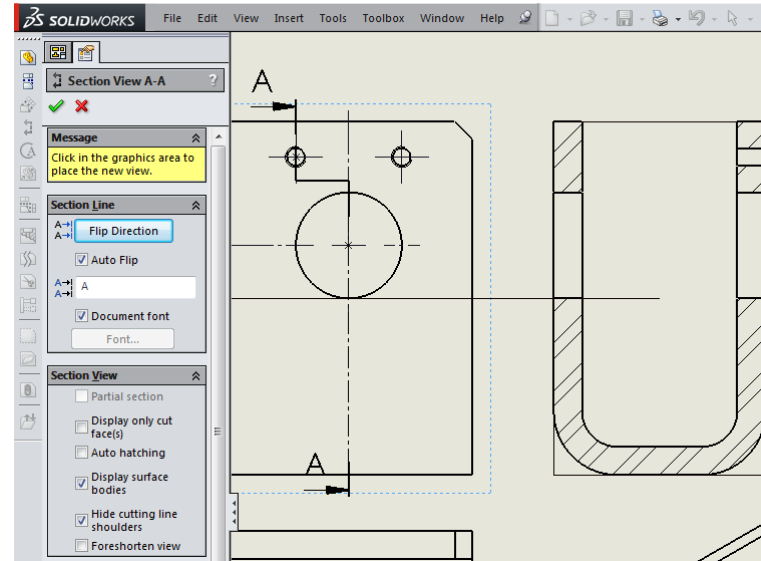
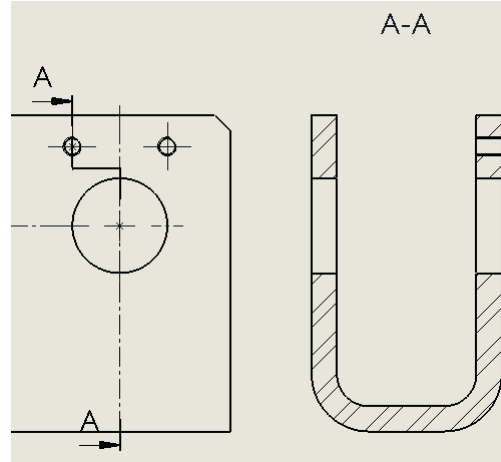


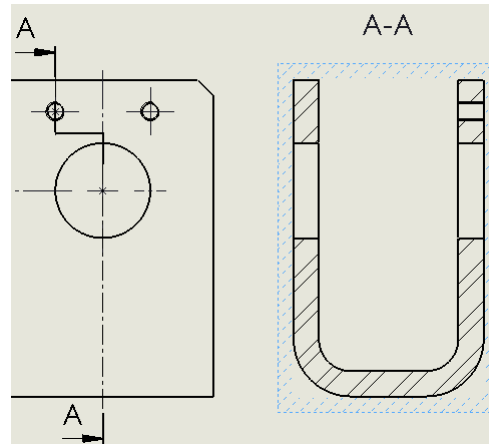
Fig7

Check the *Auto Flip* option; the identifier of the segment (A) and the *Hide cutting line shoulders* option according to (Fig7) then place the sectional view!

The sectional view has been created but some small aesthetical modifications are necessary (Fig8).

*Fig8*

- Drag the main view a bit upward to make some space; then drag the endings of the tracing line sequence away from the main view;
- Drag the identifier of the sectional view (A-A) downward to place closer to the sectional view (*Fig9*).

*Fig9*

Notice that a hatched border around the sectional view indicates that the projection is outdated. This has been happened because the section line was modified. Use the *Rebuild* tool to refresh model and all of its projections.

The hatching of the sectional view does not follow the Hungarian standard. Click on the hatched area to edit the hatch!

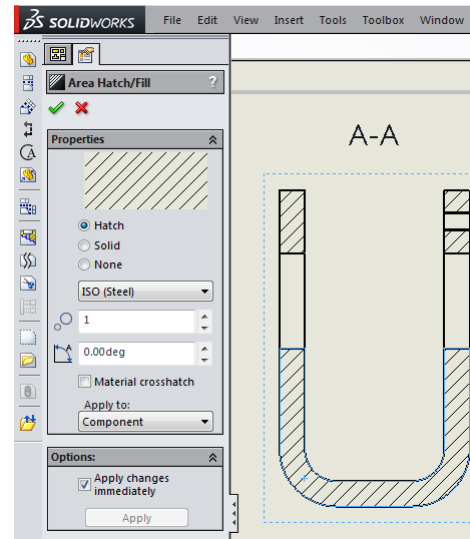


Fig10

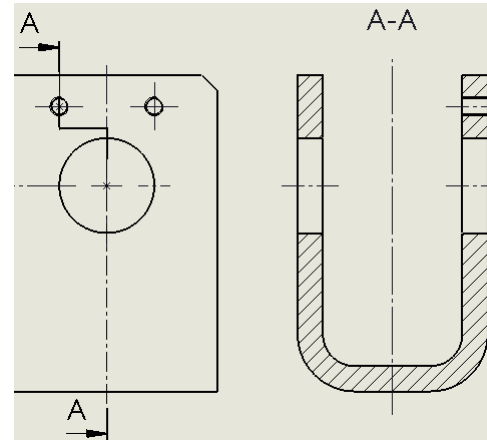
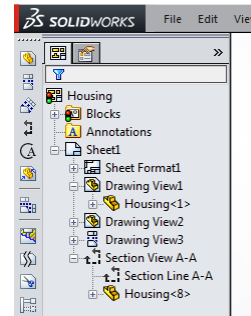


Fig11

Uncheck the Material crosshatch checkbox on the *Area Hatch/Fill* dialog to define the hatch properties manually (Fig10). Select *ISO (Steel)* from the drop-down list – this hatching pattern means the usual evenly spaced 45° stripping. Notice that the *Hatch Pattern Angle* is set to 0° - a left-slanted 45° stripping would require 90° pattern angle. Apply the hatch options for the complete component!

Finally insert the necessary centerlines into the sectional view (Fig11)!

Have a look at the *Feature Manager* (Fig12)! Not only *Section View A-A* can be found here but also its tracing line (*Section Line A-A*) – both the section view and the sketch of the section line can be edited later.

*Fig12*

Remark:

The *Point* entities of the sketch of the tracing line are visible. These elements will disappear when the drawing document will be printed.

11.5 Dimensioning

The projections (views and sectional views) inform the reader of the drawing about the shape of the described part. The drawing contains no information yet about the size of the part and its details. The drawing must be dimensioned.

The part model of the *Housing* contains the dimensions what are necessary to define the part. Fundamentally the same dimensions shall appear on the drawing so we will import the dimensions of the part into the drawing.

Select the *Insert/Model Items...* command in the *Main* drop-down menu!

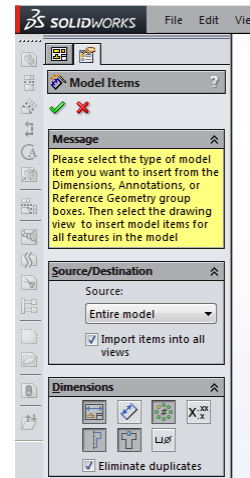


Fig1

Set up the options according to Fig1:

- The source of the import is the whole model not a selected feature;
- *SolidWorks* will determine the placement of the imported dimension using all of the projections;
- The *Dimensions Marked for Drawing*, the *Instance/Revolution Counts*, the *Hole Wizard Profiles* and the *Hole Wizard Locations* will be imported into the drawing;
- Every other option must remain unchecked.

Press *OK* to import! *SolidWorks* dumps the dimensions onto the drawing sheet (*Fig2*).

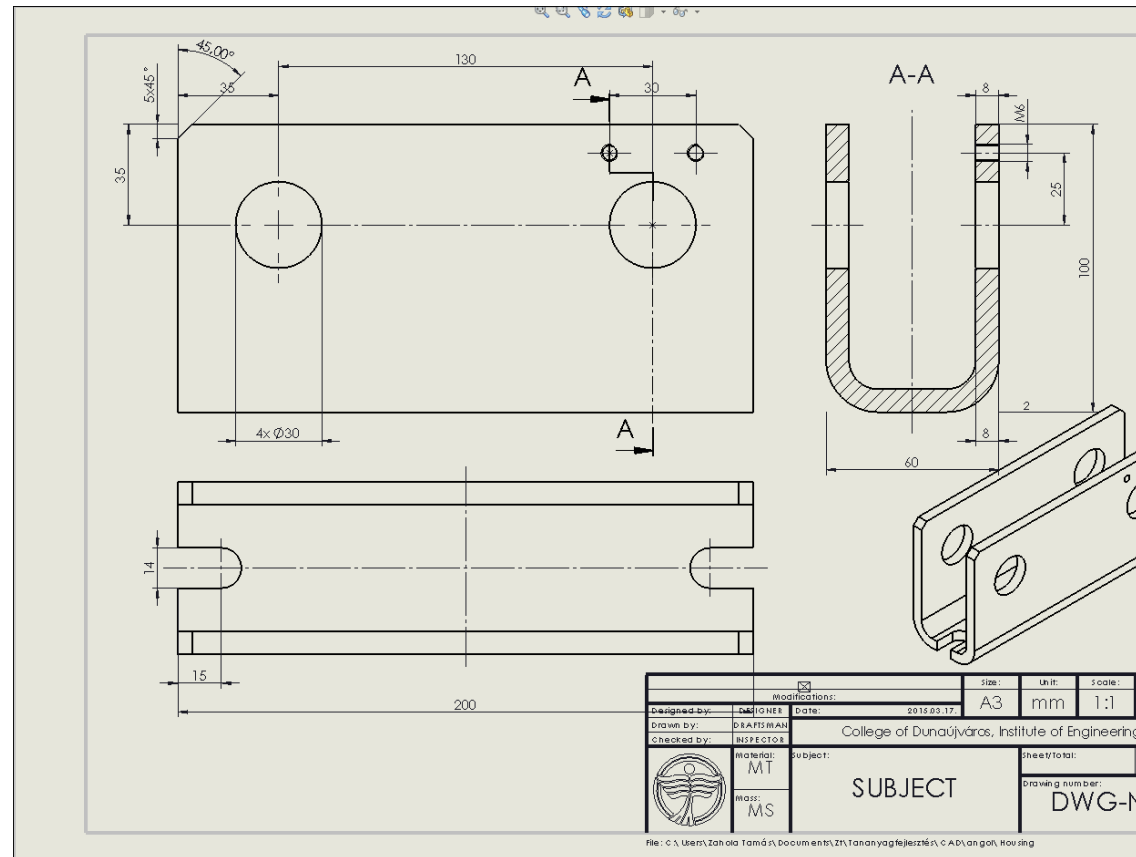


Fig2

11.6 Editing the dimensions

The imported dimensions must be thoroughly reviewed. Bring the section view into focus (*Fig1*)!

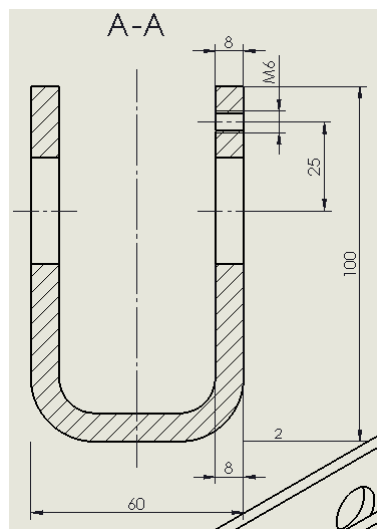


Fig1

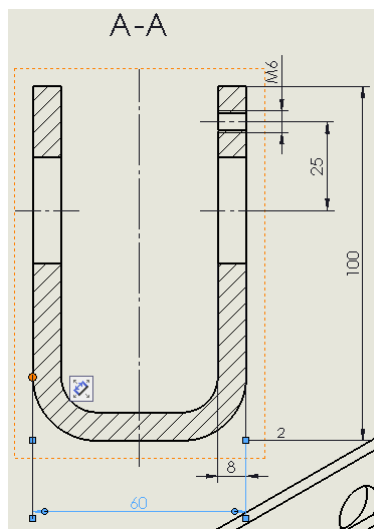


Fig2

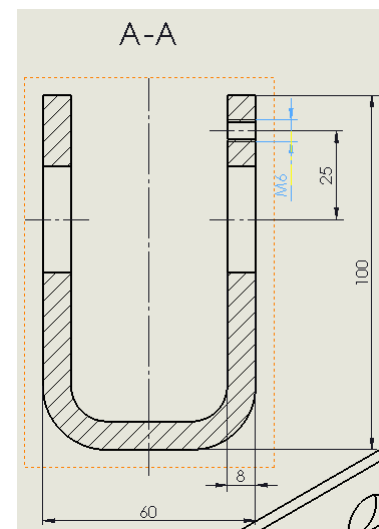


Fig3

- An 8 mm length is defined twice. One of these dimensions means the wall thickness; the other one is the depth of the threaded hole which is not necessary to prescribe. Delete the upper dimension!
- The extension lines of the width and the height start “in the air”. This happened because these dimensions had been defined when the outer fillet of the part was not present. Click on the starting point of the extension line and drag it to the outline of the sectional view (*Fig2*). Do the same for the remaining extension lines!

- A number (2) is placed at the lower right corner of the sectional view. This is the number of the instances of the Linear Pattern which created the copied large holes. Delete the number!
- The dimension line of the M6 dimension is intersected by an extension line. Drag the dimension text to the opposite side of its extension lines (Fig3)!
- There is no dimension for the bending radius. Use the Smart Dimension tool to supply the missing dimension (Fig4)! Click on the quarter circle of the fillet then place the dimension number. Leave the dimension selected! Go to the Leaders tab of the Dimension dialog and press the Inside button to place the dimension arrow inside the dimensioned arc (Fig5)!

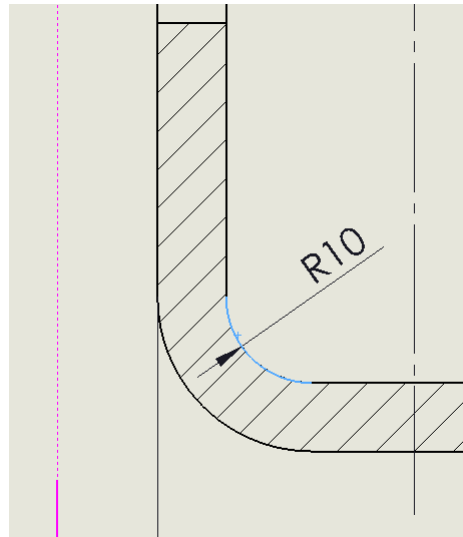


Fig4

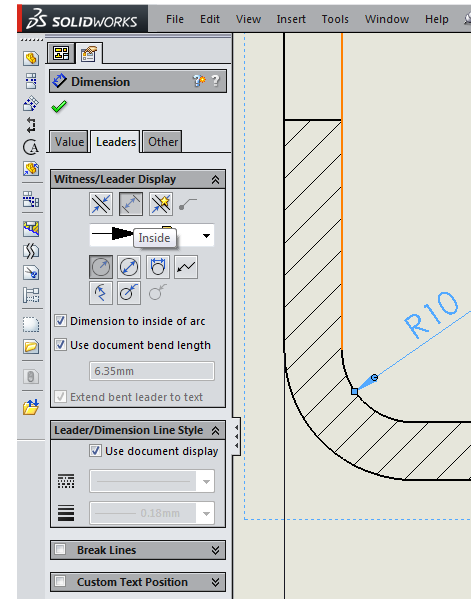


Fig5

- It would be more logical to define the vertical placement of the M6 threaded hole on the main view. This way the dimensions of hole placements would be grouped on the main view. Press *Shift* on the keyboard and drag the 25 mm dimension from the sectional view to the main view (Fig6)! Press *Esc* to deselect the dimension!

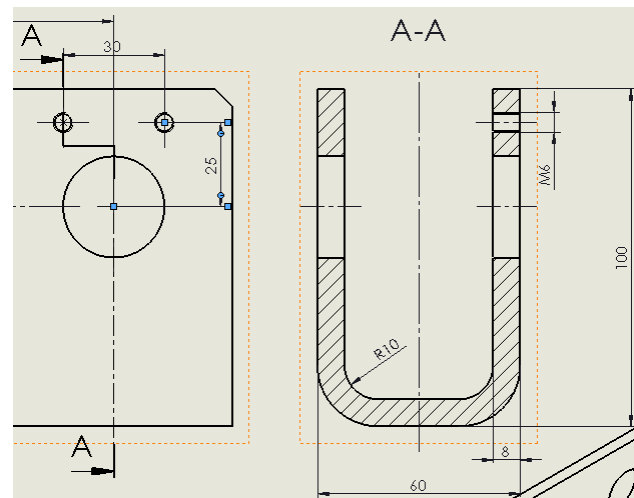


Fig6

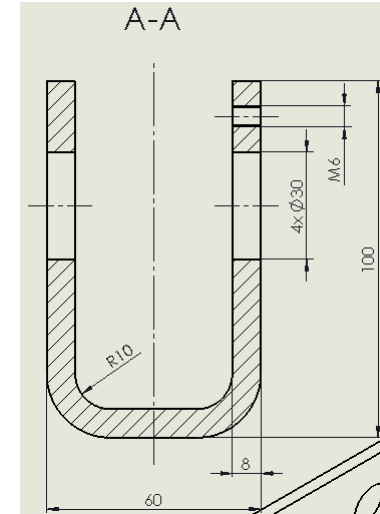


Fig7

– Collect the hole diameters on the sectional view! Shift-drag the 4xØ30 dimension from the main view to the sectional view! Arrange the dimensions of the sectional view by dragging (Fig7)!

The sectional view looks better. Focus on the main view!

- Delete the 45,00° angular dimension. This dimension is not necessary because the 5x45° text contains the same information.
- The extension line of the 130 mm dimension intersects the dimension line of the 30 mm dimension; what's more the extension line crosses the dimension number which is a serious error. Drag the 35 mm and the 130 mm dimensions to the lower side of the main view. Take care to make the dimensionlines collinear. Drag the complete main view upward if it is necessary (Fig8).
- To gain some vertical space, drag the 5x45° dimension text to the opposite side of its extension lines.

- Drag the 30 mm and the 25 mm dimension texts away from the outline of the main view!
- Finally check every extension line and drag its starting point onto the outline of the part.

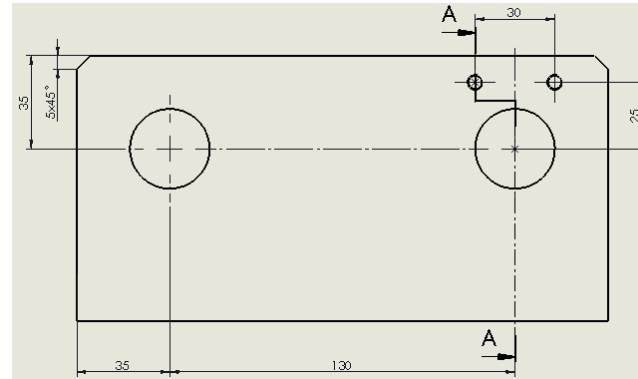


Fig8

The dimension network of the main view is satisfying (Fig8).

The 200 mm dimension on the projected view interferes with the title block. Drag the dimension to the opposite side of the projected view then drag the projected view slightly downward. Don't forget to reposition the starting points of the extension lines.

The drawing content of the Housing is completed (Fig9).

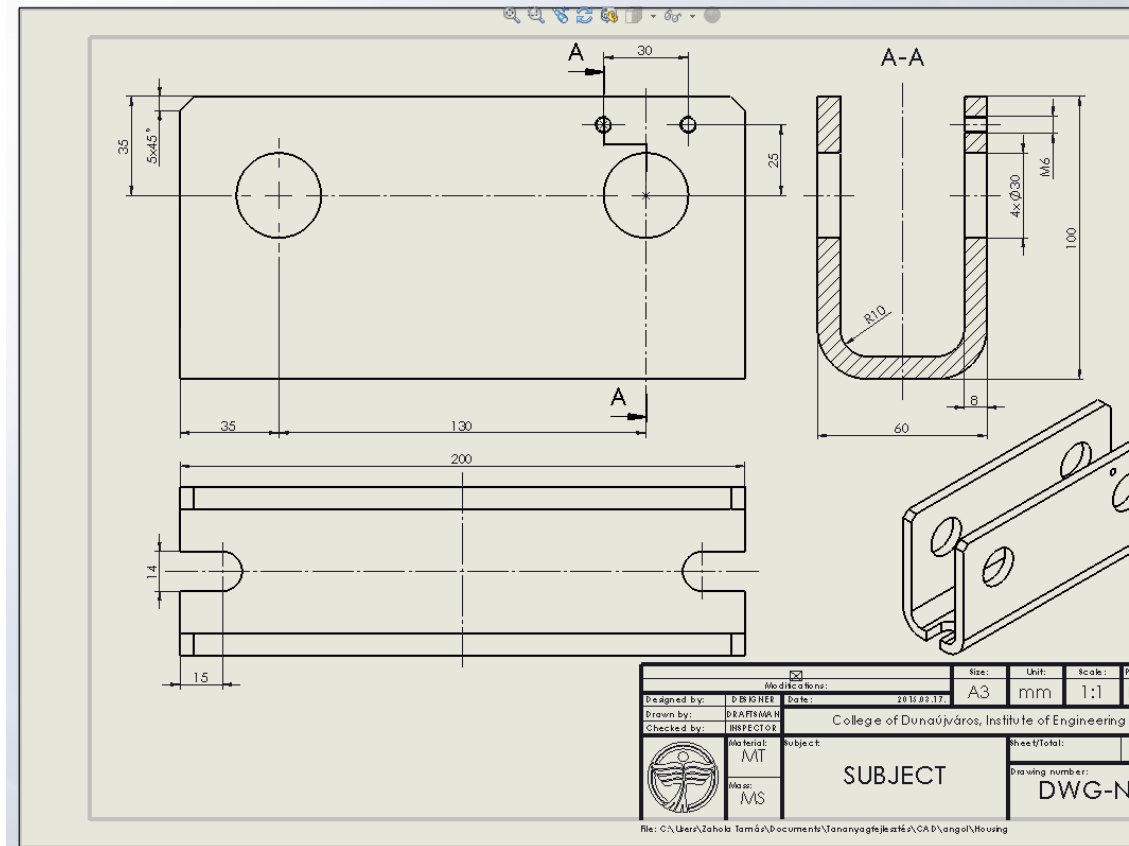


Fig9

11.7 Editing the drawing sheet

Modifications:		Size:	Unit:	Scale:	Projection:
Designed by:	DESIGNER	Date:	2015.03.17.	A3	mm 1:1
Drawn by:	DRAFTSMAN	College of Dunaújváros, Institute of Engineering			
Checked by:	INSPECTOR	Subject: SUBJECT			
	Material:	MT	Sheet/Total:		1/1
	Mass:	MS	Drawing number:		DWG-NR

File: C:\Users\Zahola Tamás\Documents\Tananyagfejlesztés\CAD\angol\Housing

Fig1

Edit the contents of the title block! Double-click on the title block to show its editable fields (Fig1).

Click in the highlighted fields to overwrite their content (Fig2).

Modifications:		Size:	Unit:	Scale:	Projection:
Designed by:	T. Zahola	Date:	2015.03.17.	A3	mm 1:1
Drawn by:	T. Zahola	College of Dunaújváros, Institute of Engineering			
Checked by:	G. Ladányi	Subject: HOUSING			
	Material:	S235	Sheet/Total:		1/1
	Mass:	2,7 kg	Drawing number:		CAD-15-02/001

File: C:\Users\Zahola Tamás\Documents\Tananyagfejlesztés\CAD\angol\Housing

Fig2

The drawing number is too long. To fit the drawing number in the corresponding field the text size must be modified. The elements of the border and the title block belong to the format of the drawing sheet so we have to edit the sheet format.

Right-click on an empty area of the drawing sheet. Select the *Edit Sheet Format* command from the pop-up menu. The content of the sheet disappears; only the elements of the sheet format are visible (Fig3).

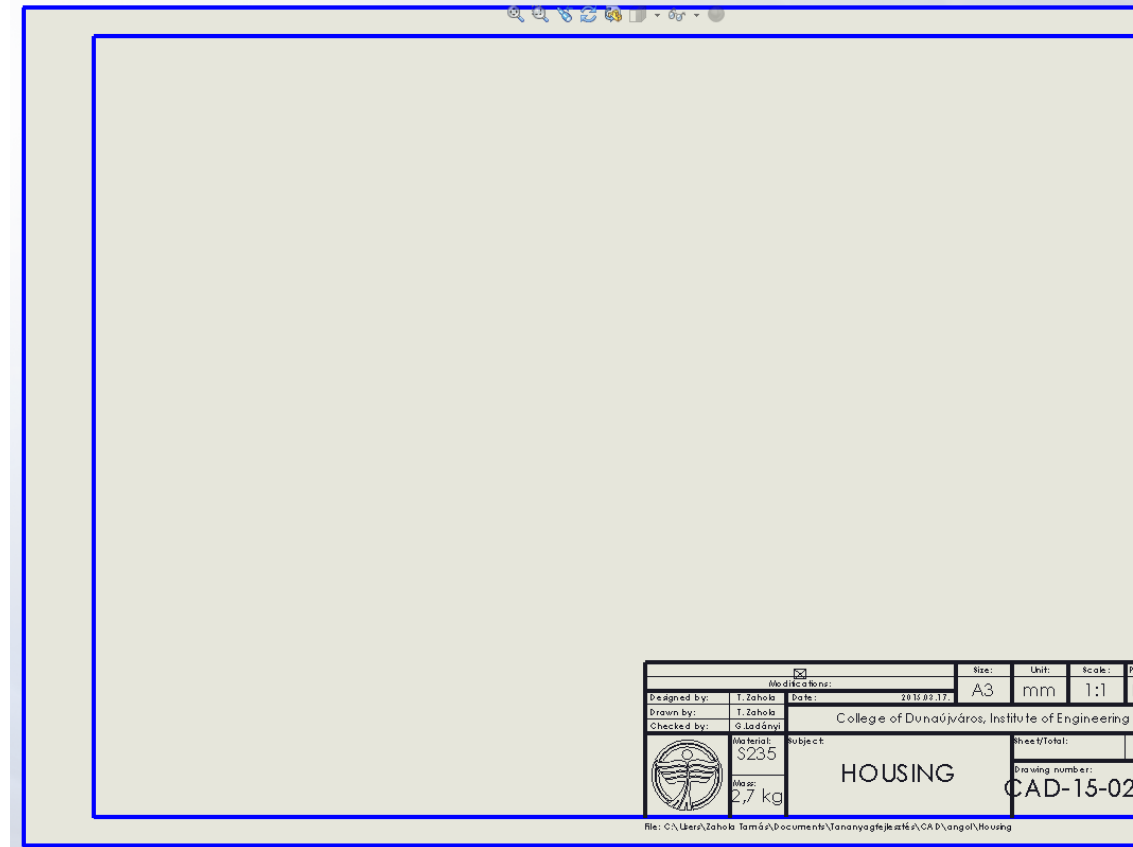


Fig3

Click on the drawing number and press the Font... button on the Note dialog (Fig4).

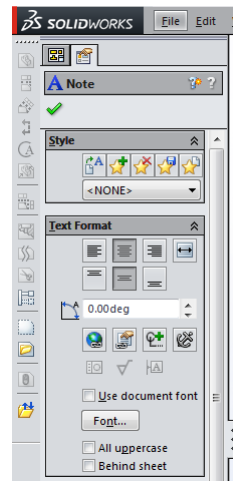


Fig4

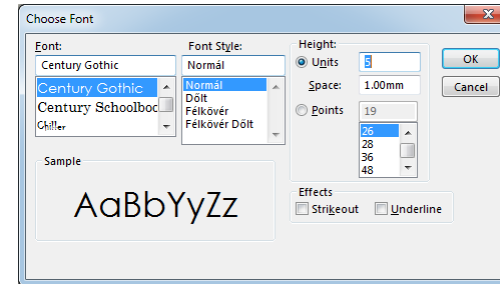


Fig5

Choose a different standard text size and type it into the *Height* field (Fig5) then press OK! The drawing number is short enough now.

Right-click on an empty area of the sheet and select Edit Sheet from the pop-up menu to return to the contents of the sheet.

The drawing of the Housing is completed (Fig6).

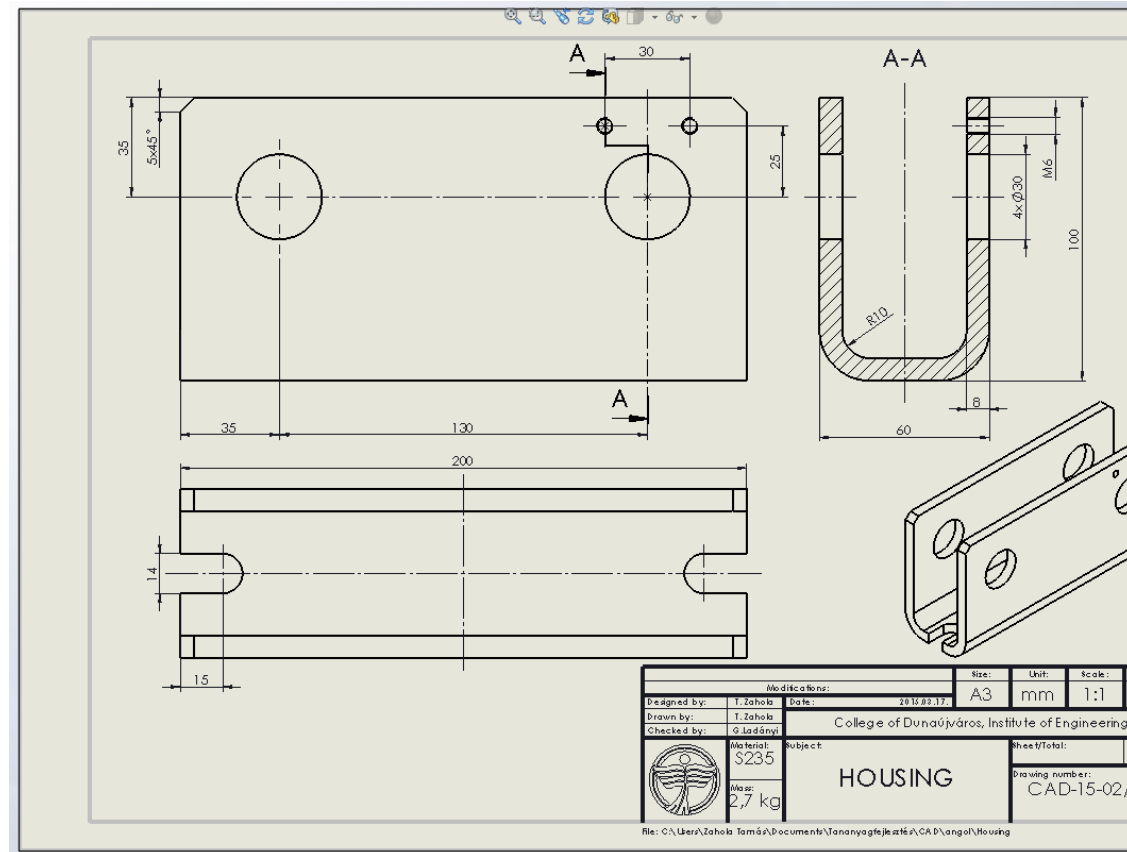


Fig6

The drawing document can be downloaded from the following link: [Housing.SLDDRW](#) The printable version can be downloaded from the following link: [Housing.pdf](#)

11.8 Creating a detail view

The most frequently used drawing techniques were demonstrated in the previous subchapters through the example of the drawing of the *Housing*. In the next subchapters the drawing of *Shaft II* will be created to demonstrate the creation of some less frequent kind of projections. These engineering projections are mentioned in the Hungarian drawing standards.

Create a new drawing document based on the *DF-A4-portrait* template! Place the *Front view* of the part onto the sheet as main projection! Set the default scale of the sheet to 1:1 (*Fig1*).

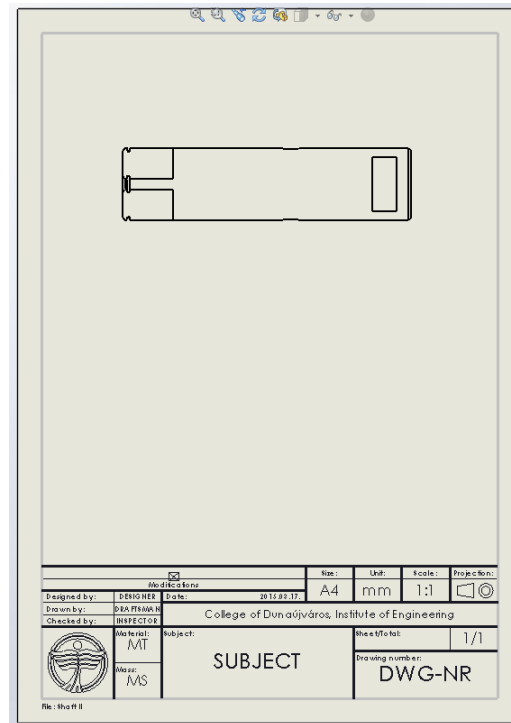


Fig1

The groove on the left end of the part is too small to describe, to read and to dimension – a detail view is necessary. Sketch the detail circle first! This element is not necessarily a circle; it can be an oval or any simple shape.

Sketch an oval shape onto the view of the part around the groove (Fig2)!

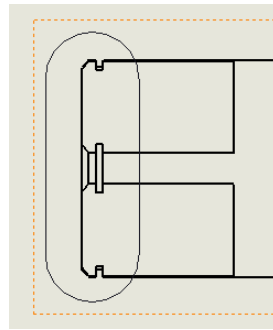


Fig2

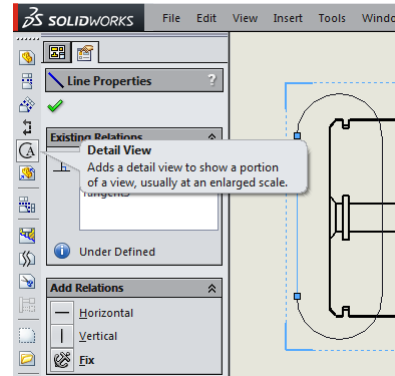


Fig3

Select one of the sketched entities and click on the *Detail View* tool in the *Drawing* toolbar (Fig3).

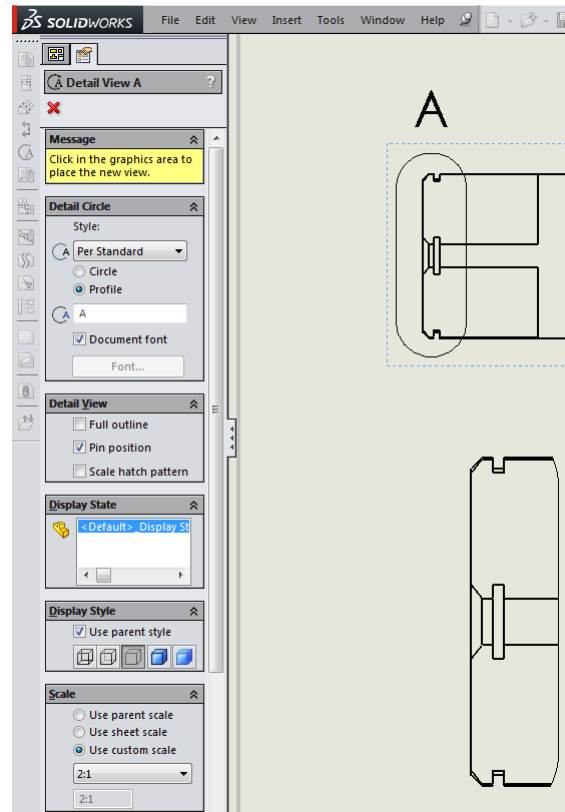


Fig4

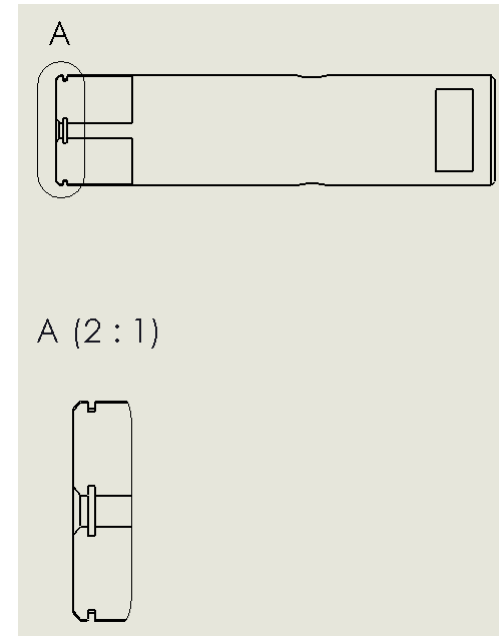


Fig5

Set up the identifier of the *Detail Circle* (A) and the *Scale* in the *Detail View* dialog then click on the sheet to place the new view (Fig4). Drag the label of the *Detail View* a bit closer to the outline of the view (Fig5). The *Detail View* is completed.

Remark:

According to the current Hungarian standards the border of the detail view must be drawn by thin, continuous freehand-lines or by thin, continuous ziz-zag lines – *SolidWorks* does not support this regulation.

11.9 Creating a breakout

To describe the radial hole of the part typically a particular section (breakout) can be used. Click on the *Broken-out Section* tool in the *Drawing* toolbar (Fig1)!

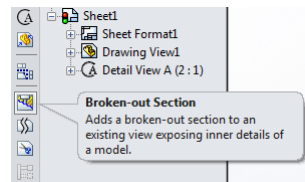


Fig1

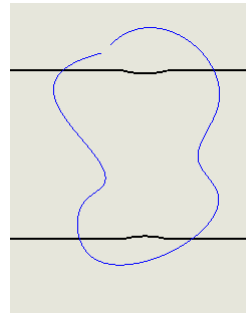


Fig2

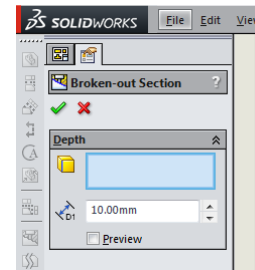


Fig3

The border of the broken-out are should be a freehand line – *SolidWorks* allows sketching a *Spline* around the radial hole. Define a few control points to sketch the *Spline* (Fig2). The first and the last control points must be coincident to create a closed curve.

After the *Spline* gets closed *SolidWorks* asks for the depth of the breakout. The necessary *Depth* can be defined numerically in the *Depth* field of the *Broken-out Section* dialog (Fig3) or graphically by selecting an edge of the part in the desired depth. Click on the *3D Drawing View* tool in the View (Heads-Up) toolbar (Fig4).

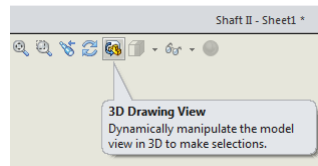


Fig4

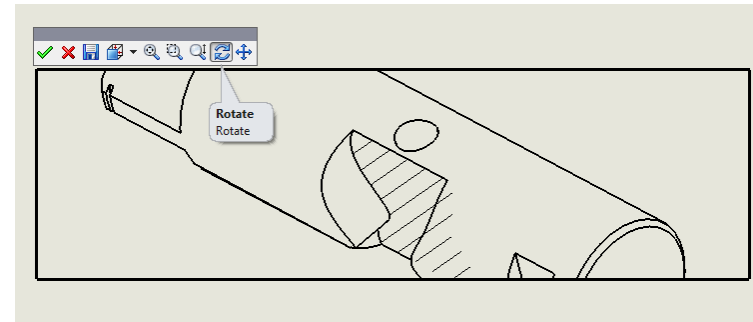


Fig5

This tool allows you to temporarily change the viewing direction of the main view. Use the *Rotate* tool to modify the viewing direction (Fig5). Press *Esc* to stop rotation and select the Silhouette Edge of the radial hole according to the original viewing direction (Fig6).

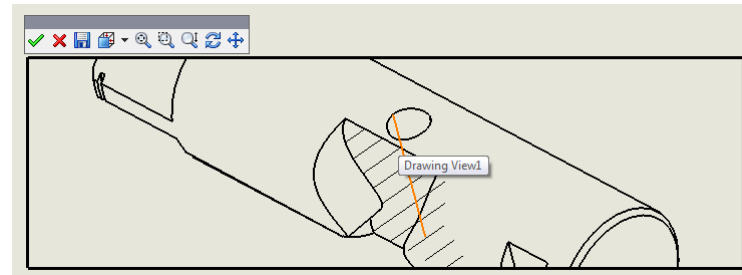


Fig6

Depress the *3D Drawing View* button to return to the original viewing direction then press *OK*. The particular section (breakout) is completed (*Fig7*).

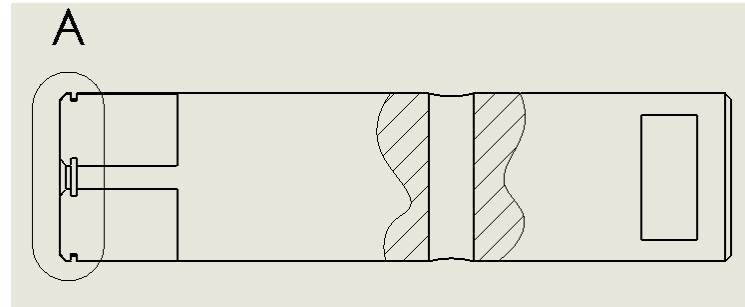


Fig7

11.10 Creating a shortened view

The projections of long, prismatic parts can be shortened by breaking out detailless regions of the projection.

Select the Break tool in the *Drawing* toolbar (*Fig1*)! Click on the view of the part and set up the *Broken View* dialog according to (*Fig2*).

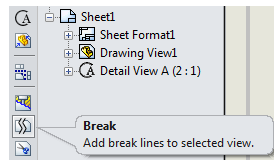


Fig1

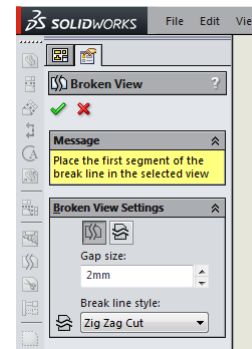
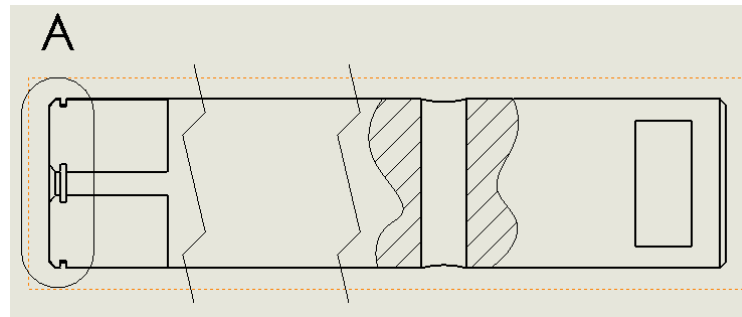
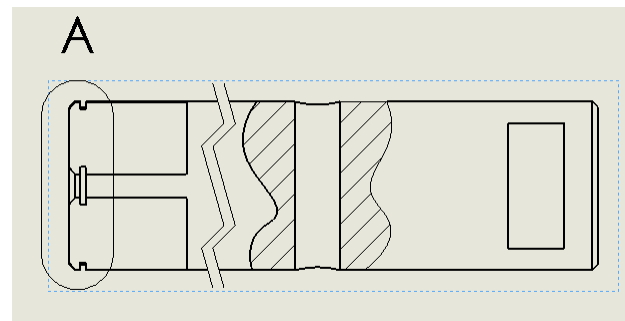


Fig2

Place two break lines to mark the region of the part to be removed (*Fig3*) then click OK.

*Fig3*

The view of the part is shortened by keeping the Gap Size between the break lines (*Fig4*).

*Fig4*

11.11 Creating a pure section

Pure sections are identical to simple sections except only the sectioned surface of the part is described; the elements behind the section plane are ignored. Create a pure section to describe the square shape of the left end of the part! Sketch a vertical *Line* onto the view of the part across the left ending (*Fig1*).

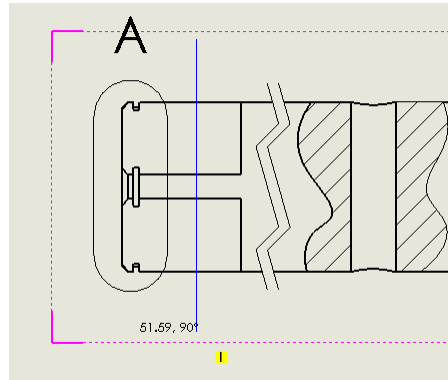


Fig1

Press *Esc* to stop sketching and select the sketched *Line*. Use the *Section View* tool.

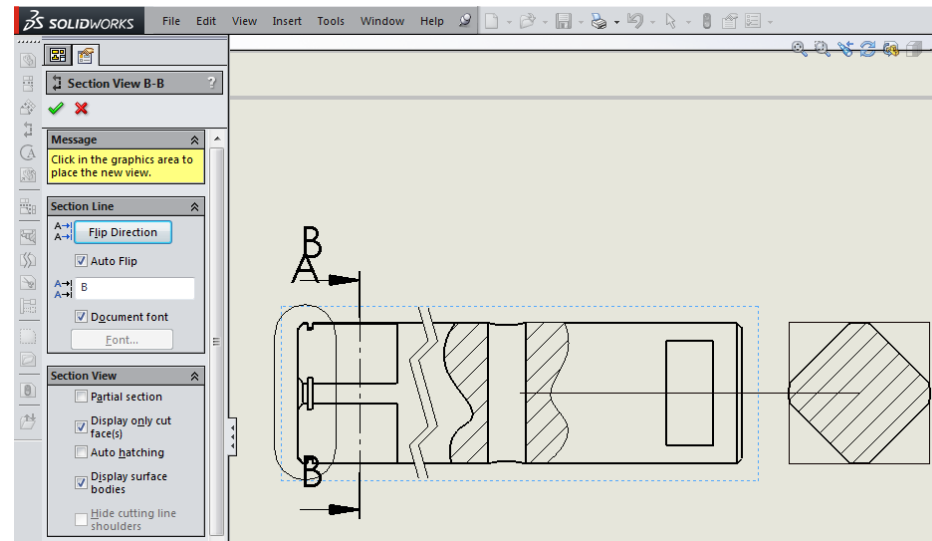


Fig2

Check the *Display only cut face(s)* checkbox in the *Section View* group of the *Section View* dialog to create a pure section instead of a sectional view (Fig2). Place the new projection. Drag the endings of the section line away from the outline of the view if it is necessary.

The pure section is placed onto the sheet keeping the arrangement with the parent view. Because the section line and the pure section are identified, the alignment is not necessary. Right-click on the pure section, and select the *Alignment/Break Alignment* command from the pop-up menu. After the alignment has been broken the pure section can be dragged anywhere on the sheet (Fig3).

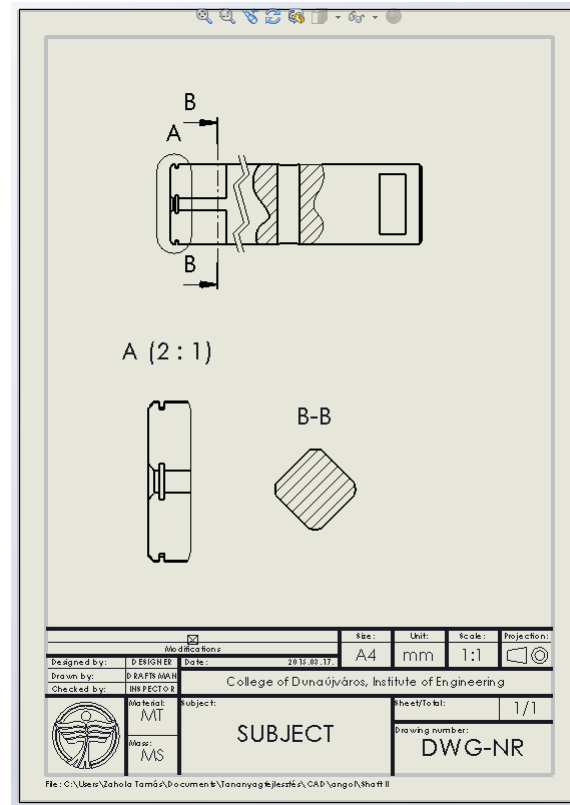


Fig3

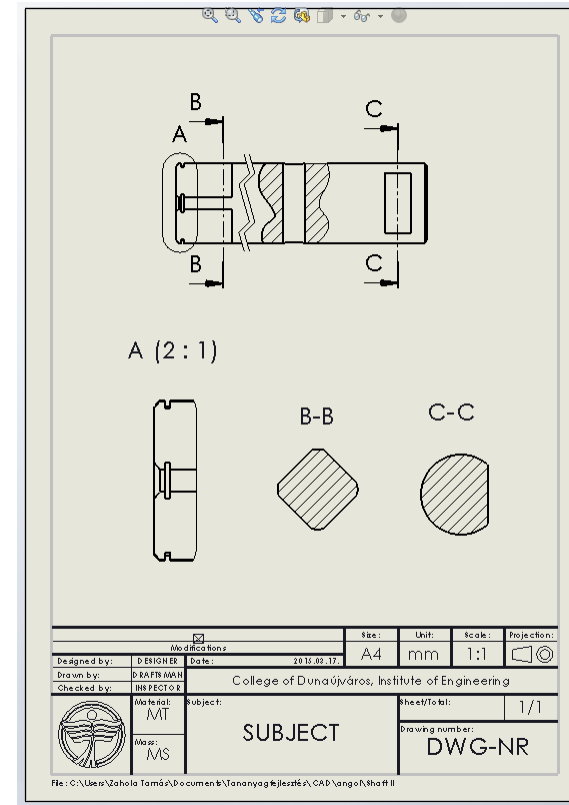


Fig4

Following the previous steps another pure section can be created to describe the cross- section of the part at the slot (Fig4).

Make the drawing complete by inserting the necessary centerlines, building the dimension network and filling the title block (Fig5)!

The drawing document can be downloaded from the following link: [Shaft II.SLDDRW](#)

The printable version can be downloaded from the following link: [Shaft II.pdf](#)

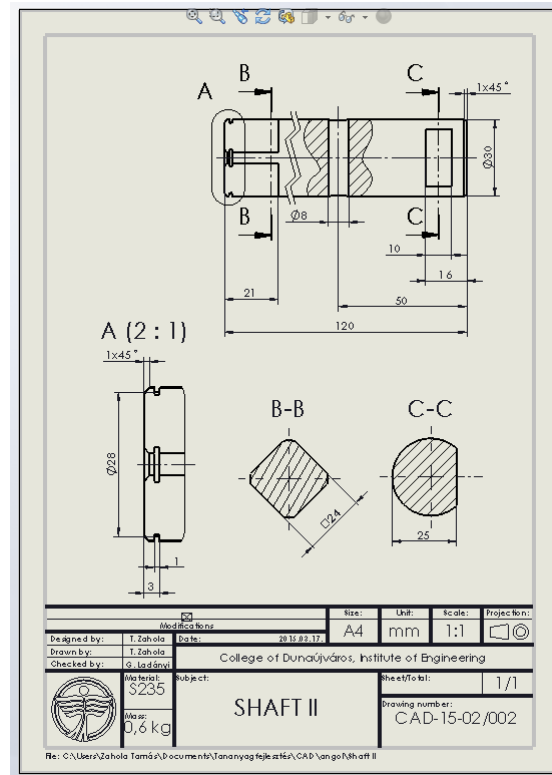


Fig5

12. fejezet

12.1

Create the engineering drawings of the remaining elements of the ratchet mechanism!

12.2

Download the compressed file from the following link and unzip its contents: [1396.zip](#)

Create the engineering drawings of the extracted parts!

12.3

Download the compressed file from the following link and unzip its contents:

[Qldham.zip](#)

Create the engineering drawings of the extracted parts!

13. fejezet

13.1 Creating drawings of assemblies

The topic of this chapter is the creation of assembly drawings. Assembly drawings give information about units which consists of several components. Each of the components has its own part drawing so an assembly drawing shows what parts are present in the unit; how do they connected to each other; and what are the characteristic dimensions of the unit.

CREATING AN OVERVIEW OF AN ASSEMBLY

Create the assembly drawing of the previously modeled ratchet mechanism! The assembly document can be downloaded from the following link:

[Ratchet mechanism.zip](#)

All of the drawing tools can be used to create the projections what are necessary to show every component in the assembly.

Create a new *Drawing* document using the *DF-A4-Portrait* drawing template.



309

13.2 Creating an item list (Bill of Material)

Assembly drawings contain an item list. The item list is a table which transmits information about the components of the described assembly. Item lists are called *Bill of Material [BOM]* in SolidWorks.

Insert a *Bill of Material* into the drawing! Select the Insert/Tables/Bill of Materials... command from the Main drop-down menu. Click on the view of the assembly to specify the source of the item list. *Fill the Bill of Materials* dialog according to Fig1.

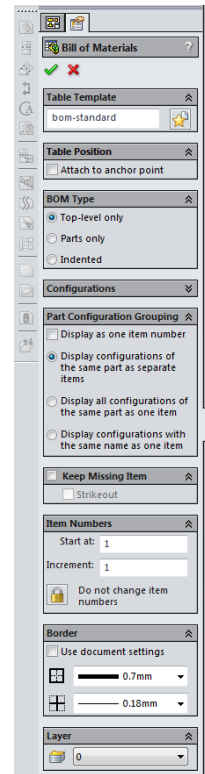


Fig1

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1		Housing	1
2		Shaft I	1
3		Fixture	1
4		Pawl	1
5		Ratchet wheel assembly	1
6		Half Spring I	1
7		Half Spring II	1
8		BOLT: ISO 1207 - M6 x 12 - A2-2	2

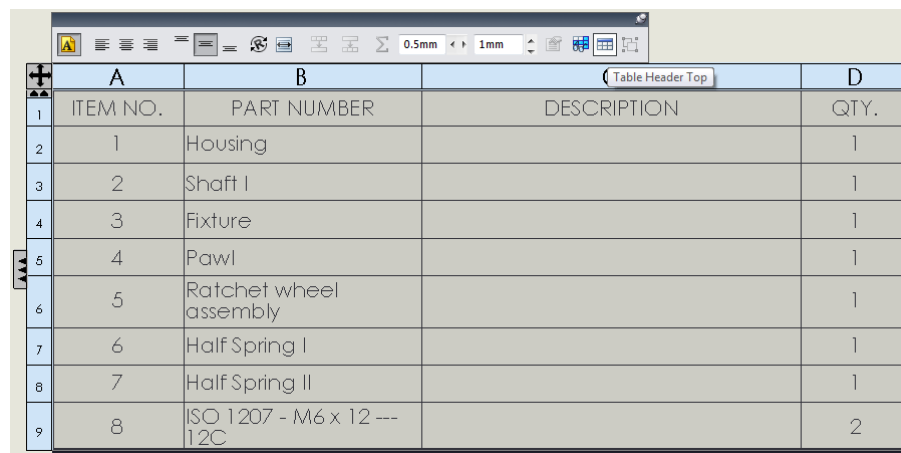
Designed by: DEZSŐ NÉMETI		Date: 2016.03.18	Size: A4	Unit: mm	Scale: 1:2	Projection: 1st
College of Dunaujváros, Institute of Engineering						
	Material: MT	Subject: RATCHET MECHANISM	Sheet/Total: 1/1	Drawing number: DWG-NR		
	Material: MS					

Fig2

The assembly contains a subassembly so it is important to set the *BOM Type* radio button correctly. It is set to *Top-level only* so the *Ratchet wheel assembly* will appear as one item in the list. The assembly contains configured parts (the screws) so the *Part Configuration Grouping* radio button is also important. The different configurations of the same part model would appear as separate items in the list.

Press *OK* then click on the drawing sheet to place the *Bill of Material* (Fig2)!

Move the pointer over the *Bill of Material* and click on the *Arrow Cross* in the upper left corner of the table to select the whole table then click on the *Table Header Top* tool in the pop-up toolbar to change it to *Table Header Bottom* (Fig3). The header of the table is the lowermost row now; item numbers are listed bottom-up.



	A	B	C	D
	ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	1	Housing		1
2	2	Shaft I		1
3	3	Fixture		1
4	4	Pawl		1
5	5	Ratchet wheel assembly		1
6	6	Half Spring I		1
7	7	Half Spring II		1
8	8	ISO 1207 - M6 x 12 --- 12C		2

Fig3

Because the table is selected its properties appear on the left. The *Bill of Materials* dialog contains the previously not displayed *Table Position* group (Fig4).

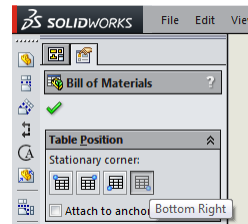


Fig4

Set the *Stationary corner* of the table to *Bottom Right*! Click on the Arrow Cross and drag the table so as to be aligned with the upper edge of the title block and the right borderline (Fig5).

	A	B	C	D
1	8	ISO 1207 - M6 x 12 --- 12C		2
2	7	Half Spring II		1
3	6	Half Spring I		1
4	5	Ratchet wheel assembly		1
5	4	Pawl		1
6	3	Fixture		1
7	2	Shaft I		1
8	1	Housing		1
9	ITEM NO.	PART NUMBER	DESCRIPTION	QTY.

☒ Modifications		Size: A4	Unit: mm	Scale: 1:2	Projection: 
Designed by: DESIGNER	Date: 2015.03.18.	College of Dunaújváros, Institute of Engineering			
Drawn by: DRAFTSMAN					
Checked by: INSPECTOR					
	Material: MT	Subject: SUBJECT	Sheet/Total: 1/1		
	Mass: MS		Drawing number: DWG-NR		

File: Ratchet mechanism

Fig5

13.3 Ballooning

The components of the assembly must be identified on the drawing with leader containing numbers according to the item list. These leaders are called Balloons in *SolidWorks*. Select the *Insert/Annotations/Auto Balloon...* command in the *Main* drop-down menu! Click on the Isometric view of the mechanism to mark it as the base of the balloon insertion.

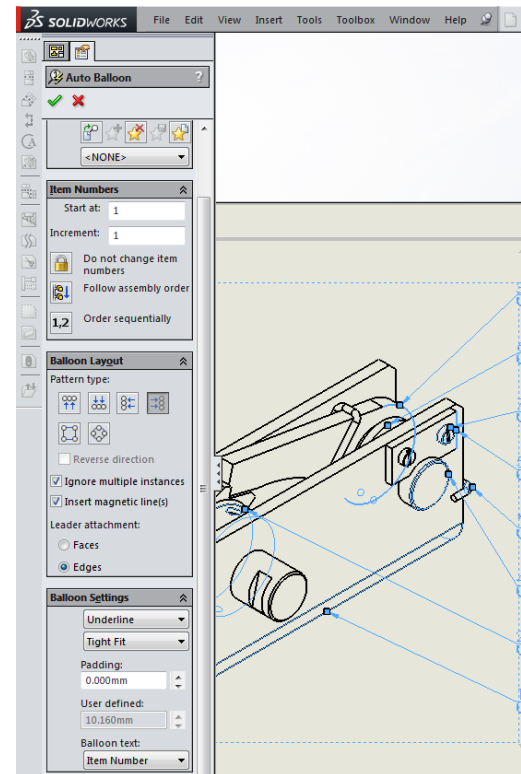


Fig1

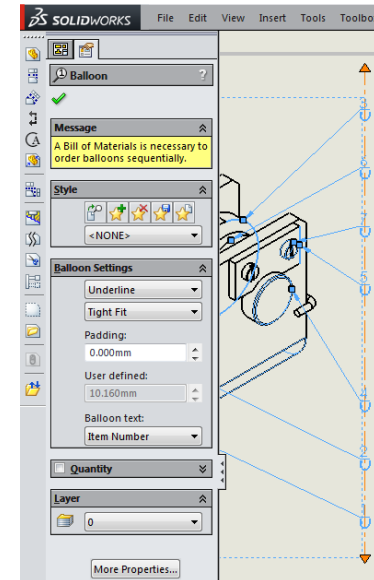


Fig2

Fill the *Auto Balloon* dialog according to Fig1 then press OK. Delete the leader of *Half Spring* !/

The balloons need some cosmetics. Select all of them and press the *More Properties...* button on the *Balloon* dialog (Fig2).

Remove the mark from the *Use document font* checkbox in the *Text Format* group of the *Note* dialog then press the *Font...* button (Fig3).

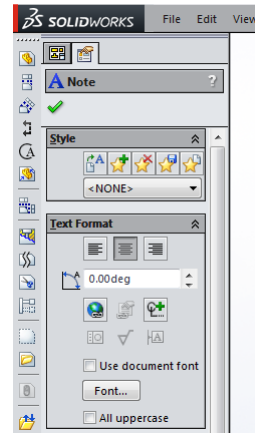


Fig3

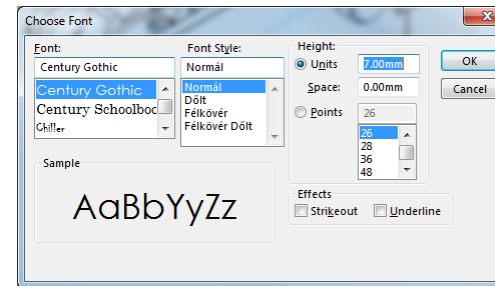


Fig4

Set the text size of the balloons to 7 mm in the *Height* field of the *Choose Font* dialog (Fig4).

Drag the starting points of the leaders and also the balloons to arrange them aesthetically (Fig5).

Notice that the ending symbol changes from an arrow to a dot as you drag the starting point of the leader from an edge to a face.

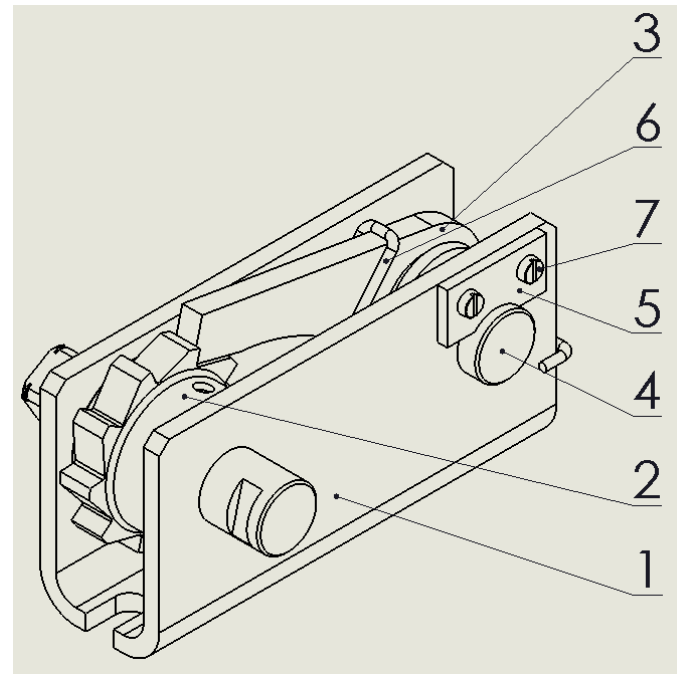
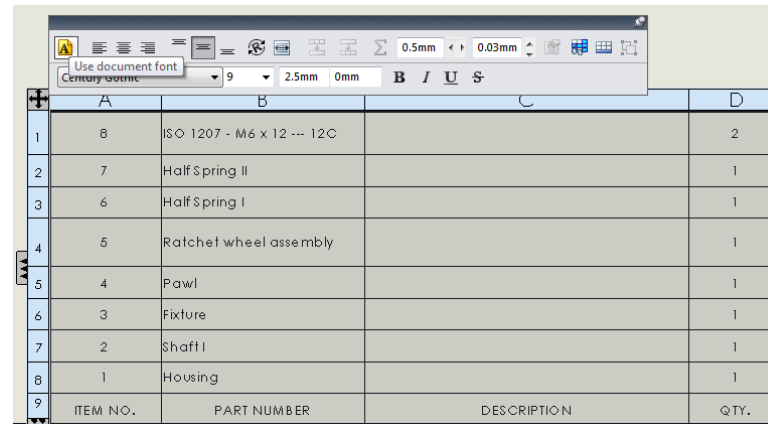


Fig5

13.4 Editing the item list

Click on the Arrow Cross to select the complete *Bill of Materials*! Depress the *Use document font* button (*Fig1*) on the local toolbar! Set the text size of the table to 2,5 mm!



	A	B	C	D
1	8	ISO 1207 - M6 x 12 --- 12C		2
2	7	Half Spring II		1
3	6	Half Spring I		1
4	5	Ratchet wheel assembly		1
5	4	Pawl		1
6	3	Fixture		1
7	2	Shaft I		1
8	1	Housing		1
9	ITEM NO.	PART NUMBER	DESCRIPTION	QTY.

Fig1

Click on the identifier of the last column of the table and drag it so as to make it the second column (*Fig2*).

	A	B	C	D
1	8	2	ISO 1207 - M6 x 12 --- 12C	
2	7	1	Half Spring II	
3	6	1	Half Spring I	
4	5	1	Ratchet wheel assembly	
5	4	1	Pawl	
6	3	1	Fixture	
7	2	1	Shaft I	
8	1	1	Housing	
9	ITEM NO.	QTY.	PART NUMBER	DESCRIPTION

Fig2

Click on the column identifier number of the top row of the table; then press *Shift* on the keyboard and click on the identifier of the next to last row. The entire table is selected except the header (Fig3).

	A	B	C	D
2	8	2	ISO 1207 - M6 x 12 --- 12C	
3	7	1	Half Spring II	
4	6	1	Half Spring I	
5	5	1	Ratchet wheel assembly	
6	4	1	Pawl	
7	3	1	Fixture	
8	2	1	Shaft I	
9	1	1	Housing	
10	ITEM NO.	QTY.	PART NUMBER	DESCRIPTION

Fig3

Right-click on the selected table area and select *Formatting/Row Height* from the pop-up menu! Set the *Row Height* of the selected region to 8 mm (Fig4).

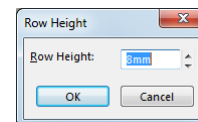


Fig4

Right-click on the selected rows again and select *Formatting/Lock Row Height* from the pop-up menu.

Set the height of the header row to 10 mm. Lock the height of this row too.

Change the font of the header line to **Bold** (Fig5).

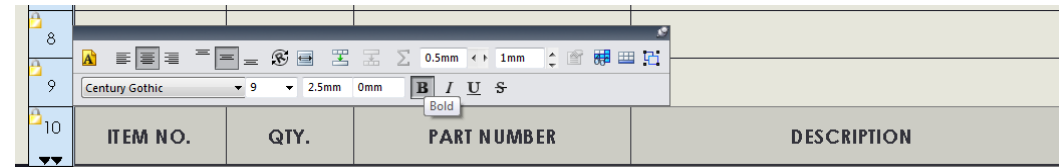


Fig5

Right-click on the identifier of the last column of the table; then select *Insert/Column Right* in the pop-up menu (Fig6).

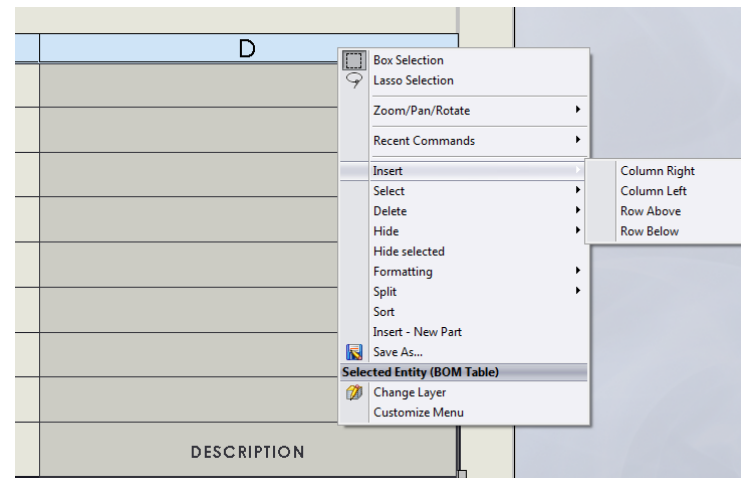


Fig6

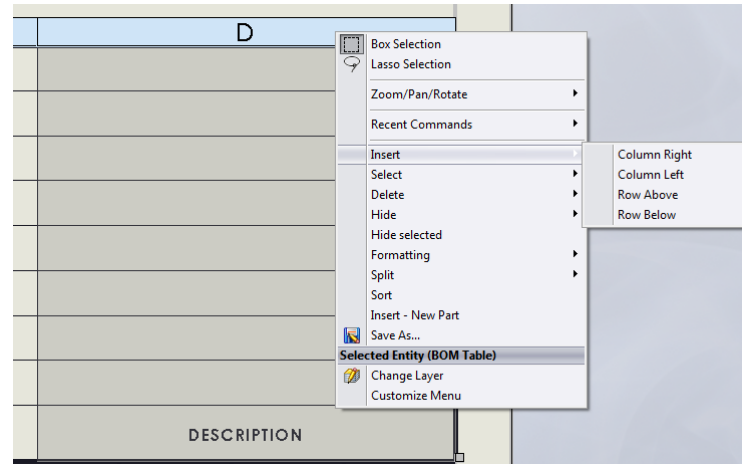


Fig6

Repeat this step to get altogether eight columns in the table. The table is way too wide now. Right-click on the identifier of the first column! Select *Formatting/Column Width* from the pop-up menu. Set the width of the column to 10 mm.

The widths of the remaining rows are 10, 70, 30, 15, 10, 10 and 35 mm. Set up the proper width for each of the columns then lock their width. The width of the title block and the item list is equal (Fig7).

8	2	ISO 1207 - M6 x 12 --- 12C					
7	1	Half Spring II					
6	1	Half Spring I					
5	1	Ratchet wheel assembly					
4	1	Pawl					
3	1	Fixture					
2	1	Shaft I					
1	1	Housing					
ITEM NO.	QTY.	PART NUMBER	DESCRIPTION				
☒ Modifications Designed by: DESIGNER Date: 2015.03.18. Drawn by: DRAFTSMAN Checked by: INSPECTOR			Size: A4	Unit: mm	Scale: 1:2	Projection: 	
 Material: MT Mass: MS			Subject: SUBJECT		Sheet/Total: 1/1 Drawing number: DWG-NR		

File: Ratchet mechanism

Fig7

Double-click on the identifier of the PART NUMBER column to set up the content of the column.

	A	B	C
2	8	2	Screw_ISO
3	7	1	
4	6	1	
5	5	1	
6	4	1	Pawl
7	3	1	Fixture
8	2	1	Shaft I
9	1	1	Housing
10	ITEM NO.	QTY.	SW-File Name(File Name)

Fig8

C	E
Slotted Cheese Head Screw_ISO	
Half Spring II	
Half Spring I	
Ratchet wheel assembly	
Pawl	Default
Fixture	Default
Shaft I	Default
Housing	Default
SW-File Name(File Name)	DESCRIPTION

Fig9

Set the *Column* type to CUSTOM PROPERTY! Select *SW-File Name*(File Name) from the *Property Name* drop-down list (Fig8). Now the file names of the components are listed as designations.

Set the *Column* type of the next column to CUSTOM PROPERTY too! Choose *SWConfiguration Name*(Configuration Name) as *Property Name* (Fig9).

Double-click in each of the cells of the header and overwrite their content according to Fig10. Use *Ctrl-Enter* to insert a new line into the text if it is necessary.

8	2	Slotted Cheese Head Screw_ISO	ISO 1207 - M6 x 12 --- 12C				
7	1	Half Spring II	Default				
6	1	Half Spring I	Default				
5	1	Ratchet wheel assembly	Default				
4	1	Pawl	Default				
3	1	Fixture	Default				
2	1	Shaft I	Default				
1	1	Housing	Default				
Item Nr.	Qty.	Designation	Reference	Material	Wt.	Total Wt.	Comment

Fig10

Two spring halves appear in the item list - one of them is not needed.

Open the *Ratchet mechanism* assembly! Right-click on *Half Spring I* in the *Feature Manager* and select the *Component Properties...* tool in the pop-up toolbar (Fig11)!

Put a mark into the *Exclude from bill of materials* checkbox in the *Component Properties* dialog (Fig12) then press OK.

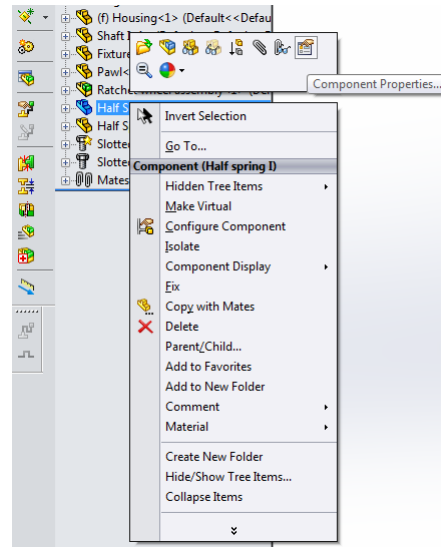


Fig11

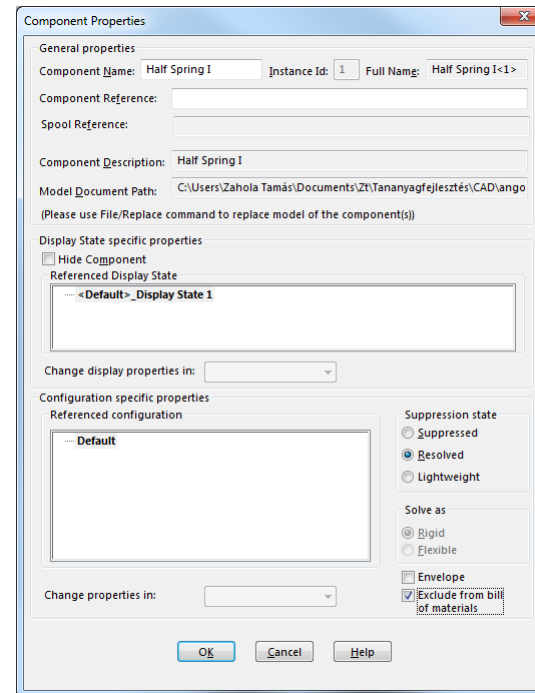


Fig12

As we return to the drawing document we can see that *Half Spring I* disappeared from the item list.

Reorder the rows of the item list by clicking on their identifiers and dragging them into the desired position (Fig13).

7	2	Slotted Cheese Head Screw_ISO	ISO 1207 - M6 x 12 --- 12C				
6	1	Half Spring II	Default				
5	1	Fixture	Default				
4	1	Shaft I	Default				
3	1	Pawl	Default				
2	1	Ratchet wheel assembly	Default				
1	1	Housing	Default				
Item Nr.	Qty.	Designation	Reference	Material	Wt.	Total Wt.	Comment

Fig13

Modify the designation of the 6th item from *Half Spring II* to *Spring!* Double-click in the corresponding field.

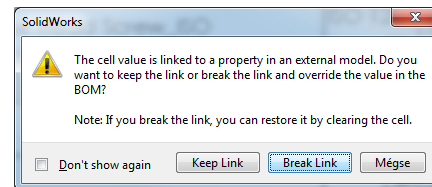


Fig14

SolidWorks asks if we really want to override the value (Fig 14). Press *Break Link* on the message box!

Modify the *Reference* cells to describe the (imaginary) drawing numbers of the nonstandard components.

Fill the remaining of the table (*Fig15*)!

7	2	Slotted Cheese Head Screw	ISO 1207 - M6 x 12 --- 12C	5.6		0,01g	
6	1	Spring	CAD-15-02/005	1.0500	9g	9g	
5	1	Fixture	CAD-15-02/004	S235	50g	50g	
4	1	Shaft I	CAD-15-02/003	S235	0,4kg	0,4kg	
3	1	Pawl	CAD-15-02/002	S235	0,5kg	0,5kg	
2	1	Ratchet wheel assembly	CAD-15-02/010		1,4kg	1,4kg	
1	1	Housing	CAD-15-02/001	S235	2,7kg	2,7kg	
Item Nr.	Qty.	Designation	Reference	Material	Wt.	Total Wt.	Comment

Fig15

Don't forget to fill the title block finally (*Fig16*)!

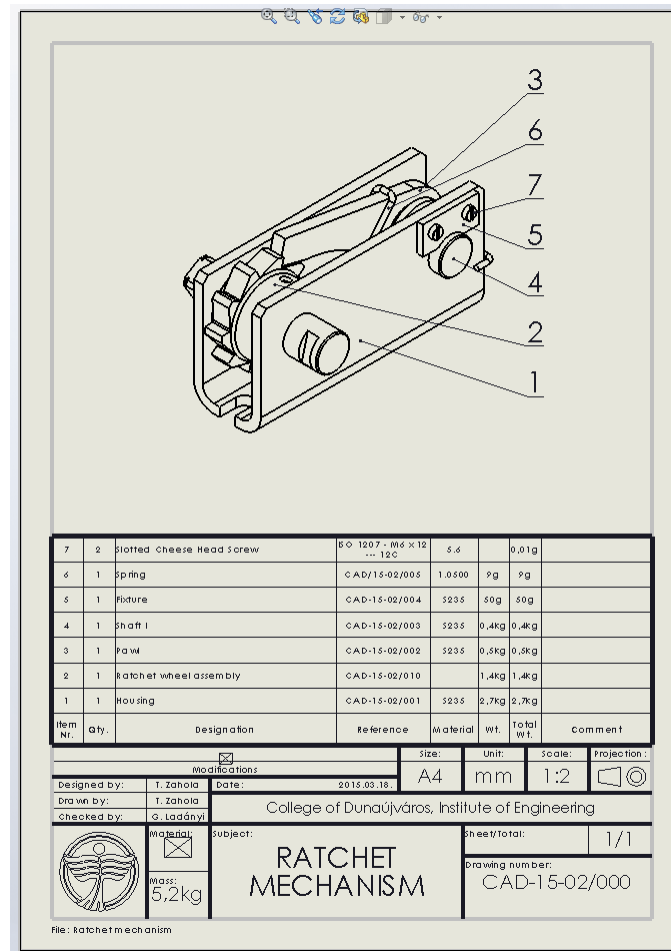


Fig16

The assembly drawing of the mechanism is completed.

The drawing document can be downloaded from the following link:

[Ratchet mechanism.SLDDRW](#)

The printable version can be downloaded from the following link:

[Ratchet mechanism.pdf](#)

14. fejezet

14.1 Practices

PART DRAWINGS

Create the part drawing of the remaining components of the *Ratchet mechanism*!

The part documents can be downloaded from the following links:

[Fixture.SLDPRT](#)

[Pawl.SLDPRT](#)

[Pin.SLDPRT](#)

[Ratchet wheel.SLDPRT](#)

[Shaft I.SLDPRT](#)

[Spring.SLDPRT](#)

ASSEMBLY DRAWINGS

Create the assembly drawing of the *Ratchet wheel assembly*!

The assembly document together with the referenced part documents can be downloaded as a compressed archive from the following link:

[Ratchet wheel assembly.zip](#)

15. fejezet

15.1 Test (Drawings)

To check the students' drawing skills a test is performed. A typical test can be seen below.

Download the part document from the following link:

[Stand.SLDPRT](#)

Recreate the drawing of the part according to *Fig1!*

The printable version of the drawing can be downloaded from the following link:

[Stand.pdf](#)

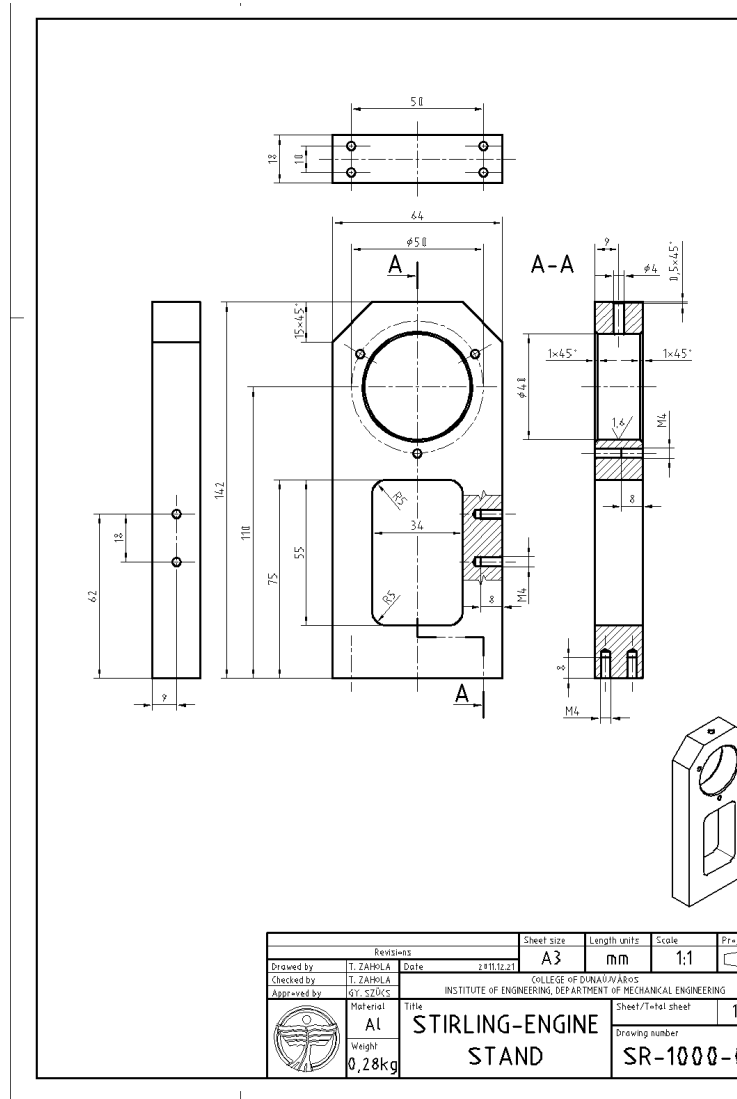


Fig1

16. fejezet

16.1 Parts and drawings

The following links contains engineering drawings of machine parts.

- Create the 3D model of each described part!
- Recreate the drawings of the parts!

Part01.pdf	Part07.pdf	Part13.pdf
Part02.pdf	Part08.pdf	Part14.pdf
Part03.pdf	Part09.pdf	Part15.pdf
Part04.pdf	Part10.pdf	Part16.pdf
Part05.pdf	Part11.pdf	Part17.pdf
Part06.pdf	Part12.pdf	Part18.pdf

16. 2 Parts, assemblies and drawings

The following links contains engineering drawings of units consisting of several machine parts. The parts are joined by welding.

- Create the 3D model of each described part!
- Build the assemblies of the units!
- Recreate the drawing documents!

Ignore the welded joints during the assembly modeling process and the drawing creation!

[Asm01.pdf](#)

[Asm02.pdf](#)

[Asm03.pdf](#)

[Asm04.pdf](#)

[Asm05.pdf](#)

16.3 Synthetical practices

PARTS AND DRAWINGS

The following links contains engineering drawings of machine parts.

- Create the 3D model of each described part!
- Recreate the drawings of the parts!

Part01.pdf	Part07.pdf	Part13.pdf	Part02.pdf	Part08.pdf	Part14.pdf
Part03.pdf	Part09.pdf	Part15.pdf	Part04.pdf	Part10.pdf	Part16.pdf
Part05.pdf	Part11.pdf	Part17.pdf	Part06.pdf	Part12.pdf	Part18.pdf

PARTS, ASSEMBLIES AND DRAWINGS

The following links contains engineering drawings of units consisting of several machine parts. The parts are joined by welding.

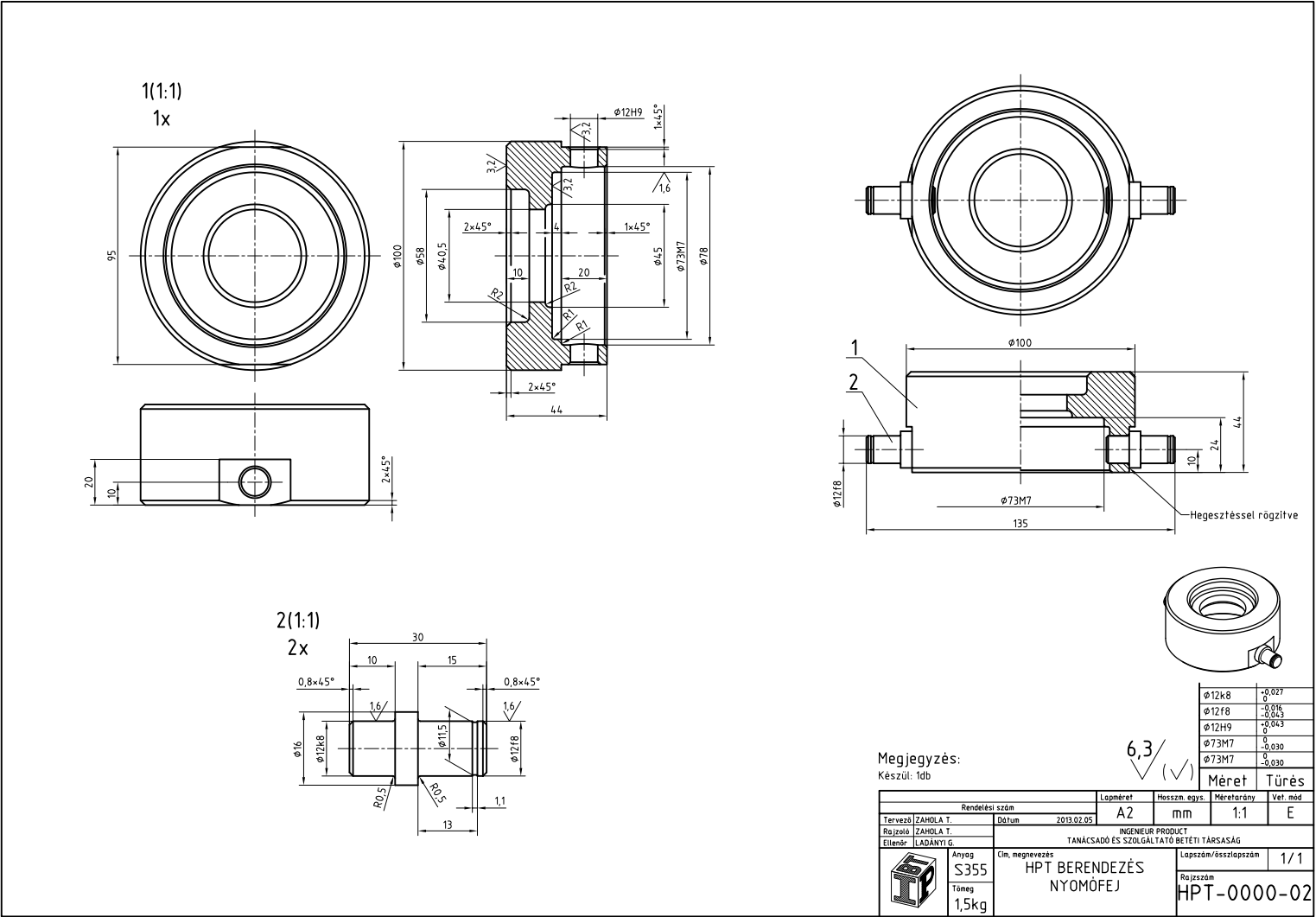
- Create the 3D model of each described part!
- Build the assemblies of the units!
- Recreate the drawing documents!

Ignore the welded joints during the assembly modeling process and the drawing creation!

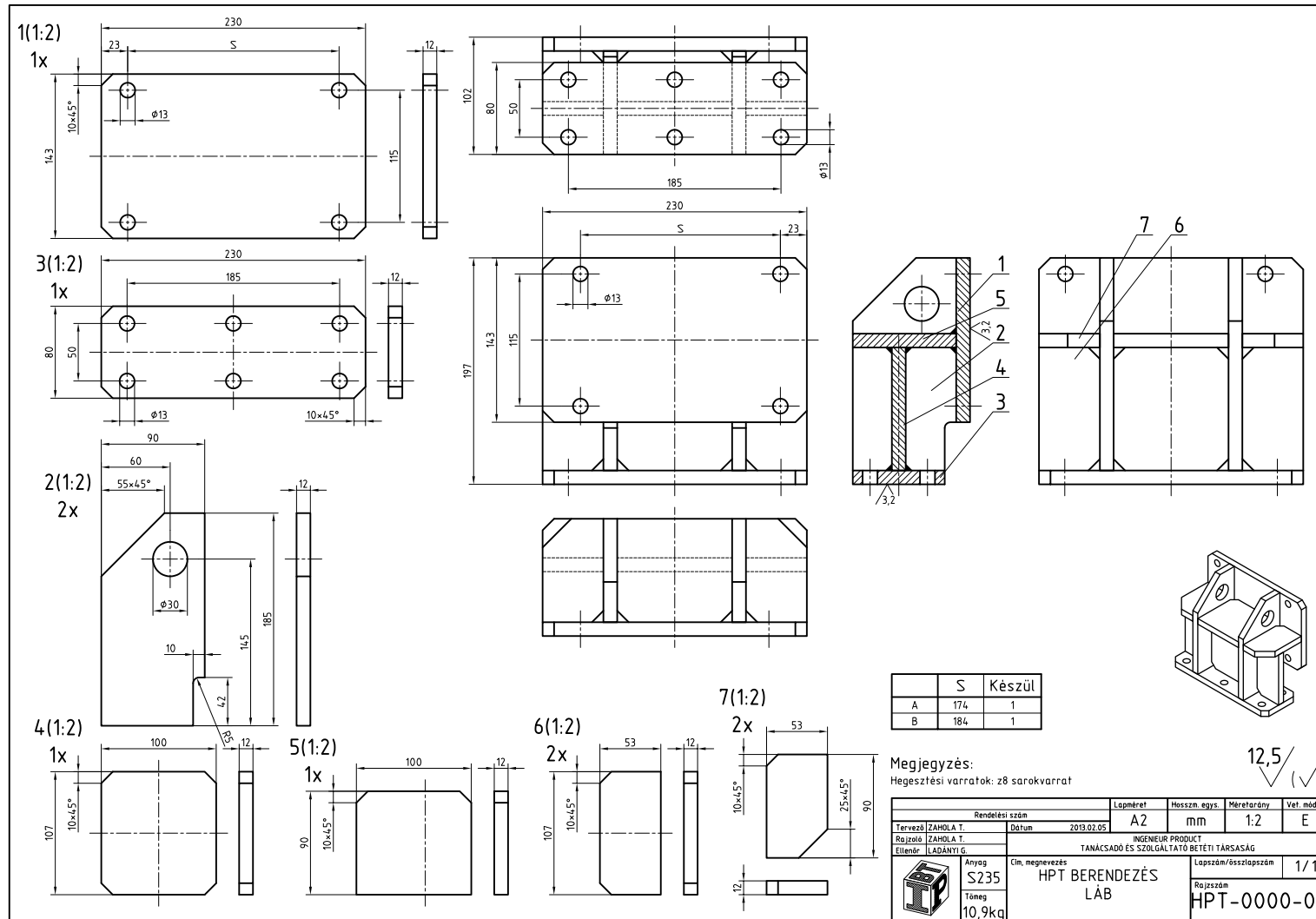
Asm01.pdf	Asm02.pdf	Asm03.pdf	Asm04.pdf	Asm05.pdf
---------------------------	---------------------------	---------------------------	---------------------------	---------------------------

Drawings

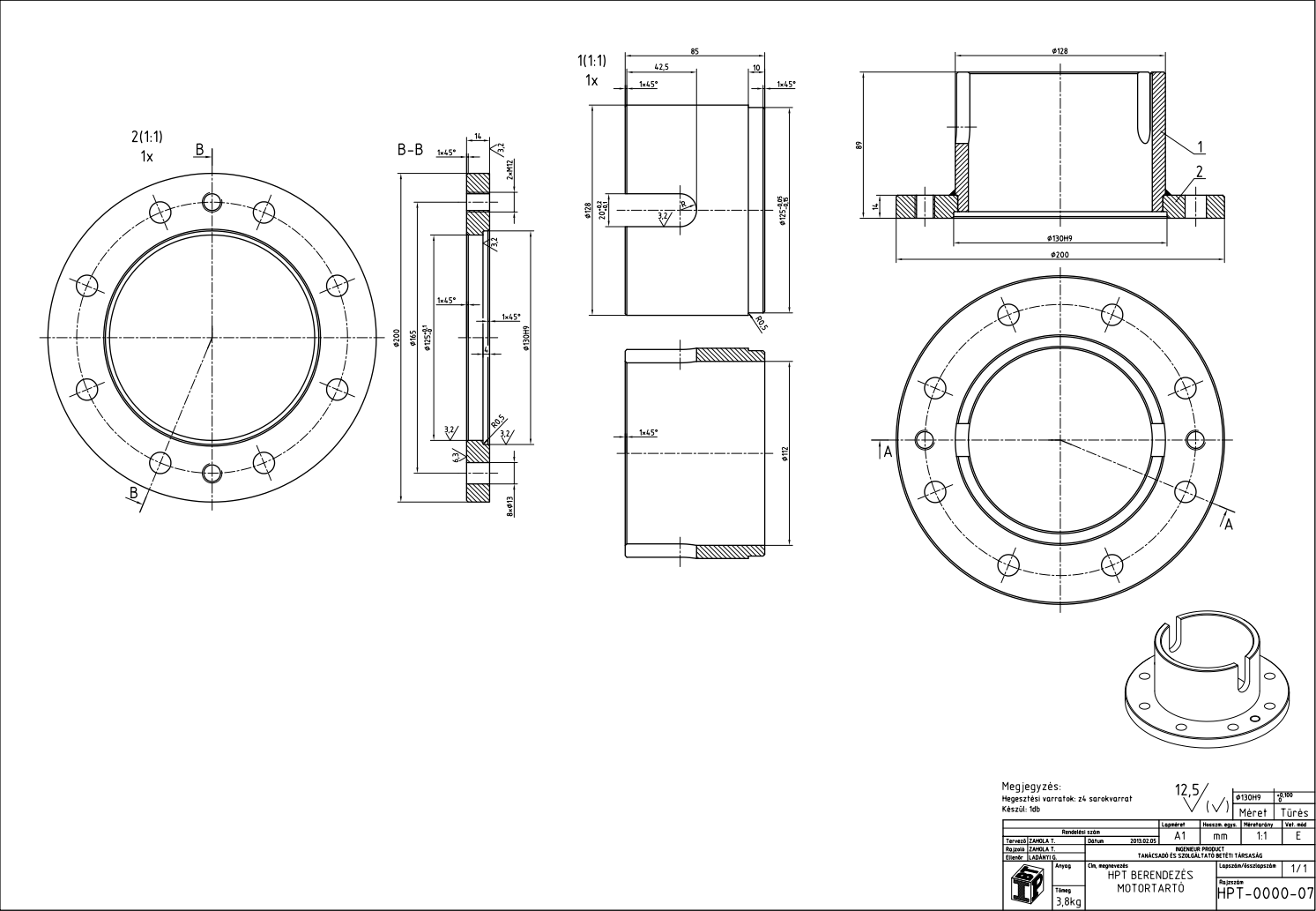
Asm01



Asm02

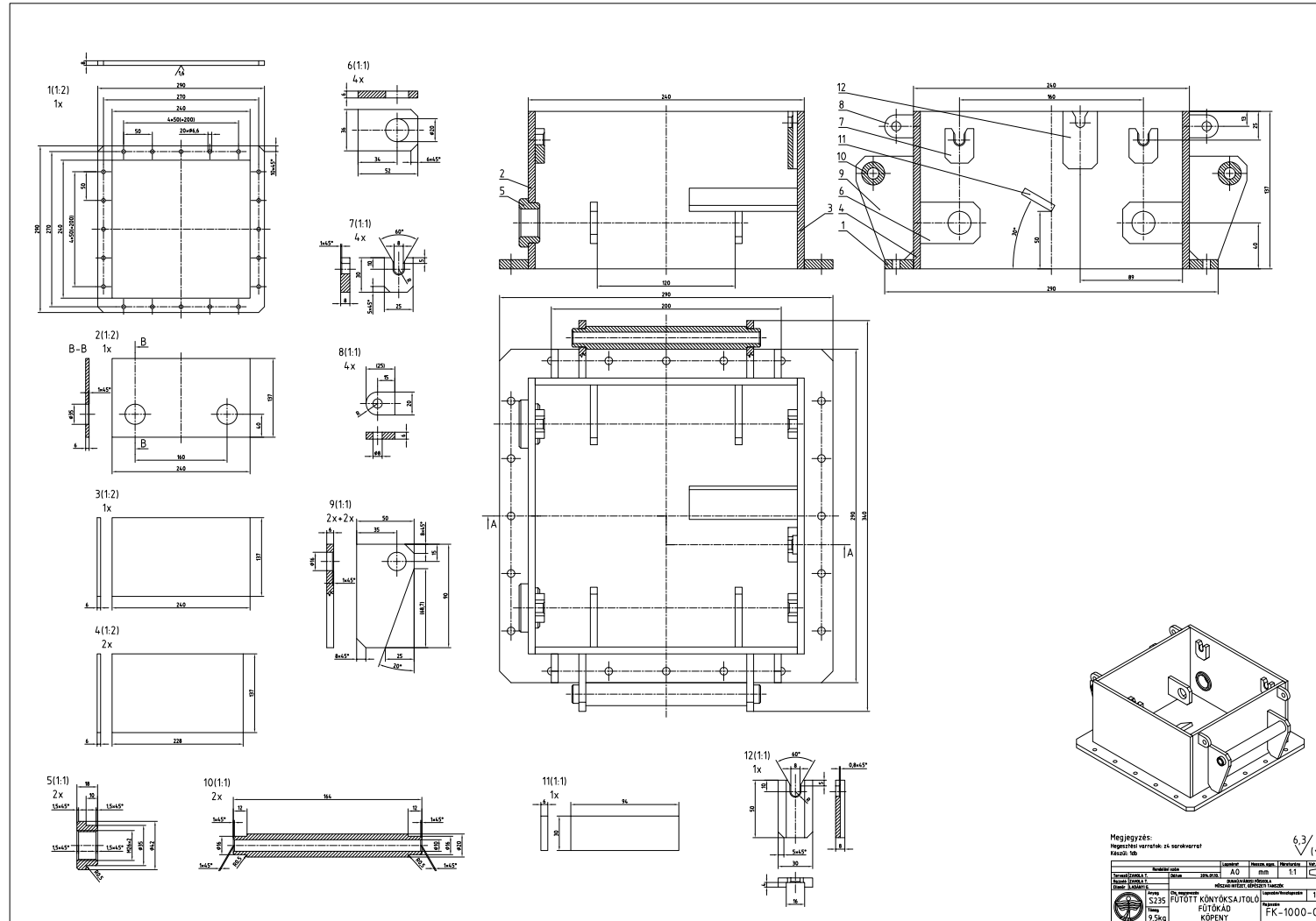


Asm03

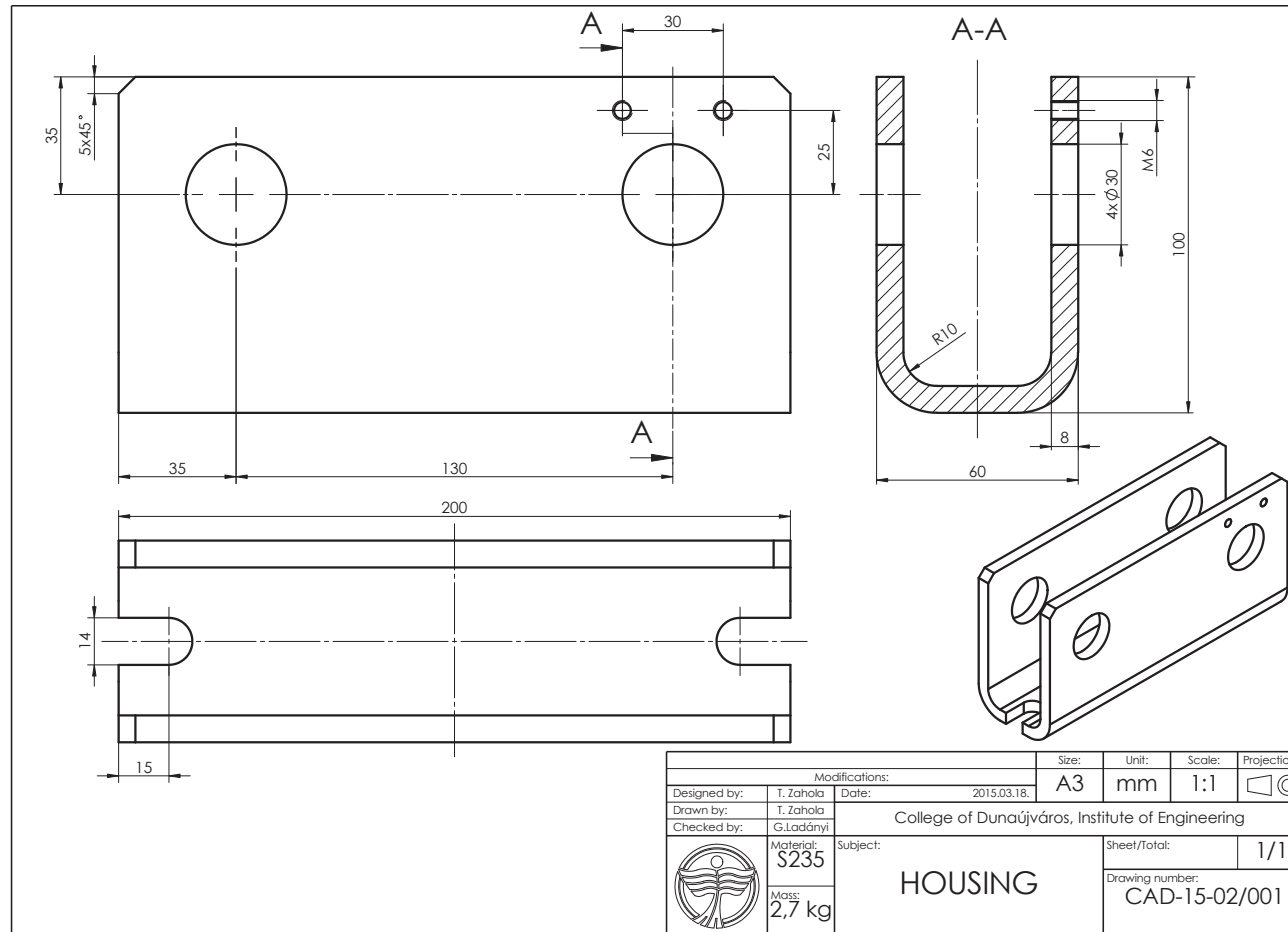


[illegible]

Asm05

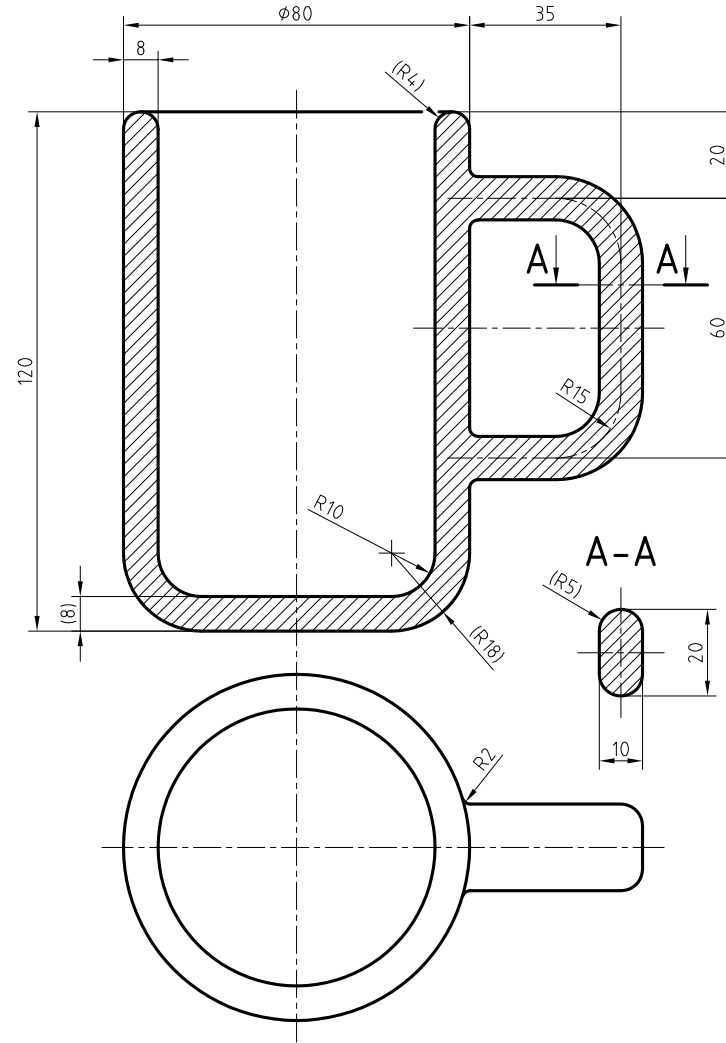


Housing

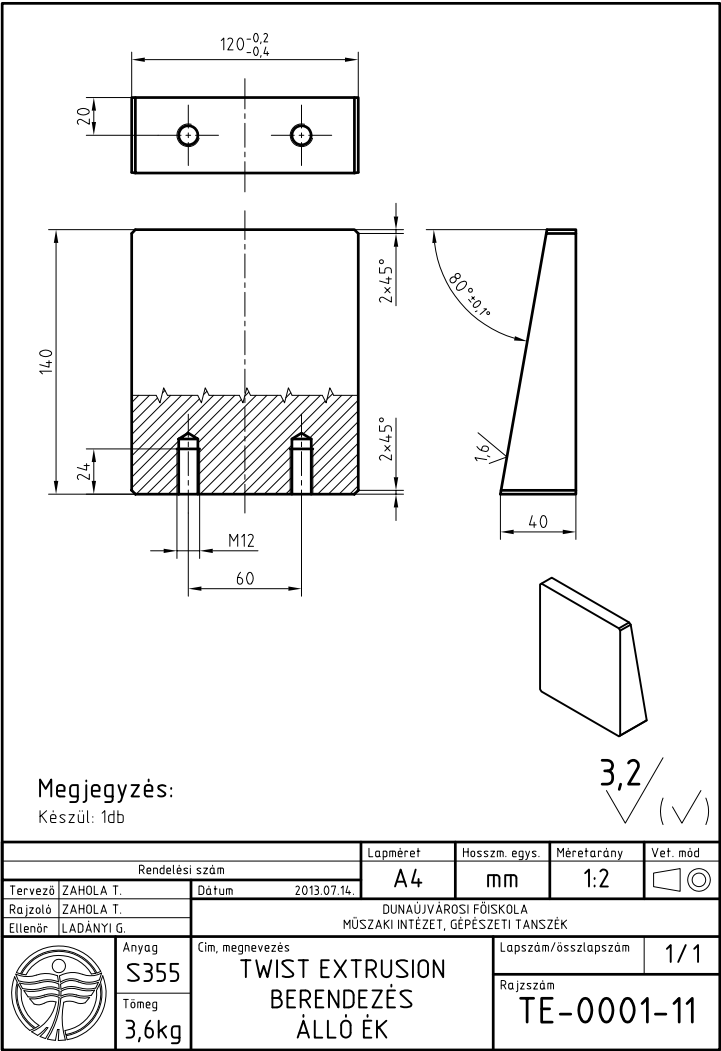


File: C:\Users\Zahala Tamás\Documents\ZF\Tananyagfejlesztés\CAD\angol\Housing

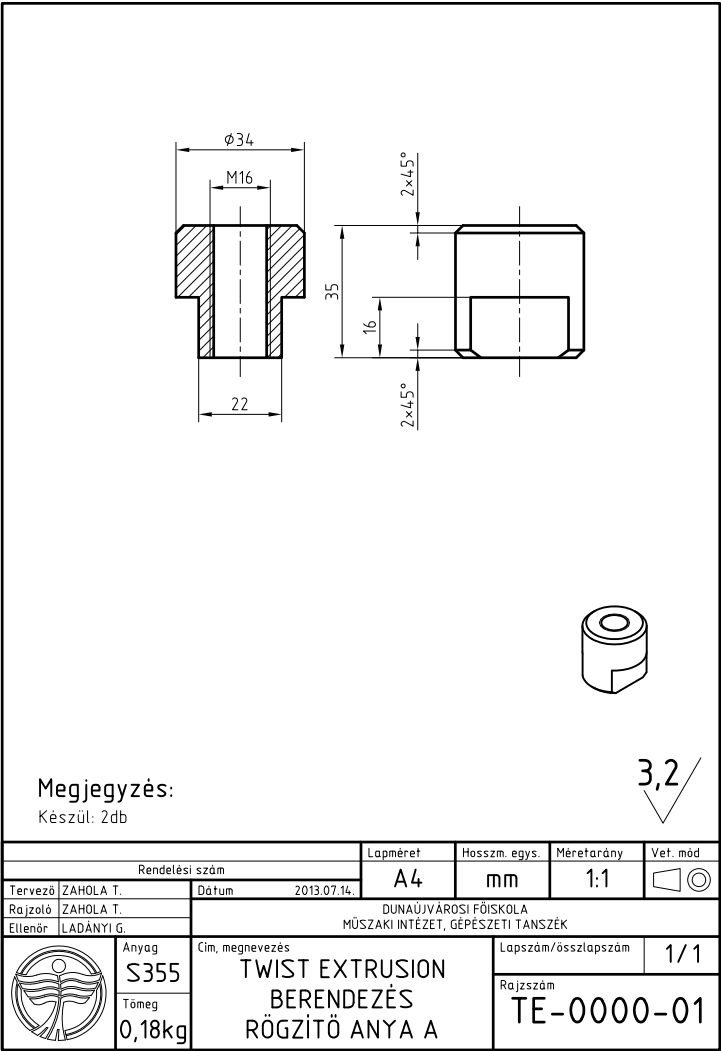
Mug



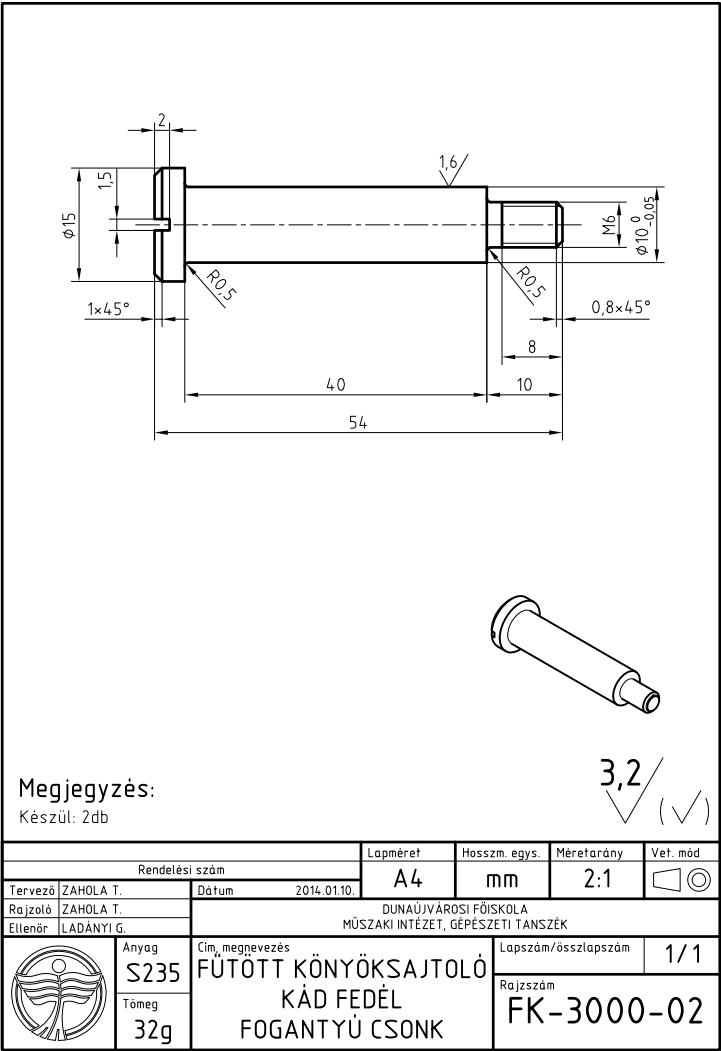
Part01



Part02



Part03



Megjegyzés:
Készül: 1db

3,2/✓

Technical drawing of a shaft-hub assembly. The drawing shows a shaft with a diameter of $\phi 24k8$ and a hub with a bore diameter of $\phi 28k8$. The shaft has a total length of 104 mm. The hub has a bore length of 52 mm. The shaft has a shoulder with a diameter of 24 mm and a length of 45 mm. The hub has a shoulder with a diameter of 20 mm and a length of 45 mm. The shaft has a fillet radius of R0.5. The drawing includes a 3D perspective view of the shaft-hub assembly.

Megjegyzés:

Készül: 1db

3,2/ (✓)

8P9	-0.015 -0.051
$\phi 24k8$	+0.033 0
$\phi 28k8$	+0.033 0
Méret	Tűrész

Rendelési szám		Lapméret	Hosszm. egys.	Méretarány	Vet. mód
		A4	mm	1:1	E

Tervező	ZAHOLA T.	Datum	2013.02.05	INGENIEUR PRODUCT TANÁCSADÓ ÉS SZOLGÁLTATÓ BETÉTI TÁRSASÁG	
Rajzoló	ZAHOLA T.				
Ellenőr	LADÁNYI G.				

Anyag

Cím, megnevezés

HPT BERENDEZÉS
TENGYELCSONK

Tömeg

0,4kg

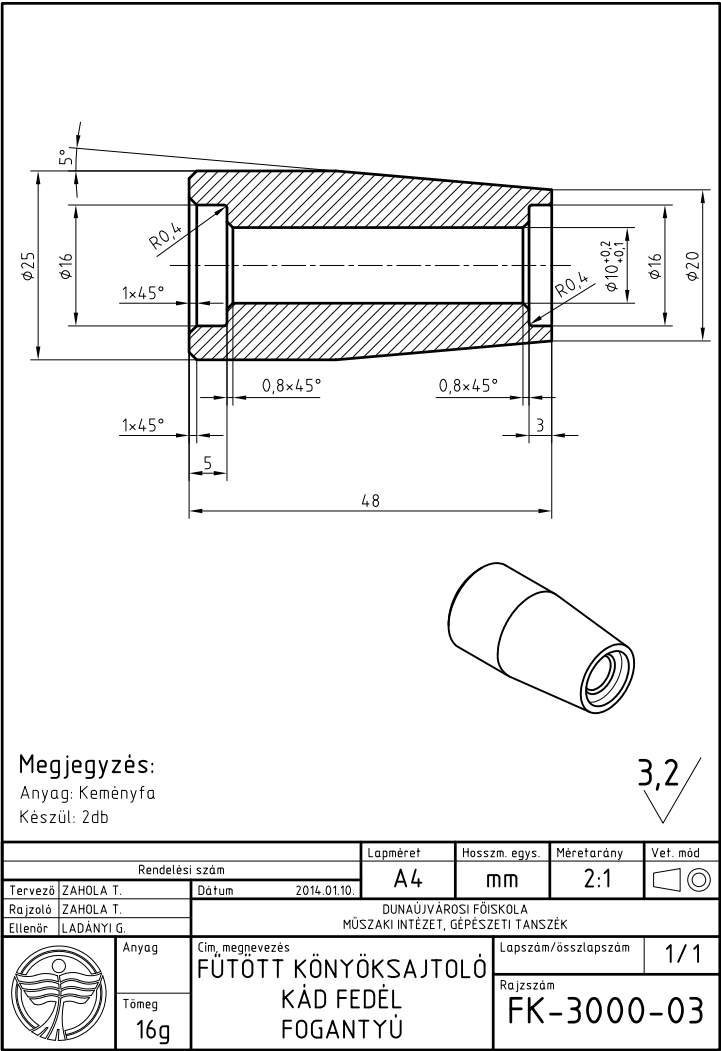
Lapszám/összlapszám

1/1

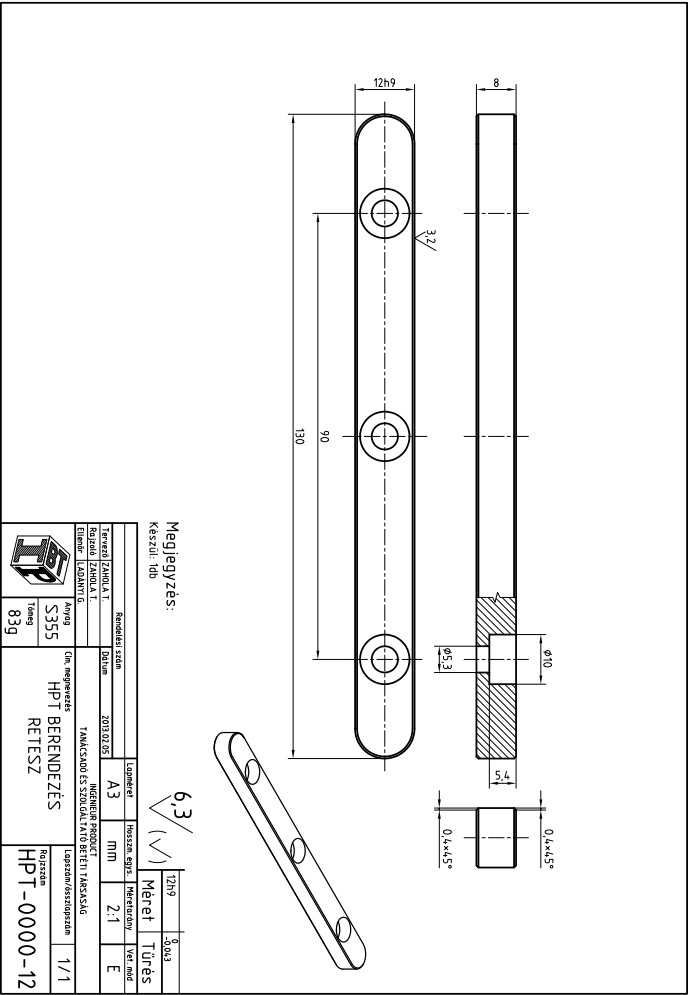
Rajzsám

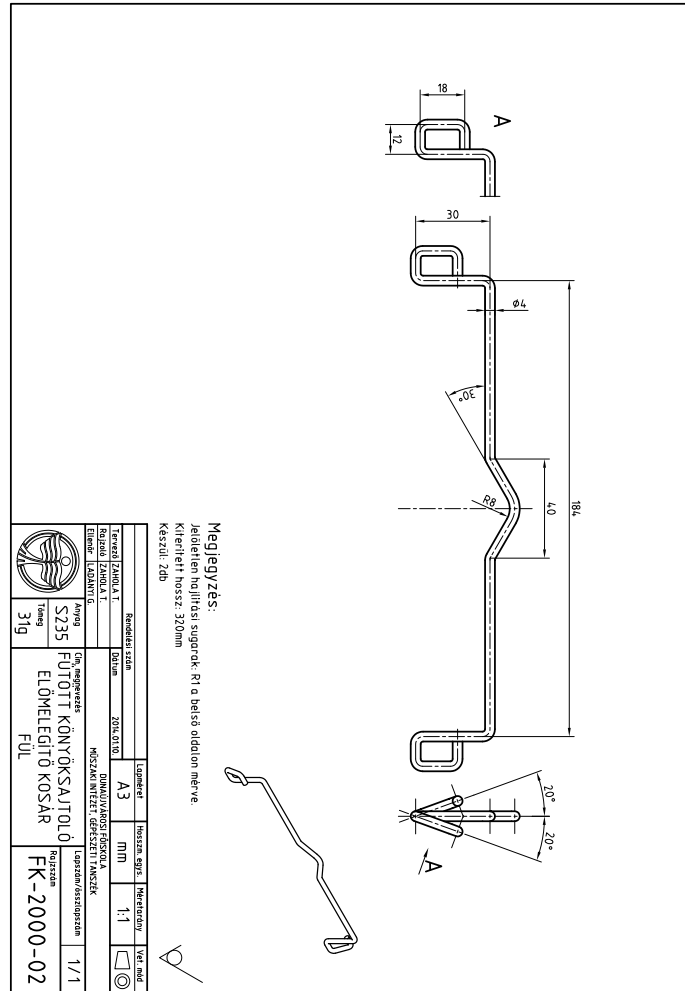
HPT-0000-09

Part06



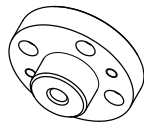
Part07



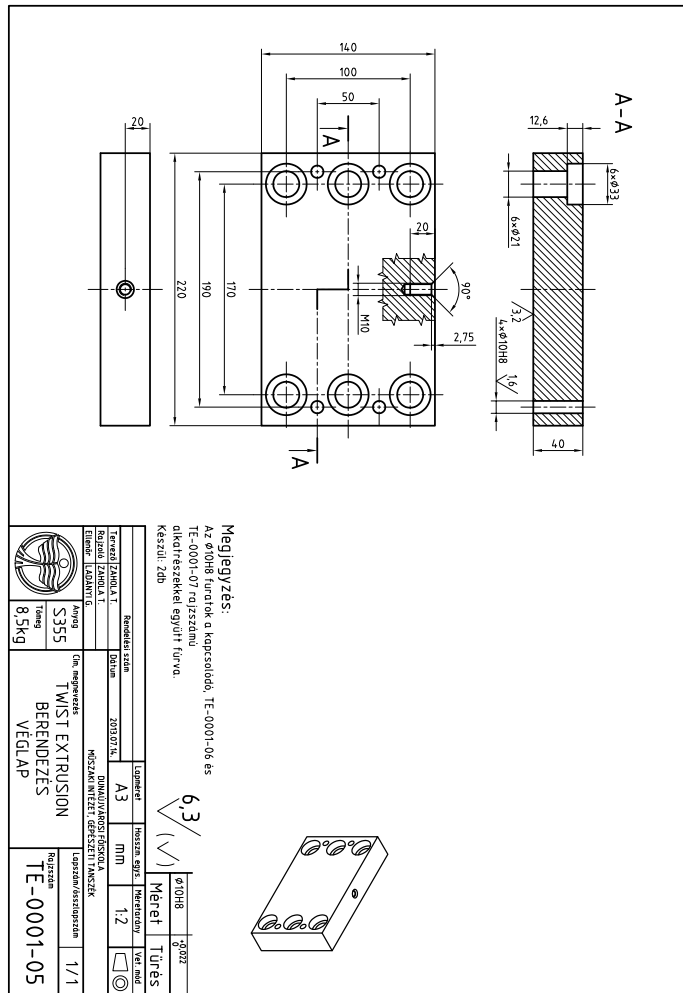




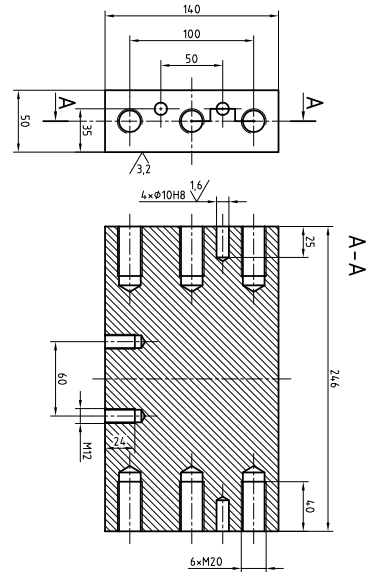




Part12



358



[illegible]

B(2:1)

A-A

B

A

A

6,3

(✓)

φ110k8 ±0,033

Méret Tűrés

Hossz: 1.1

Méretarány: 1:1

Vet. mód: E

Rendelési szám: A3

Dátum: 2013.02.05

Tervező: ZAHOLA T.

Rajzoló: ZAHOLA T.

Ellenőrző: LADANYI G.

Anyag: C60

Tömeg: 0,9kg

Cin. megnevezés: HPT BERENDEZÉS NYOMÓLAP

Lapszám/össztlapszám: 1/1

Rajzszám: HPT-0000-06

Technical drawing of a book rest (Fűtőkád fűszék) showing front, side, and cross-sectional views with dimensions and a table of specifications.

Dimensions:

- Top view: 125 (length), 78 (width), 25 (height), 12 (width of top flange), 6xM6 (top flange holes).
- Side view: 70 (height), 50 $\pm 0,5$ (height of main body), 140 (length), 112 $\pm 0,15$ (length of main body), 160 (length of base), 2x45° (fillet).
- Section A-A: 12 (width of top flange), 8 (width of main body), 8 (width of base), 8 (width of base).

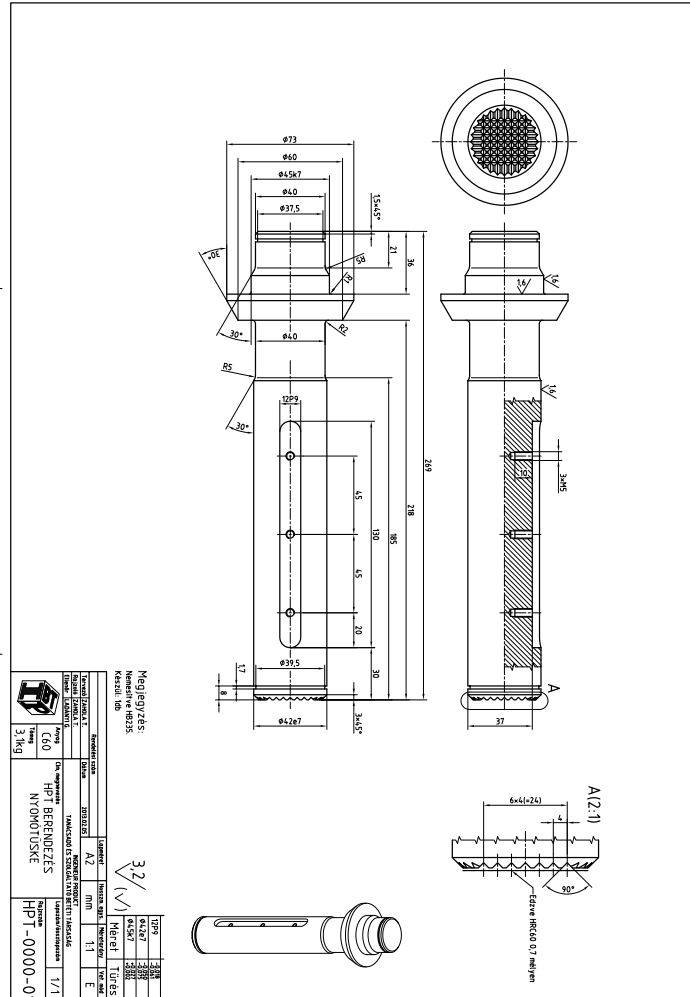
Table of Specifications:

Rendelési szám		Logó méret	Hossz, mm	Méretarány	Vet. mód
Tervező: ZAHOLA A. T.	Dátum: 2014.01.10.	A3	mm	1:1	☒
Rajzoló: ZAHOLA A. T.	DUNAÚJVÁROSI FŐISKOLA MŰSZAKI INTÉZET, GÉPÉSZETI TANSZÉK				
Ellenlő: LADÁNYI G.	Lapszám/össztapszám				
Anyag: S235		Cím, megnevezés: FÜTÖTT KÖNYÖKSÁJTOLO		Lapszám/össztapszám	
Tömeg: 1,4 kg		FÜTŐKÁD FÉSZÉK		Rajzszám: FK-1000-04	

Notes:

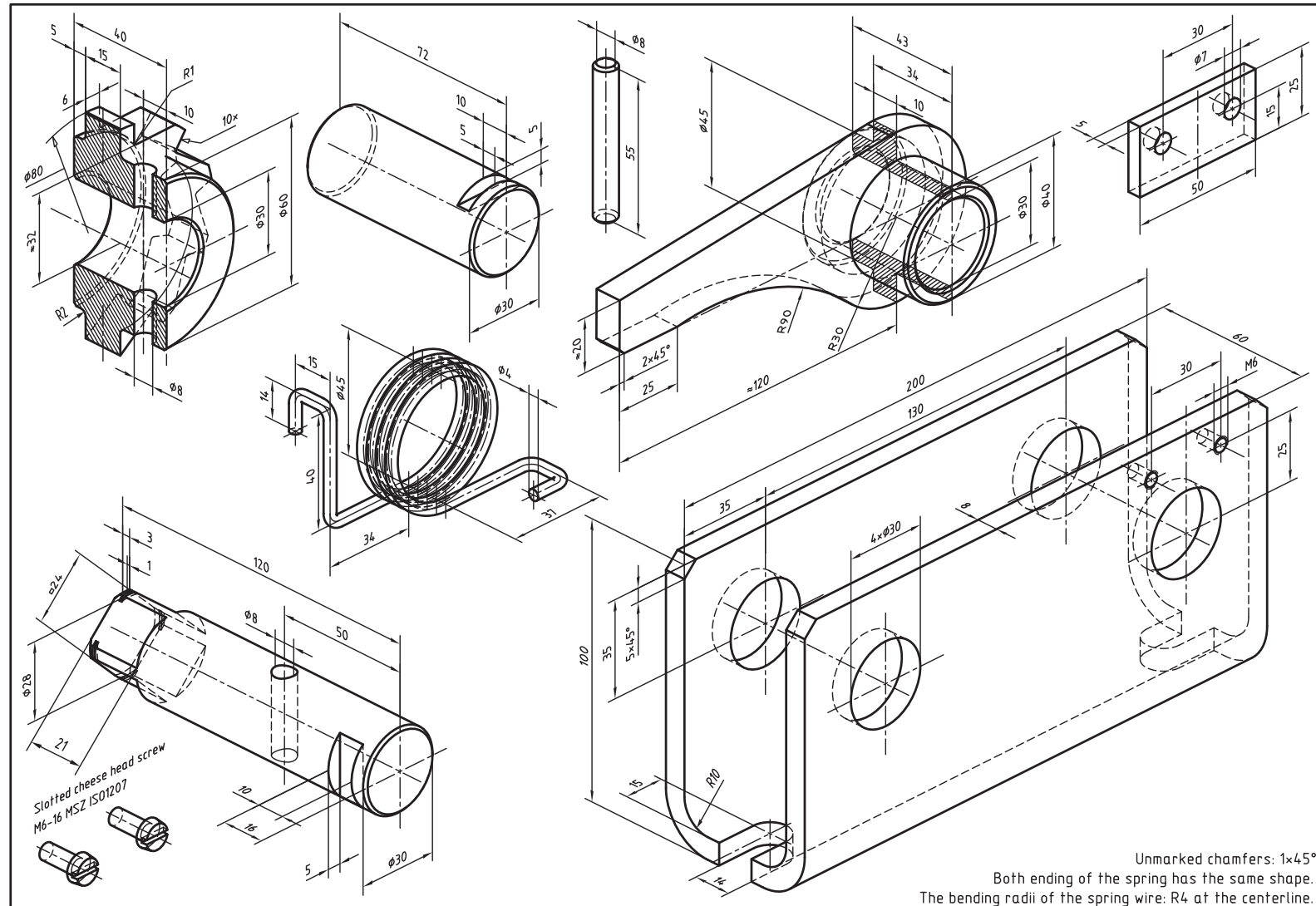
- Megjegyzés: Készül: 1db
- 3,2

Part17

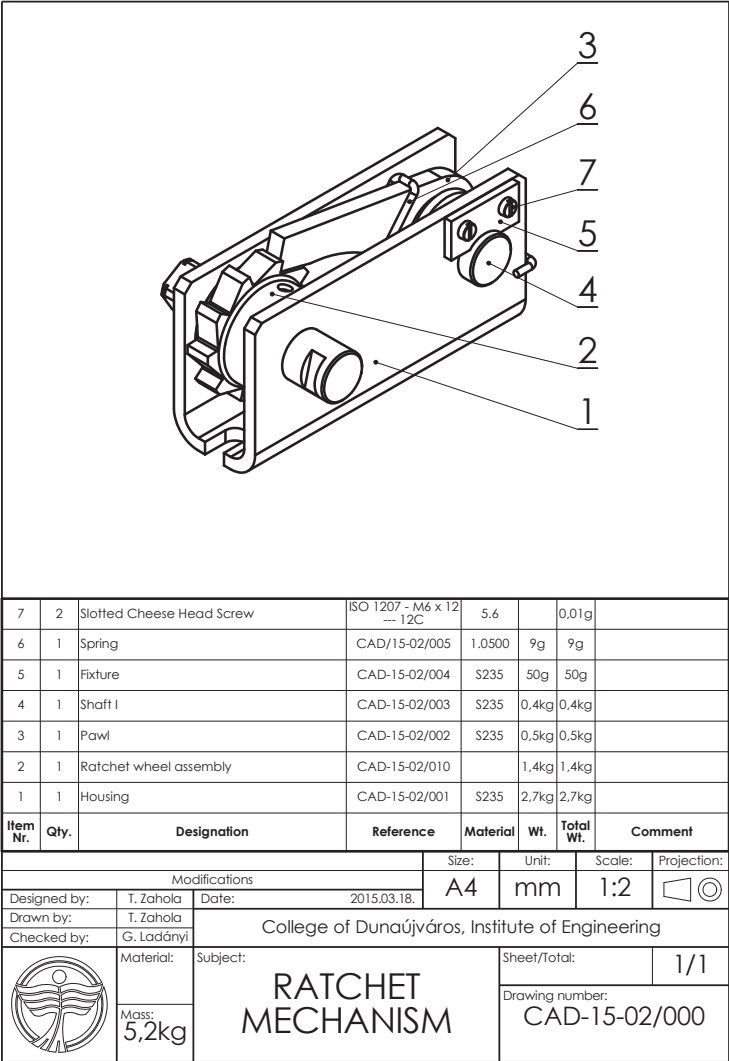


The drawing shows a rectangular metal plate with a central circular hole. The plate has a width of 230 mm and a height of 482 mm. The central hole has a diameter of 100 mm. There are four corner holes with a diameter of 12.6 mm, spaced 15 mm from the corners. There are also four side holes with a diameter of 12.6 mm, spaced 50 mm from the sides. The plate is shown in a perspective view and a cross-section view A-A. The cross-section view A-A shows the plate's thickness of 12.6 mm and the internal structure of the central hole, which has a diameter of 100 mm and a depth of 12.6 mm. The plate is labeled with 'A-A' and '12.6'.

Ratchet mechanism1

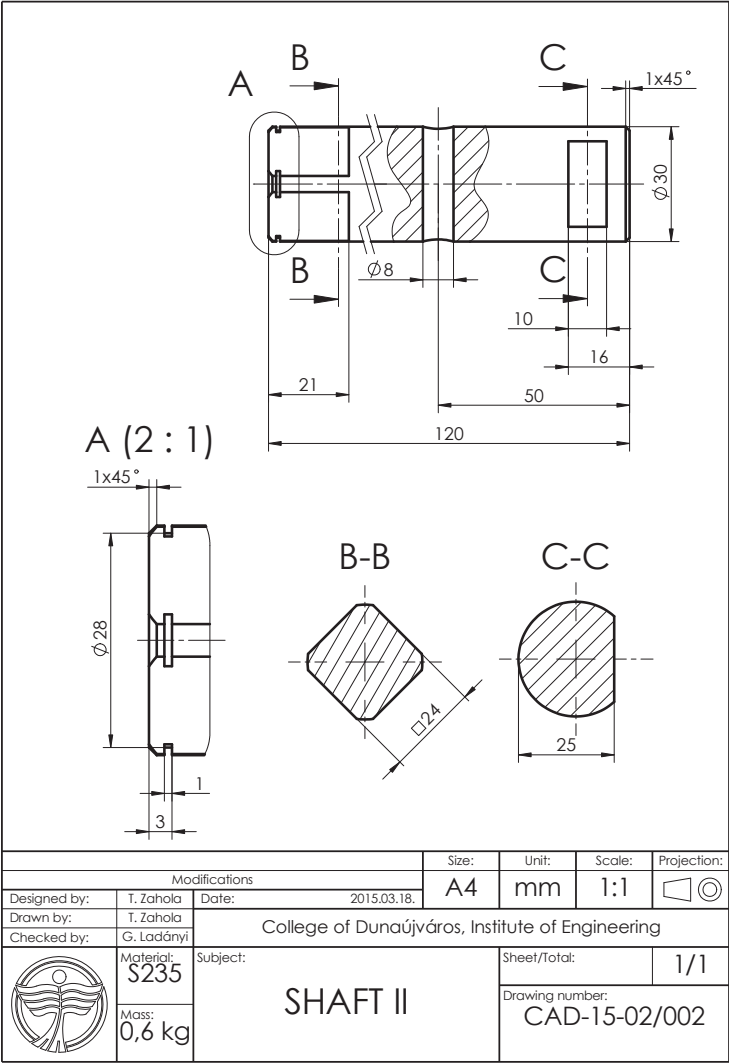


Ratchet mechanism2



File: Ratchet mechanism

Schaft II



File: C:\Users\Zahola Tamás\Documents\Zt\Tananyagfejlesztés\CAD\angol\Shaft II

Stand

