Saimaa University of Applied Sciences
Faculty of Technology, Lappeenranta
Degree Programme in Mechanical Engineering and Production Technology

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Comparison of 3d modeling software

Thesis 2017

Abstract

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Saimaa University of Applied Sciences
Faculty of Technology, Lappeenranta
Degree Programme in Mechanical Engineering and Production Technology
Thesis 2017
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The purpose of the study was to do a research and get knowledge about three different 3D modeling Softwares and to figure out the best option for a Mechanical Engineering student at Saimaa University of Applied Sciences and also identify the advantages and disadvantages of each program.

The thesis work consists of a theoretical part, which includes the key data about the 3D modeling process, applications, use in Mechanical engineering field and information about the tested programs. The practical part analyses the 3D modelling process of a chosen software and considers the functions and features of these programs. All the data for this thesis was collected from network and books.

The result of the study shows a selection of the best 3D modeling programs for students at Saimaa University and arguments for this choice.

Keywords: 3D modeling, software, SolidWorks, FreeCad, Kompas 3D, drawings, assembling

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1 Introduction

1.1 Definition

3d modelling in engineering is the process of creation of three-dimensional model of object in order to create, modify, analyze or optimize the design of the object using special Software. 3d modelling software is used to create a virtual model of object, which takes into the account dimensions, materials, tolerances and stresses which affect the object. This design leads to better quality of product after manufacturing, increasing the productivity, improvements in the quality of design. (https://en.wikipedia.org/wiki/Computer-aided_design)

1.2 History

The first 3d Software appeared in the 1960s, however the leap forward occurred in the 1990s when computer technologies became massive and became available to ordinary users. Implementation of 3d modelling software opened up wide opportunities for creating models with complex forms, simplified the design and planning process, there was an opportunity to identify and eliminate errors at the design stage, which allowed society to take a step forward and bring the design and production process to a new level. (https://3d-innovations.com/blog/the-history-of-computer-aided-design-cad/)

CAD-systems became an important part in the field of engineering design. These serve for the creation of 3D models of machine-building units, products, buildings, etc., the formation and processing of a set of drawings together with a full set of design documentation necessary for the release of the product or the construction of the object.

1.3 Applications

Nowadays 3d Modelling has a wide range of applications:

- Architecture. 3D modeling is a tool in the field of architecture. Use of 3D modeling technologies gives an opportunity for engineers and customers to see how the structure or building will look after construction.

(http://www.steves-digicams.com/knowledge-center/how-tos/video-software/6-industries-that-use-3d-modeling-software.html#b)

- Advertising and filming. In the field of film industry and TV advertising 3D technologies can reduce the expenses cost during the shooting process. Moreover, sometimes it is easier to use 3D technologies than shoot some scenes in reality. (http://www.steves-digicams.com/knowledge-center/how-tos/video-software/6-industries-that-use-3d-modeling-software.html#b)
- Interior Design. 3D models of furniture and other attributes allow to simulate the interior design of building even before it is built, thereby it helps to save money and time because it allows to avoid wrong selection and arrangement of components. (http://www.steves-digicams.com/knowledge-center/how-tos/video-software/6-industries-that-use-3d-modeling-software.html#b)
- Web design. Fast development of computer graphics causes high demand in 3D technologies in the process of creation of computer games and web sites. (http://www.steves-digicams.com/knowledge-center/how-tos/video-software/6-industries-that-use-3d-modeling-software.html#b)
- Science and Industry. In the era of widespread implementation of new technologies 3D modeling is an important part in the fields of science and industry. All modern production begins with preliminary 3D modeling, which helps to organize the process in more efficient way by saving money and time, because it helps to solve the problems and to fix it before establishing the production. (http://www.steves-digicams.com/knowledge-center/how-tos/video-software/6-industries-that-use-3d-modeling-software.html#b)

1.4 Importance of 3D modelling in Mechanical Engineering

In the era of modern technologies and implementation of 3D technologies it has become easier and faster to get the needed results for manufacturers and companies. Today the quality of final product depends on the simulated model which was developed and tested during the first stages of manufacturing process. Deep integration of 3D modeling brought the field of manufacturing to the new level of development, which was caused by decrease in the chance of error during engineering design and production processes. (http://www.indovance.com/how-3d-modeling-made-things-smooth-in-the-engineering/)

3D modeling using special software has almost replaced hand drafts and drawings which engineers used before. With the help of 3D modeling software creating virtual projections for mechanical and industrial goals has become more comfortable and easier.

The influence of 3D modelling has increased in all industries. In mechanical engineering field engineers use the technology in order to create 3D models, which help them to understand the view of the object: dimensions or shapes, it opens different viewpoints, that improves the understanding by a greater degree. Use of 3D models simplifies this task a lot, while use of 2D sketches makes it much more difficult for understanding, because if the design of the object has complex shapes, it is difficult to imagine or visualize it. 3D models simplify visualization, conceptualization and prototyping. (http://www.indovance.com/how-3d-modeling-made-things-smooth-in-the-engineering/)

In addition, 3D models are the real objects which will be manufactured. The technology of 3D modeling gives a possibility to make a cut for the object from different sections and rotate it in order to have a better view from inside and outside. It gives a good option for engineers to discover the object better, so chances to commit errors go down, it makes the product development stage faster that leads to the preservation of money. (http://www.indovance.com/how-3d-modeling-made-things-smooth-in-the-engineering/)

The next advantage is that it is possible to change the colors, shapes, making changes in the colors, texture, forms, shapes, etc. is very easy. (http://www.indovance.com/how-3d-modeling-made-things-smooth-in-the-engineering/)

1.5 Description of Work

The purpose of this thesis is to do a research about three 3d modelling softwares in order to define the advantages and disadvantages of them and choose the most user- friendly and suitable Software for the most common tasks of engineering students at Saimaa University of Applied Sciences

2 Software for comparison

For the purpose of research such computer programs as Solidworks, FreeCad and Kompas 3D were taken.

2.1 Solidworks

SolidWorks is a solid modeling computer-aided design and computer-aided engineering computer program that runs on Microsoft Windows. (https://en.wikipedia.org/wiki/SolidWorks)

SolidWorks company was established by Jon Hirschtick in December 1993. The first version of SolidWorks software was released in November 1995. Since that time 24 versions of Software were published and today Solidworks takes the leading position on the market of engineering 3D modelling software. (https://en.wikipedia.org/wiki/SolidWorks)

2.1.1 Technology

Solidworks uses the principle of parametric modelling in its technology. The idea of this technology is to express the features of body which consists of complex shapes and geometry with the help of entities, the form of which is set using parameters and relations between these parameters. The parameters are: coordinates of points, lengths of edges, angles, diameters of circles, radius of ellipses etc.

2.1.2 Features

- Parts and assembling of components

The main feature of Solidworks is creation of parts. The parts can be different, with simple or complex shapes. When all the parts have been done and saved it is easy to use the feature of assembling which allows to connect all the parts into one system in the needed way and order. (http://shoutmetutorials.com/solidworks-basics/)

- Drawings

After creating a part or assembly it is possible to do the technical drawing for all the components of assembly or the drawing for assembly itself. It is necessary to make sure that the object parts fit properly and operate in the needed way during assembling, that is why the drawings for each part and the whole assembly have to be done. Technical drawings include the dimensions of object, bill of materials, quantity of parts, index number of part and other needed information. Also it includes the views of object from different angles that makes the detail better for understanding and helps customers and manufacturers to understand the shapes and features of object. (http://shoutmetutorials.com/solidworks-basics/)

- Simulation & Analysis

Solidworks allows users to do the simulation and analysis for the object which was created in this program. The engineering design of the structure can be checked in the conditions set by users, the program creates virtual environment depending on the settings and tests the objects. It is possible to check the behavior of object under static or dynamic loads, check the stability under these loads, also tests for fatigue, thermal expansions and other stresses are available. (http://shoutmetutorials.com/solidworks-basics/)

- Animation & Rendering in 3D

The possibility to animate the part or assembly is a good advantage of Solidworks. This feature creates an animation which can be saved as a video file. This opportunity gives many advantages during the product development stage and it helps to understand the functionality of the object better. The use of animation helps to save the time, increase productivity and develops the marketing of the product. Engineers can easily communicate with the customers who are far from the 3D modeling field by showing high quality and realistic animations with the product. (http://shoutmetutorials.com/solidworks-basics/)

2.2 Kompas 3D

Kompas 3D is a software for 3D modelling and creation of technical drawings which is produced by Russian company ”Ascon” since 2000. (https://ru.wikipedia.org/wiki/Компас_(САПР))

Kompas 3D is more focused on the Russian market being in the group of leaders in the field of 3D modelling software. Nowadays the company is trying to enter the international market, but still remains quite unknown for foreign users. (https://ru.wikipedia.org/wiki/Компас_(САПР))

2.2.1 Technology and features

Kompas 3D uses the same principle as Solidworks and operates as parametric modeling software, including all main tools for standard parametric 3D modeling, 2D drafting and designing, creation of needed documentation and technical drawings.

2.3 FreeCad

FreeCAD is a free 3D modeling software which is used for designing of real- life components with different shapes and sizes. The first version of software was released by Jürgen Riegel, Werner Mayer and Yorik van Havre in October, 2002. (https://en.wikipedia.org/wiki/FreeCAD)

In contrast to the comparative in this thesis program FreeCad is a multiplatform and totally free software. Moreover, it is an open-source program with the possibilities for customizing, use the scripts and extensions. (https://www.freecadweb.org/wiki/About_FreeCAD)

The Program uses a parametric model. The shapes and views of all objects depend on its properties. The Program allows to do the drawings of an object and make 2D views of a 3D object on sheet.

3 Practical comparison of Software

3.1 Description of object

For the practical comparison the model of a chain tightener was taken. The construction of the object is shown in Figure 1.

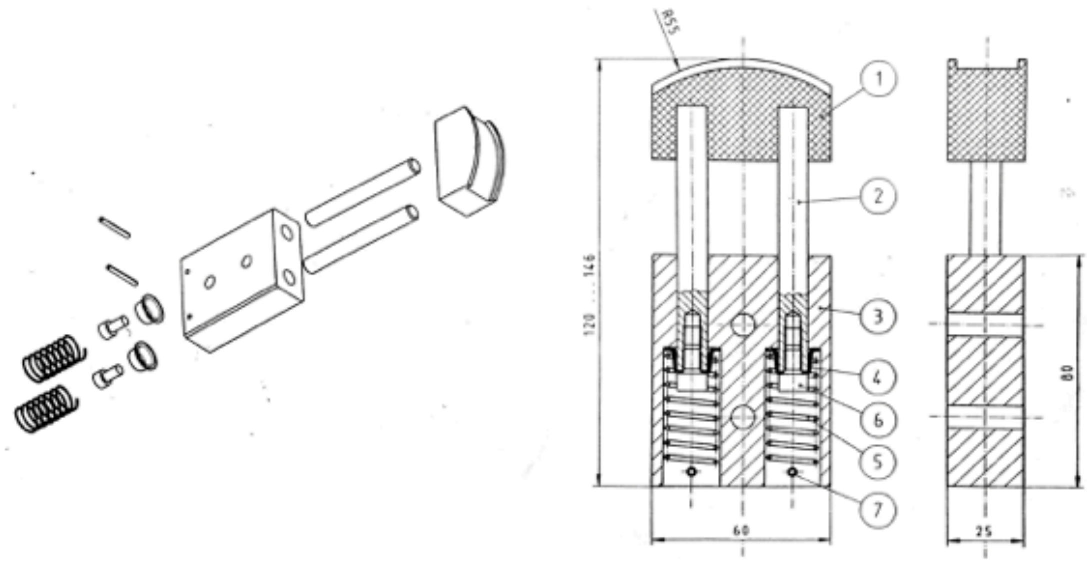

Figure 1. Chain tightener (Jukka Nisonen 2017)

As can be seen from the picture the object consists of several parts and has some special parts, such as springs and bolts.

During the practical comparison of the software all the parts will be created in three different softwares in order to compare the procedures and define the advantages and disadvantages of each software.

Parts to build:

- Plastic slide

- Main body
- Shaft
- Press cap
- Bolt
- Pin
- Spring

The parts have different features and complex shapes, so it is a good case for comparison of the software for different features and identify its strengths and weaknesses.

3.2 User interface

This chapter shows the main features of interfaces of the chosen programs.

3.2.1 Solidworks

The interface of Solidworks is organically organized and user- friendly. All the operations can be done easily because of a well thought out interface scheme. The main elements of the interface are shown in Figure 2.

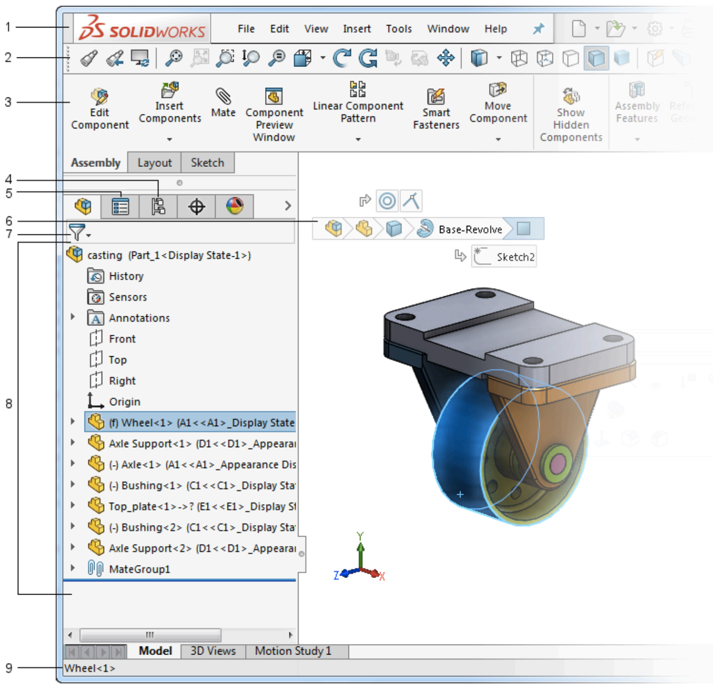

Figure 2. Interface of Solidworks. (Solidworks 2016)

1- Menu bar. This bar contains the most used tool buttons and commands and other Solidworks menus, search and help options.
2- Toolbars. Sets of tools.
3- CommandManager. It is a context toolbar, sensitive to toolbar which is used, and available tools are updating according to chosen command.
4- ConfigurationManager. This tool is responsible for creation, selection of configurations of parts and assemblies.
5- PropertyManager. The user can set the properties for Solidworks commands using this tool.
6- Selection Breadcrumbs. This tool shows the related elements of part, using hierarchical tree.
7- FeatureManager Design Tree Filter. This tool helps to search specific features of parts and assemblies.
8- FeatureManager Design Tree. This tool shows all the components of part, assembly of drawings. Using this tool it is easy to control the process of construction.
9- Status Bar. Provides the information during the process.

(http://help.solidworks.com/2016/English/SolidWorks/sldworks/c_user_interface_overview.htm)

3.2.2 Kompas-3D

The interface of Kompas- 3D has a standard appearance for 3D modelling software. It has a well organized structure, that makes the process of 3D modelling comfortable and fast. The main features of the interface are shown in Figure 3.

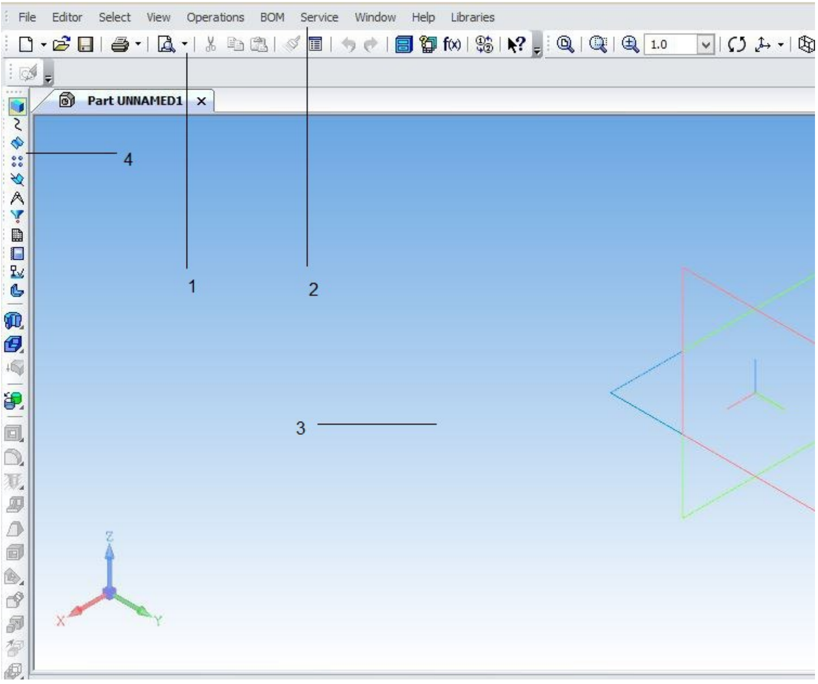

Figure 3. Kompas- 3D interface.

1- Tools bar. This bar contains the most common used operations during the modeling process: creation of file, open, save. Also it is possible to set the scale, zoom and set the orientation of object.
2- Menu bar. This bar is used for standard operations, contains the most common used commands and settings.
3- Workspace.
4- Command manager. Contains all the features which can be done. The features are different and depend on the process. Part modelling, assembly building, drawings have its unique set of features and tools.

3.2.3 FreeCAD

As for freeCAD, this software has the different software from Solidworks and Kompas- 3D. The main feature of the interface is the availability of different Workbenches, that helps to demarcate the sets of tools and features, depending on the process. The interface looks modern and user- friendly. The main elements of the interface are displayed in Figure 4.

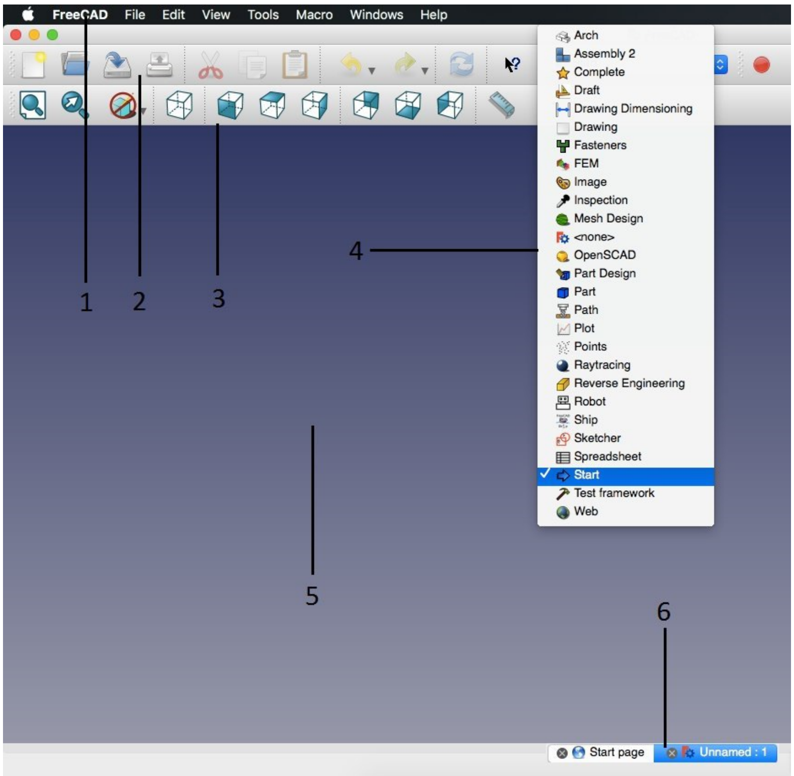

Figure 4. FreeCAD interface.

1- Menu bar. Standard bar, which contains the basic operations which can be done with the document.
2- Tools bar. Tools for most common used operations: Open, Create file, Cut, Undo etc.
3- Command bar. This bar contains the tools and features which are available in specific workbenches.
4- Workbenches. The heart of FreeCAD. This bar contains all sets of tools available in FreeCAD. Each set has the unique tools and features.
5- Workspace.
6- Files bar. This bar shows opened pages and documents.

3.3 Process of creating a body

The body of the chain tightener consists of two pieces: plastic side and housing in which other components of the object are inserted.

The housing has the holes with different diameters on the surface and the edges have the fillets. The plastic side has a specific shape, one of its sides is rounded off and has the groove on this side as well.

All the needed dimensions can be taken from the drawings or estimated by the person who creates 3d models.

3.3.1 Creating a body and plastic slide in Solidworks

The process of modelling a body in Solidworks begins with the creation of a new file, choosing "Part" template. After that the user gets to the workspace of Solidworks. From the "Tree Items" toolbar it is necessary to choose the plane on which the sketch drawing will be created and then the process starts.

For chain tightener the body has a shape of parallelepiped, it means that the basis of this part is a rectangle.

The sketch drawing of the basis of the body can be seen from the picture, it is the rectangular with the width of 60 mm and height of 80 mm. The dimensions of the components are set with "Smart Dimension" feature from Toolbox. The sketch for the basis of the body is shown in Figure 5.

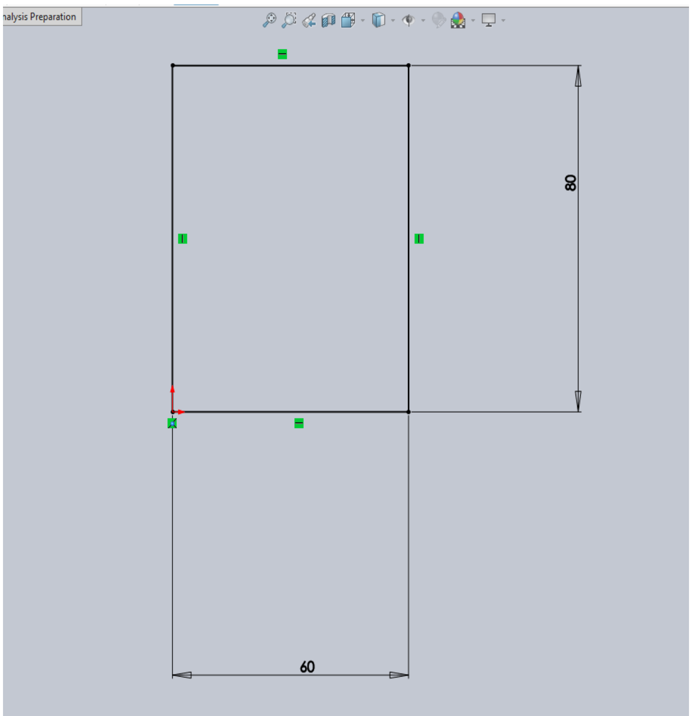

Figure 5. Sketch of body in Solidworks.

Then it is necessary to give a volume for the sketch, for this purpose “Extruded boss/base” feature is used. For this operation it is needed to set the depth of extrusion and the direction, all this can be set in the window which appears after starting this feature. After this operation the solid body with thickness of 25 mm was created.

The result of creation of solid body is shown in Figure 6.

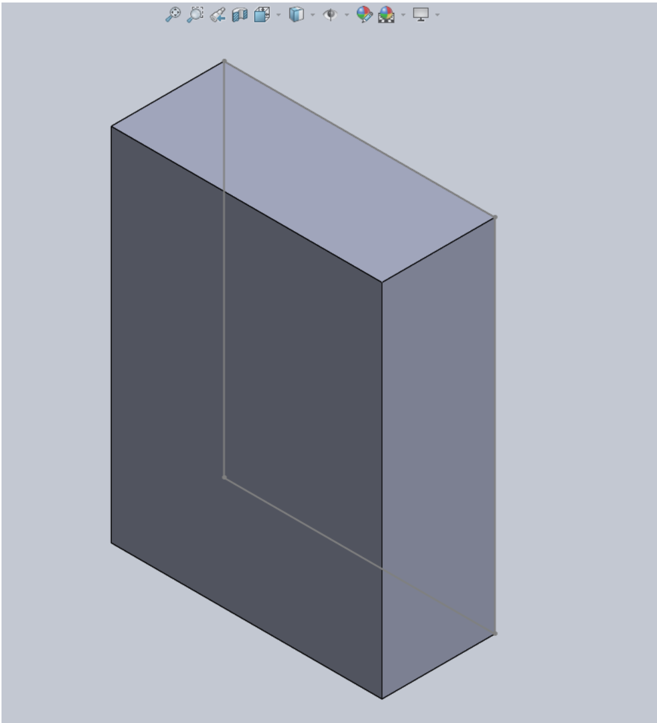

Figure 6. Solid model of body in Solidworks.

The next step was to do all the holes on the surface of the body. For this goal it is needed to choose the side of the body where the holes should be located and then to draw the sketch on this side. The basis of the hole is a circle and the diameter of this circle is the diameter of the hole.

The sketch of the holes for springs can be seen in Figure 7. The diameter of the holes is 18 mm and the centers are located in 13 mm from the edges of the body. The centers of the circles are located on the center- line which splits upper and lower bases for two equal parts.

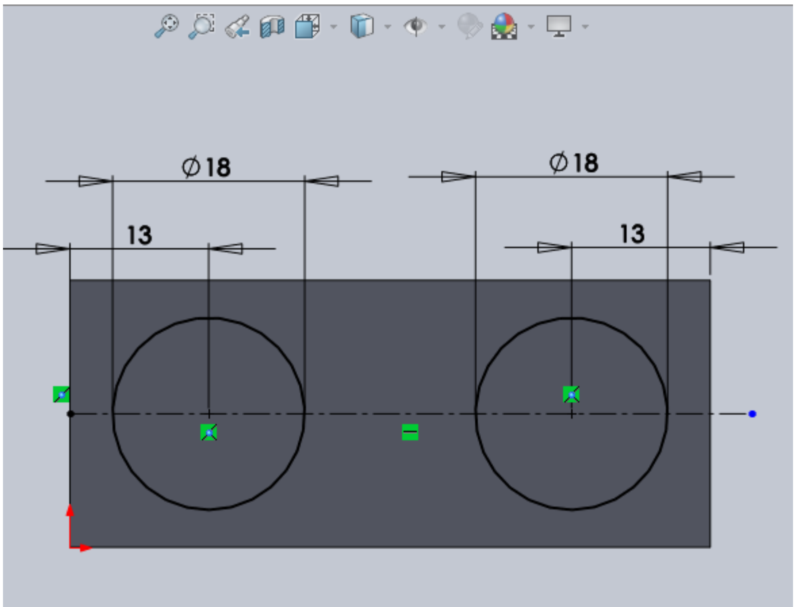

Figure 7. Sketch of the holes.

Then the feature of extruded cut should be used in order to remove all extra material. In the case of holes for springs the depth of remove is 45 mm, after this operation the solid block with cylindrical holes is formed.

All the other holes can be done with the same principle, taking into account the locations and the dimensions of the needed holes. After completing this operation, the solid body with the holes on the upper, front and back sides was formed.

The last step was to do the fillets for the edges of body of chain tightener. For this task the feature “Fillet” was applied. This feature helps to create a rounded face on the part.

The main parameter for fillet is its radius, for the given part it is 1 mm. After setting this parameter and choosing the edges the feature can be applied.

After the last step the body was ready, the model takes into account the shapes of the body, the dimensions and features of the object. The result can be seen below in Figure 8.

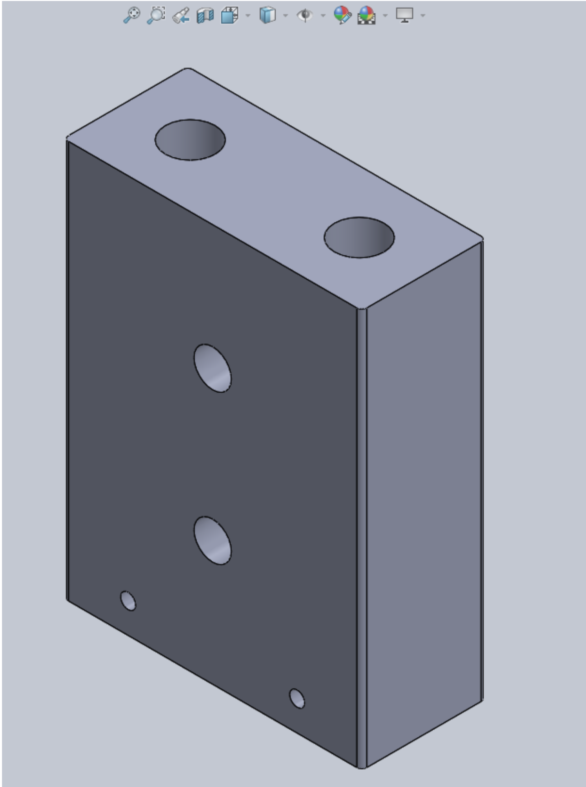

Figure 8. Body of chain tightener in Solidworks.

The main principle for creation of the plastic slide of chain tightener is the same as in the case with the body. But the shape of this part is different from the body, one of the sides of object is rounded off and there is a groove on this side. That is why the modelling process of the plastic slide is different in some steps and described in this chapter.

First of all, the rectangular sketch with dimensions 60 mm x 35 mm was created on the front plane and then extruded for 12.5 mm.

To get a rounded shape of surface it was needed to choose the side of object and to draw a center-line. The next step was to do a sketch of “centerpoint arc” with the center in 55 mm away from the upper side of the body and draw the arc the with ends on the sides of body.

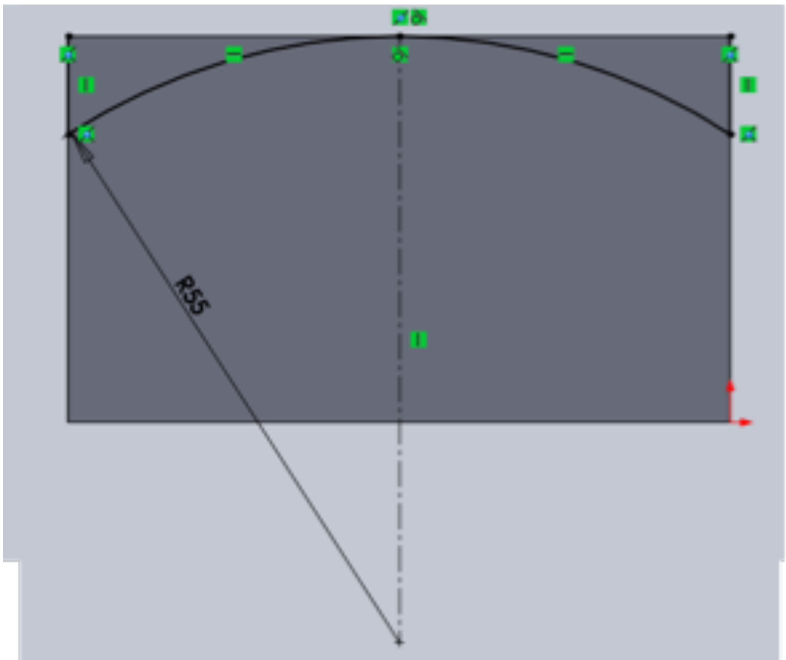

Figure 9. Sketch for plastic slide.

Then all extra material which is located out of arc should be deleted using “Extruded cut” feature with depth of 12.5 mm.

To get the groove the same procedures were applied, the sketch with “Centerpoint Arc” with radius of 53 mm from the point on center-line to the point on the upper side was created. After “Extruded cut” feature the groove was formed.

After this operation the solid object with right shapes was done, but it was the half of the needed body with the thickness of 12.5 mm. To get a full shape it is necessary to use “Mirror” feature in Solidworks which allows to create opposite-hand versions of objects.

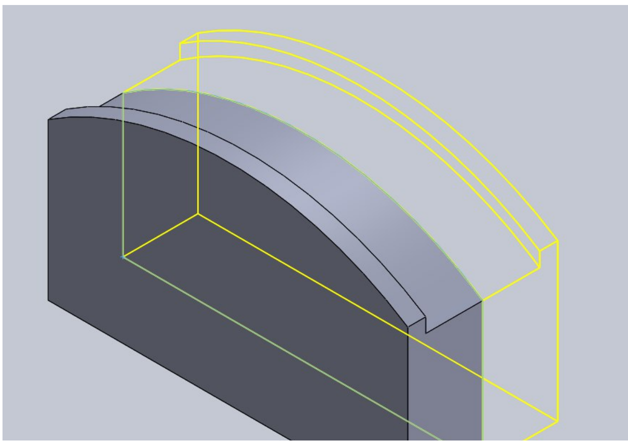

Figure 10. Mirror operation in Solidworks

The last step was to do the holes in down side of object and after this step the part was ready.

3.3.2 Creating a body and plastic slide in Kompas-3D

Mostly Kompas-3D has the same modelling system as Solidworks but the interface of the program is different, it looks a little outdated and less user-centric, so the process of creation took more time and effort. Anyway, the results were good and the parts were created.

The process of creation starts with the creation of a new file. From the list of templates, it is necessary to take “Part” then the program moves the user to thw workspace. After choosing the plane the process of modelling begins.

After enabling sketch mode and choosing “Geometry” from toolbox it is possible to do the sketch.

For the body of chain tightener the basis shape is rectangular, so the rectangular tool was taken. Starting the sketching from origin the program offers to set the dimension in the special box where the height and width of object or the coordinates of two opposite top points can be set. After setting these features Kompas-3D builds the sketch.

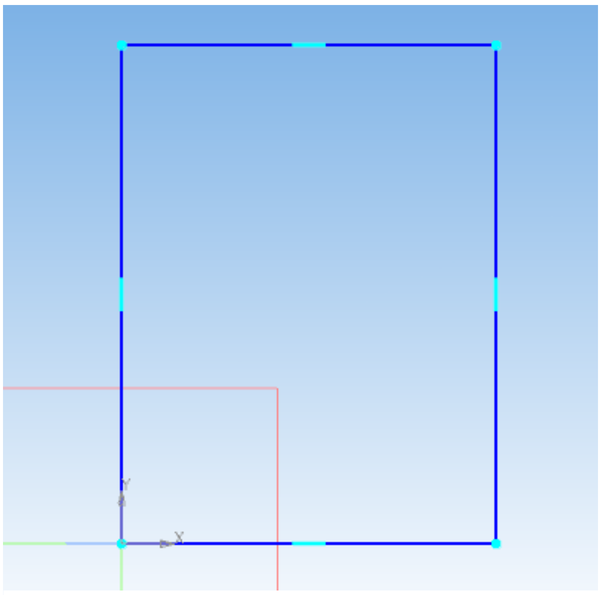

Figure 11. Sketch of body in Kompas- 3D

After creating a sketch, the user switches to “Edit” mode from the toolbox and enable “Extrude” feature which allows to extrude the sketch in the direction perpendicular to its plane. This feature can be set in a special window which allows to choose the direction of extrusion, distance, angle and other properties. There are many settings in this box, which can be used in modelling of a more complex system. So for the chain tightener it wasenough to set the distance and press Enter. After this the program creates a solid object with the needed dimensions.

It can be seen that the procedure has many common points with Solidworks, as a matter of fact being its “clone”, minor differences can be seen only in the visual components of the software.

As for creating the holes it has the same principle. All needed sketches were created on the surface of the body, all the dimensions were done according to the data from the drawings. After drawing the sketches, the feature “Cut- extrude” should be applied. In this feature it is possible to set the dimensions, direction and angles of inclination. After finishing this step the body was almost ready.

The last step was to create the fillets. There is no need to describe this process, as it does not have any notable points.

The body was ready and the result is shown below in Figure 12.

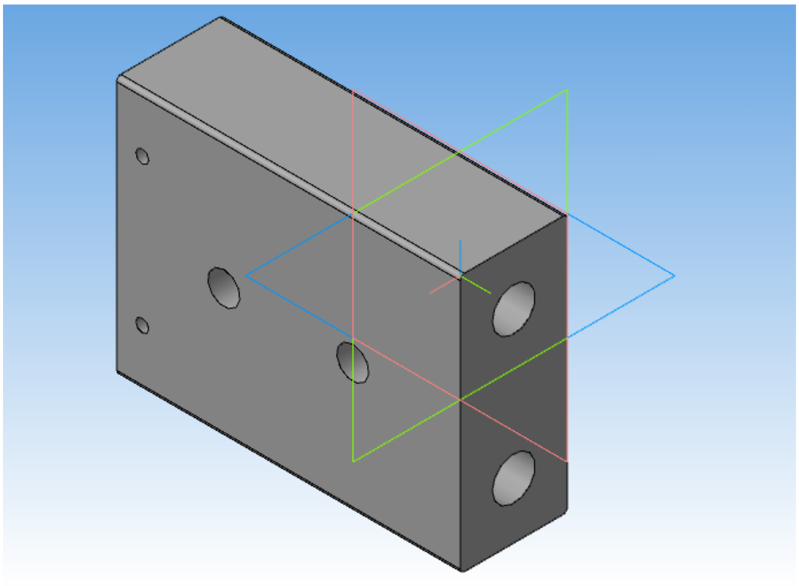

Figure 12. Body of chain tightener in Kompas- 3D

For unknown reasons the process of creation of plastic slide in Kompas- 3D took much more time than expected. It was hard to deal with the geometry of the object but finally the right way was found.

The modelling began with the creation of parallelepiped from rectangular and then the process was similar to the process in Solidworks. The same principle of modelling was used, all the steps were copied from Solidworks. The result is shown in Figure 13.

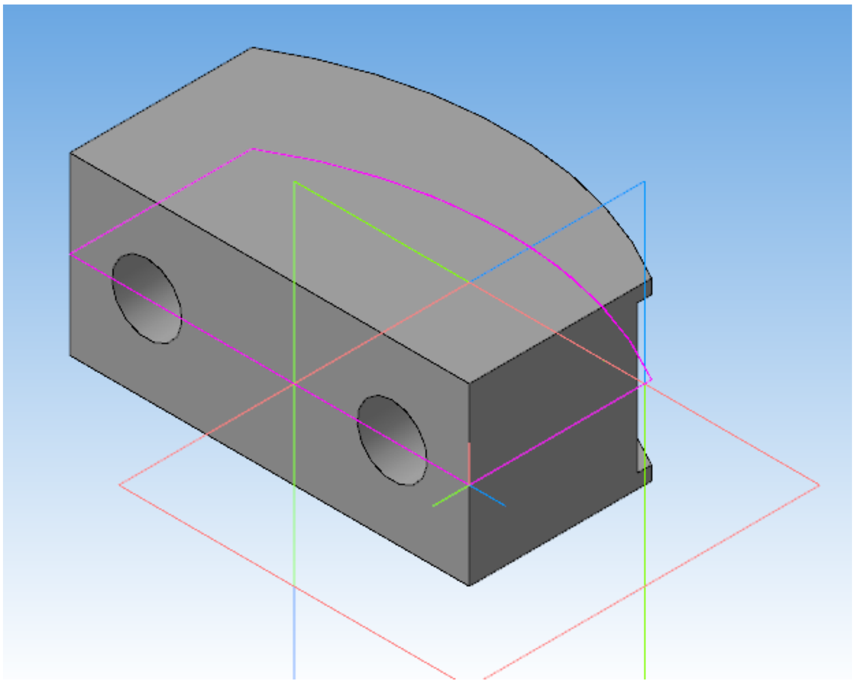

Figure 13. Plastic Slide in Kompas- 3D

In this part the similarity of two software is confirmed again. Kompas- 3D uses all the same principles and operations as Solidworks. It is quite easy to operate at this stage in this program using the experience of working in Solidworks. Nevertheless, Kompas- 3D is not so user friendly and intuitive.

3.3.3 Process of creating a body and plastic slide in FreeCad

Unlike to the previous programs the process of modelling, despite of the similarities in principles is a little bit different and the experience of working in this software is different also.

First of all, it is necessary to create a new part in the launched program. Then the user chooses the workbench which he needs for working. Workbenches in this software are the special sets of tools that are collected inside these workbenches for use in a particular fields of 3D modeling process. (https://www.freecadweb .org/wiki/Feature_list)

There are multiple fields where this software can be used, that is why there are many different workbenches installed by default, such as: workbench for ships design, architectural, robot and others.

One of the biggest advantages of FreeCAD is the possibility to add new workbenches according to the needs of users. For this thesis work workbenches with the tools for fasteners modelling and for assembling parts were added but it will be considered later.

For the creation of the body, it is needed to switch to "Part Design" workbench. Click "Create new sketch" on the toolbox and choose sketch orientation.

Then in the toolbox with primitive objects rectangular was chosen. After creating this sketch it was needed to set the dimensions of the object. For this purpose, the tool for fixing the length of line or distance between two vertex should be taken.

All the dimensions were taken from data on drawings and after setting this information the object with right shapes and dimensions was formed.

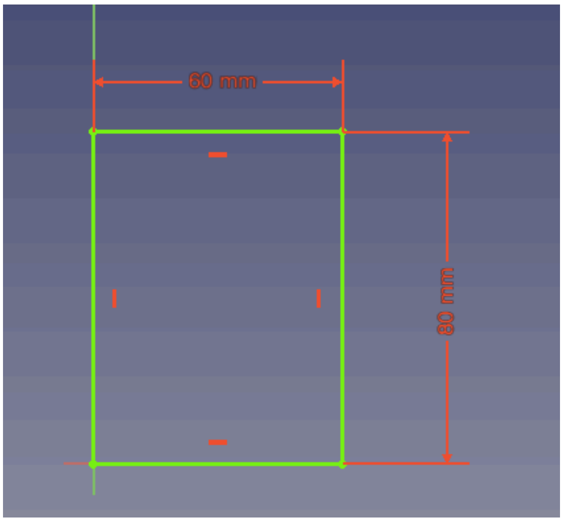

Figure 14. Sketch of body in FreeCAD

After this step the sketch was extruded by the feature which is called “Pad sketch”. Selecting this feature it is enough to put the needed thickness and the object will be ready.

It is clear, that the process of creating a solid body is similar to the processes which were described before. The program has good interface and bright colors that sometimes help to understand the features and shapes of the object better.

Then the procedure is almost equal to two other cases. The program has the same principle and more or less same features for modelling, but in some cases these features have different names from familiar in Solidworks.

As the result the object was ready and had the same features as the original part. It is shown in Figure 15.

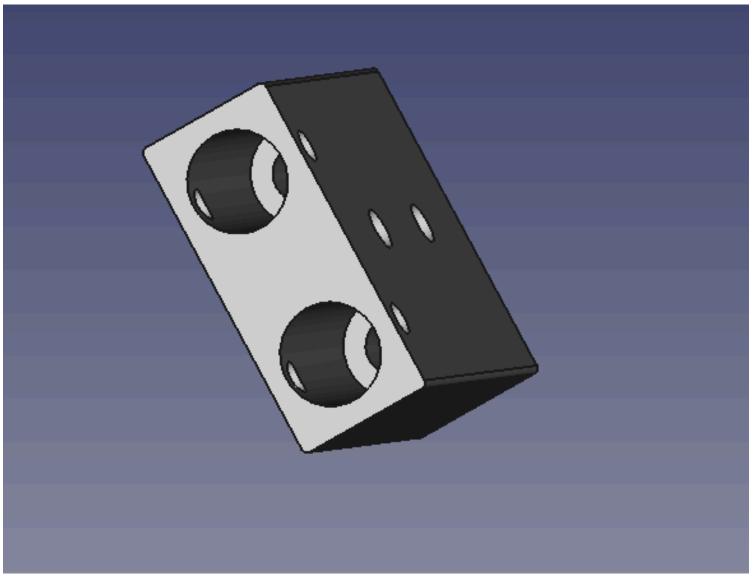

Figure 15. Ready body in FreeCAD

As for plastic slide it did not take much effort as well. The main problem in this part was to draw all the lines correct and set all dimensions and shapes in the right place. Otherwise the procedure was the same. The ready plastic slide is shown in Figure 16. It is clearly seen that the object has the same geometry as the original part.

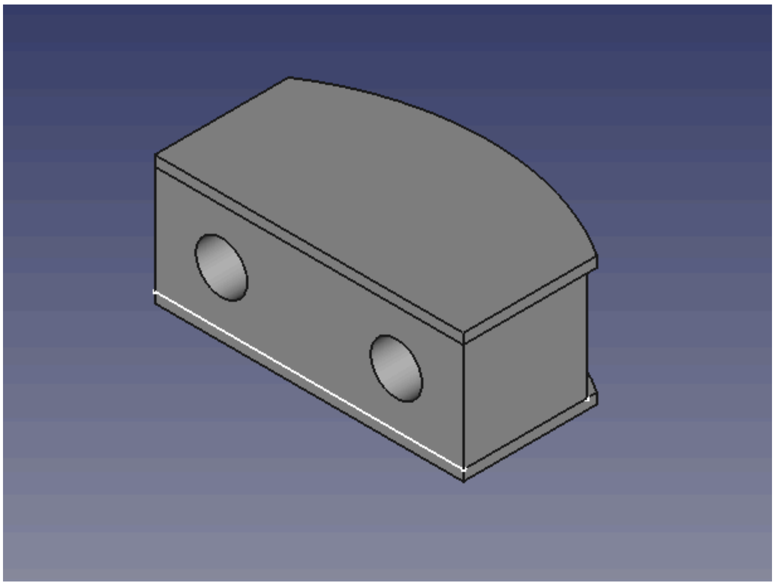

Figure 16. Plastic slide in FreeCAD

3.3.4 Results of modelling of the body and plastic slide

After finishing this step in all three softwares it is clear that the processes of creating simple solid objects in these programs are almost similar. When the principles of modelling are clear in Solidworks, it is easier to use other software, all the difficulty is to deal with the program interface. It takes some time, but after finishing several parts, the understanding comes and it gets easy to operate in different software. The available features in these programs have the same principles of building, the only difference is in process names and locations of the features because of different interfaces.

3.4 Special components

In this chapter the process of modelling of special parts is described. Under these words such parts as springs and bolts are meant. All the similarities and differences are discovered and mentioned below.

3.4.1 Springs in Solidworks

The modelling of springs in Solidworks begins with the creating of basis primitive for spring, which is circle. The dimension of circle should be calculated before creating a sketch. The diameter of circle should take into account the thickness of the spring. The circle is just a skeleton for the spring which will be created after several steps.

For the chain tightener the diameter of the basis was estimated as 16 mm. This diameter takes into account the profile of spring, which will increase the final diameter of the spring. The process started with drawing the sketch, this sketch included a circle with the diameter of 16 mm.

The next is to do a skeleton of the spring. For this purpose, it is necessary to click “Insert” in the Menu, then choose Curve and from the list of options choose “Helix/Spiral”. In the dialog window it is needed to do the settings for the spring choosing the needed parameters. For the chain tightener the settings were estimated as the pitch is constant, it means the distances between the coils of springs remain constant. The pitch was set 4.7 mm and the number of revolutions is 7.5. The start angle of the spring was defined by Solidworks and used by default as 135° and the coils go in clockwise direction.

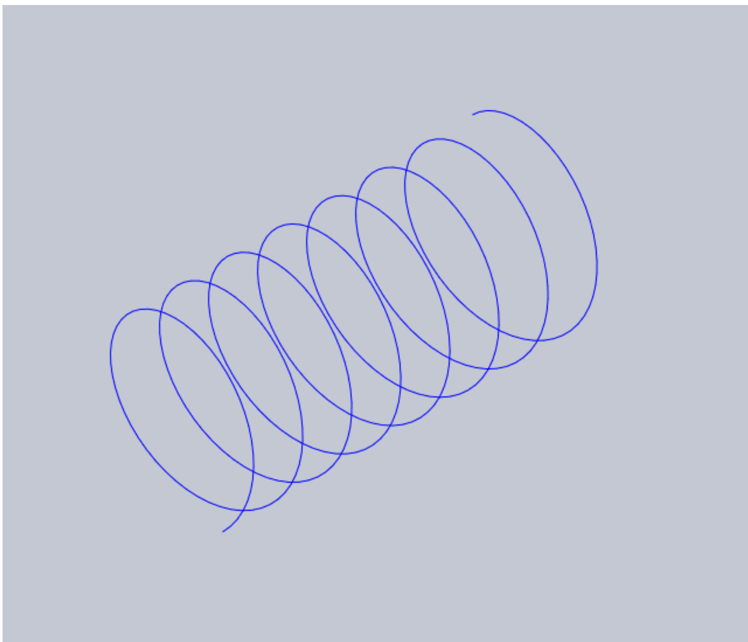

Figure 17. Skeleton of spring in Solidworks

After this the profile of spring, in this case it is a circle, should be sketched at the start point of spring. For this step, it is necessary to switch the orientation to top plane The diameter of the profile for spring was estimated as 2 mm. This dimension was applied by “Smart Dimensions” feature.

The next step was to use “Swept Boss/Base” feature in order to stretch the profile along the skeleton of spring. In the windows of settings, it was necessary to choose the profile for applying the feature, which the sketched circle, and choose the path, which was the skeleton of spring. After this operation the spring was ready. The result is shown in Figure 18.

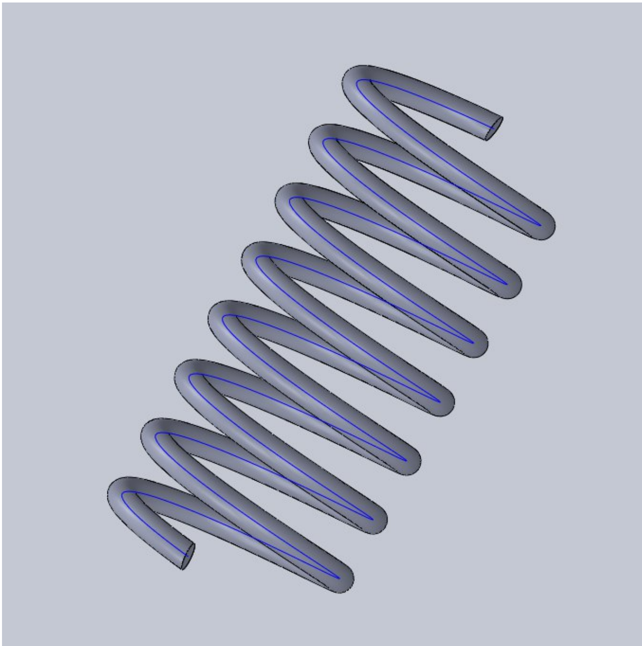

Figure 18. Model of spring in Solidworks

3.4.2 Springs in Kompas-3D

Like the competitors in this thesis, Kompas-3D also has a possibility to do the models of springs and the creation process is quite comparable with the other two programs.

First of all, it is needed to choose the plane in the new “Part” file on which the spring will be created. Then in the tool box the section “3D lines” includes the options which are required, it is called “Cylindrical Spiral”. After taking this feature, the user should set the spiral in the special dialog box. Here such parameters as number of revolutions, step (pitch), direction and spiral side can be set. On the second page of the dialog box the designer should put the diameter of the needed spring. All the dimensions for the spring of the chain tightener were taken from the data, which was used during the similar process in Solildworks. After completing this step the spring is ready.

The next step is to set the thickness of coils. For this purpose it is necessary to draw a profile of the spring by creating a new sketch. The process is the same like it is in Solidworks and FreeCAD. Then the profile can be stretched out by the feature “Sweep”, which is located in the group of “Extrude”. In the dialog box the user should choose the profile of the spring and the object which will be the path for sweep. Then the spring was ready.

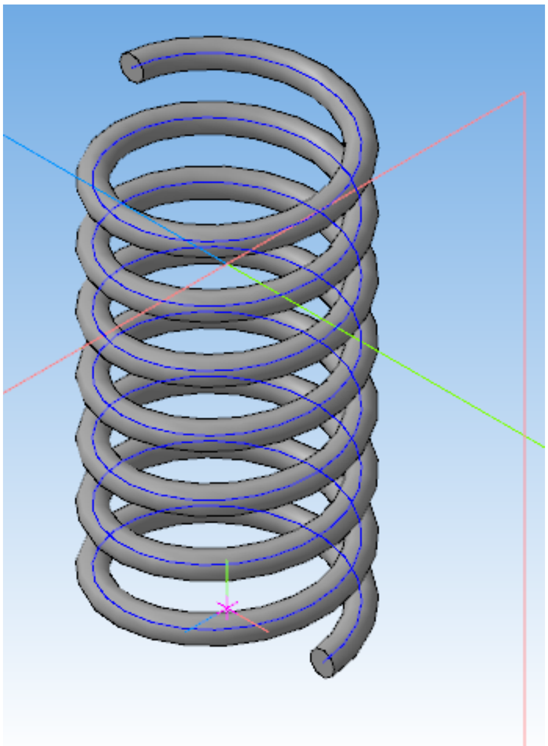

Figure 19. Spring model in Kompas- 3D

3.4.3 Springs in FreeCAD

As for creating the springs FreeCAD is also a good tool in order to reach the good result and get the spring which is needed for the designer. The modelling process is not much different from the other two programs, but it has its own peculiarities, which are described in this chapter.

The designing process begins in workbench “Part”. In the toolbox with geometric figures it is necessary to choose the section “Creation of parameterized geometric primitives”. From the list of option “Helix” should be taken. Then it is possible to parameterize the feature of the future spring. The settings kit is almost the same as in other two programs, the only difference is in the names of some parameters. Also instead of the number of revolutions the height of spring should be inserted. This is not critical, just the matter of habit. This value can be easily calculated by multiplying the number of needed revolutions by the pitch which is taken for the spring. In case of chain tightener the needed height is 32.25 mm.

After completing these settings the designer gets the skeleton of the spring. Then it is necessary to build up the volume for this skeleton. For this purpose, it is needed to switch to "Sketcher" workbench. This workbench gives more opportunities for creating and editing the sketches. After choosing the needed plane the circle with the radius of 1 mm on the distance of 8 mm from the origin was sketched. In this case the center of circle was exactly coincided with the beginning of the coil spring, this is very important condition, because it was taken into account later on in the process of creating the thickness for the coils of spring.

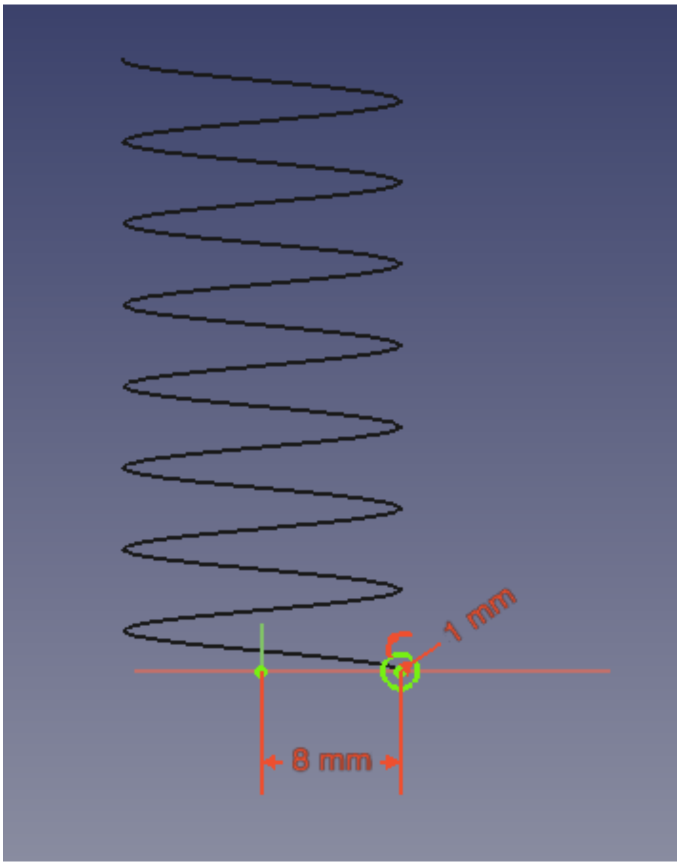

Figure 20. Creation of spring in FreeCAD

The next step should be done in “Part” workbench. “Utility to sweep” is the tool which has to be used, this tool allows to create a solid shape from the profiles, projected along a path. (https://www.freecadweb.org/wiki/Part_Sweep)

While applying this feature the designer should take the object which will be used for sweeping and the pass for this operation. So in case of spring the object for sweeping is the sketch of circle which was created in the last step and the sweep path is the skeleton of spring. After doing these settings, the spring was ready. The result is shown in Figure 21.

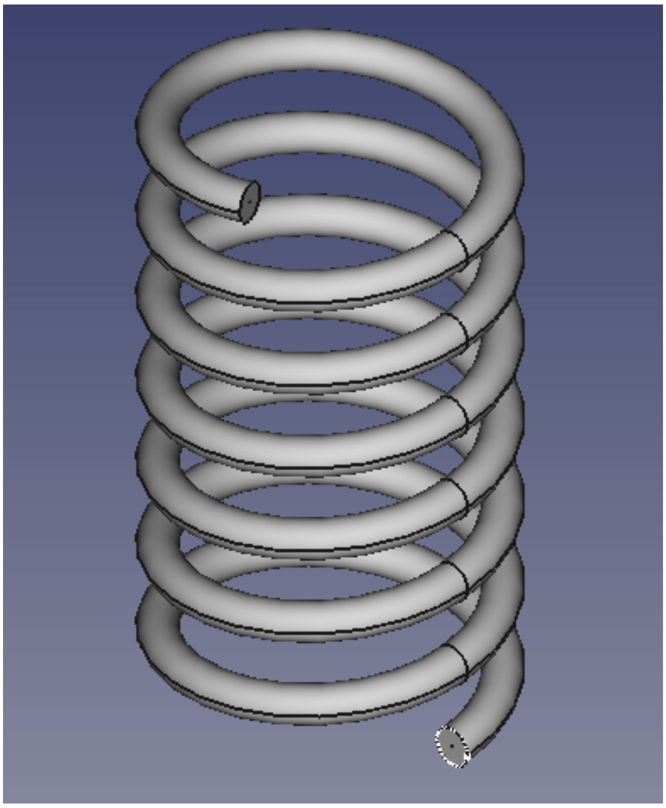

Figure 21. Model of spring in FreeCAD

3.4.4 Modelling of bolts in Solidworks

This step was the easiest during the project, because there is a library with all needed fasteners in version of Solidworks which is installed on the computers at Saimaa University of Applied Sciences.

This library includes the fasteners with standards for Europe (ISO), Germany (DIN), Japan (JIS), South Korea (KS), India (IS), China (GB), USA (ANSI, MIL), Australia (AS).

The library includes different types of tools: bearings, bolts and screws, keys, nuts, o- rings, pins, power transmission, structural members and washers.

So there was no need to create the bolt for the tested chain tightener, it was taken directly from the library during the assembling process.

In case of tightener the bolt with needed dimensions was found fast. It was Hex Socket Cheese Head ISO 14580 bolt with the following features: #4 x 25 x 23.6 – 4.8-N. The properties of the bolt are the following: size 4, length 25, thread length 23.6.

3.4.5 Modelling of bolts in FreeCAD

By default, there are no special tools for fasteners modelling in FreeCAD. But one of the biggest advantages of FreeCAD is the possibility to download add-ons and install them, in case of this program the solution is to add a special workbench, which would include the needed tools for modelling of fasteners.

After doing a research on the Web the right workbench was found, it was “Fasteners” workbench. Adding this add- on gives an access to the kit of bolts and nuts. While choosing the right option the program offers to choose the size of bolt and set it in accordance with needs. Most of the bolts have ISO standard, but there are also EN and DIN standards.

This a big advantage for mechanical engineers and this feature gives the point to the rating of FreeCAD.

In case of chain tightener the same bolt ISO 14580, which was chosen in Solidworks library, was found and taken. The properties were set with the same dimensions.

3.4.6 Modelling of bolts in Kompas-3D

There is no special feature for modelling the fasteners in the English version of Kompas- 3D. That was the first and biggest minus of Kompas-3D for the moment.

This situation can be explained by the fact, that Kompas-3D was created by a Russian software company. The program is more focused on Russian customers, so the fasteners which could be available for users, would use the Russian standard GOST. It is possible, that for this reason the feature is not available in the current international version of the program.

The primitive model of the bolt was created in order to get the needed part. This model followed all the needed shapes and dimensions compared to the models of the bolts which were taken in Solidworks and FreeCAD.

3.4.7 Results of modelling of special parts

In this step the difference between the programs was more noticeable than in the previous one.

As for spring modelling all three programs have the needed feature, but it is implemented a little differently. The modelling processes in FreeCAD and Kompas-3D look easier and faster than in Solidworks, it requires to do less steps and the process is more simplified, the feature organization is better.

In case of fasteners the situation is different. Solidworks has the biggest library of fasteners, which are grouped according to the standards and types. It offers the wide range of different bolts, nuts, washers and pins.

At the same time FreeCAD also has the similar feature, it is realized in special Workbench, which gives the positive experience of use. The quantity of options is enough for regular users, the fasteners can be set depending on the needs.

In contrast to competitors, Kompas-3D does not have the similar feature in the international version of the program. It is a major drawback with which it is necessary to put up, remembering all the advantages of this program. To solve this problem there is a list of guides how to create the models of bolts and nuts on the Web.

3.5 Creation of pin, shaft and press cap

The process of modelling of the pin and shaft can be combined into one chapter, because the procedure was the same for three tested programs.

The process started from creating a sketch. The basic profile for both components was the circle. In case of pin the diameter of the circle was 3 mm and for shaft 10 mm. Then these circles were extruded in accordance with the needed lengths, 25 mm for pin and 85 mm for shaft. The processes of extrusion for Solidworks, Kompas-3D and FreeCAD were described before, so there is no difference for these parts.

In case of the shaft, the hole for the bolt was done by “Cut-Extrude” feature. The principle of this feature is clear for all software and can be neglected in this chapter.

As for press the cap, it was decided to use the same principle of modelling for all software. The procedure is described below.

The first step was to do a sketch and draw the profile of cap, taking into account the shapes and the geometry of object. The sketch was the same in all programs.

After doing the profile of cap, the next step was to twist this profile around the vertical axis. Solidworks, Kompas- 3D and FreeCAD organized this feature in the same way. The operation is called “Revolve”. Choosing this feature it is necessary to select the object for revolving, axis and angle of revolving, in case of press cap the angle was 360°. After applying this feature, the object was ready.

3.6 Assembly

The next step after creating the parts is to assemble these components. This step is very important, because on this stage the parts are joined in one system. During the mating process the designer defines the possible miscalculations which were done during the designing process. It happens if the geometry of objects is wrong, in this case it is necessary to go back and do the changes in 3D models of components, which are used in the system.

After the creation of an integrated system that includes all components of object, it is possible to check the features of body, how the parts move and interact with each other.

In this chapter the feature of assembling is discovered, described and explained.

3.6.1 Assembly Solidworks

In case of Solidworks, the program has the special workspace for doing the assembly.

First, the designer thinks over the sequence of joining parts together and how to organize this process. This plan helps to simplify and speed up the process.

In case of chain tightener it was decided to start the process with the mating of the body and the press cap. For this purpose, it was necessary to add these objects by clicking “Insert Components” in the toolbox “Assembly”.

Then the parts should be mated by selecting the appropriate feature “Mate” in the toolbox. In this steps it is needed to choose the surfaces for mating and type of mating. For chain tightener it was enough to use “Coincident” and “Concentric” types of mates.

In case of “Coincident” mates selected objects will be positioned on the same infinite plane. In case of “Concentric” the chosen objects which have in its basis a circle will be positioned in the way, that they will share the same centerline. (http://www.webpages.uidaho.edu/mindworks/Solidworks/Topic%20References/Types%20of%20Mates.pdf)

To do the assembly of body and press cap these two types are needed. First they can be connected with “Concentric” type of mating, after this step the centerline of hole will be joined with the centerline of press cap. Then the components should be connected by applying “Coincident” mating, after this the surfaces of parts are joined.

The next step was to insert the springs at the same holes. For this purpose, the same types of mating were used. Then the shafts were inserted from the sides. After this, the bolts were placed in their places. Finally, the plastic slide was placed on the free sides of shafts and two pins closed the holes from the sides of springs. All these mates used just two types of mates: “Coincident” and “Concentric”.

After these steps the assembly was ready. It was clear that the design of parts was correct, because all the components were connected and joined in the appropriate way. The assembly of chain tightener is shown in Figure 22.

The feature of assembling in Solidworks is designed in an organic way, giving the possibility to work with the parts efficiently, combining them in the needed way.

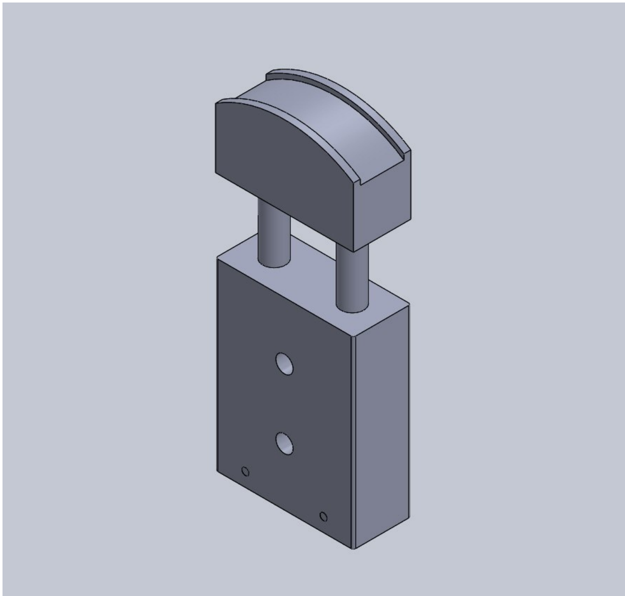

Figure 22. Assembly of chain tightener in Solidworks.

3.6.2 Assembly Kompas-3D

Assembly feature in Kompas-3D is realized in the way similar to Solidworks. For this purpose, there is a special template in the program. In this workspace all the needed tools for assembling are collected. The operational principle is the same also.

First, the user should add the needed components for assembling and then it is time to launch the mating procedure by clicking “Mating” in the toolbar.

At this stage the program works on the principle of Solidworks, it is necessary to choose the surface or contours of the parts and decide which type of mating should be applied. For chain tightener two methods of mating were used:

“Coaxial” and “Coincidence of objects”. These types are the alternative options from Kompas- 3D, as they act in the same way as “Coincident” and “Concentric” mates in the previous chapter.

For the case which is under consideration in this thesis, the sequence of operations was the same. The process started with assembling of the body and press cap. Then the parts were assembled in the following way: springs, shafts, bolts, plastic slide and pins.

As the result the working mechanism was assembled, it is shown in Figure 23.

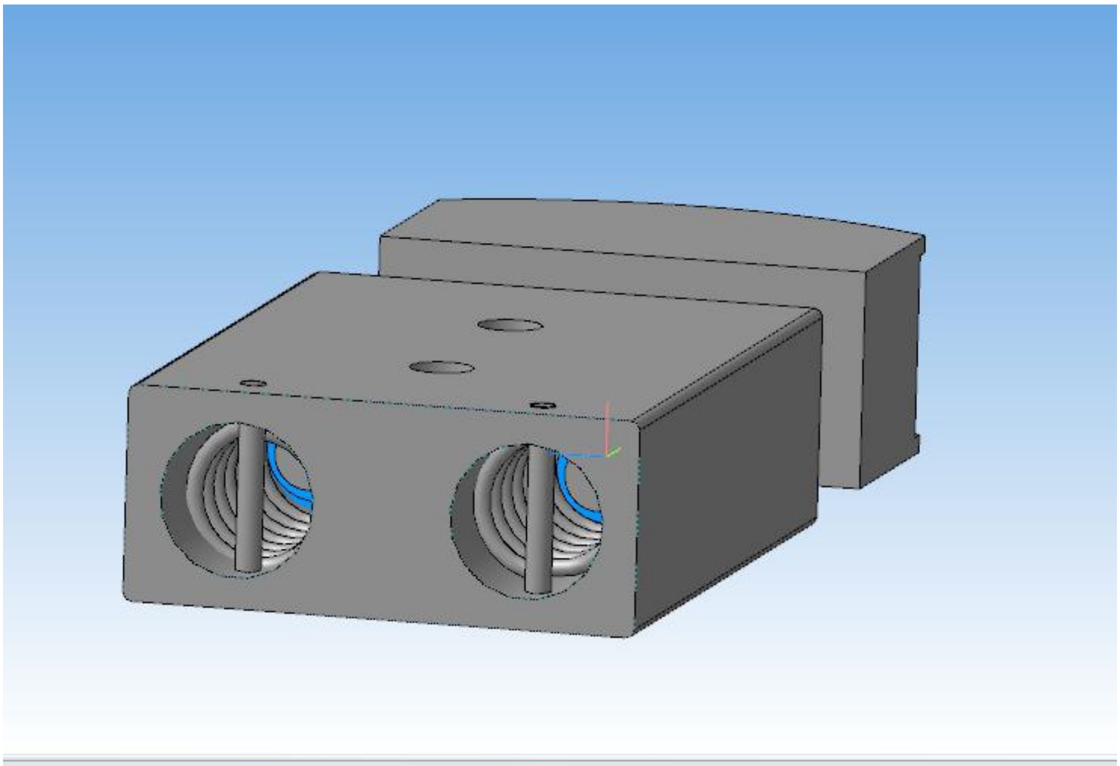

Figure 23. Chain tightener assembly in Kompas- 3D

3.6.3 Assembly FreeCAD

In case of FreeCAD there is no special feature for assembling in the workbenches by default. For this purpose, it is necessary to download a special workbench on the official website. The name of the workbench is “Assebly 2”. This workbench gives the standard possibilities for assembling and joining of parts in one system.

The workbench has the same features and settings for assembling as Solidworks and Kompas-3D.

The process begins with adding of parts which have to be mated. Then the mating surfaces and mating type should be selected.

There are five types of mating available in “Assembly 2”:

- Circular constraint
- Plane constraint
- Axial constraint
- Angular constraint
- Spherical surface constraint

For chain tightener it was enough to use “Circular constraint” and “Plane constraint” mating types, the rest types are being used for more complex systems with complex shapes and for more advanced mating.

In case of “Circular constraint”, it is used for objects, which have a circle in the basis, the parts will be mated in the way, that they will have the same centerline.

“Plane constraint” is the method of mating which is applying for the object which should be placed on the same infinite plane.

The sequence for mating the parts was the similar to the steps which were used in Solidworks and FreeCAD. The base part was the body, and then all the parts were assembled in a certain way, beginning from the press cap and finishing with the pins. The ready assembly is shown in Figure 24.

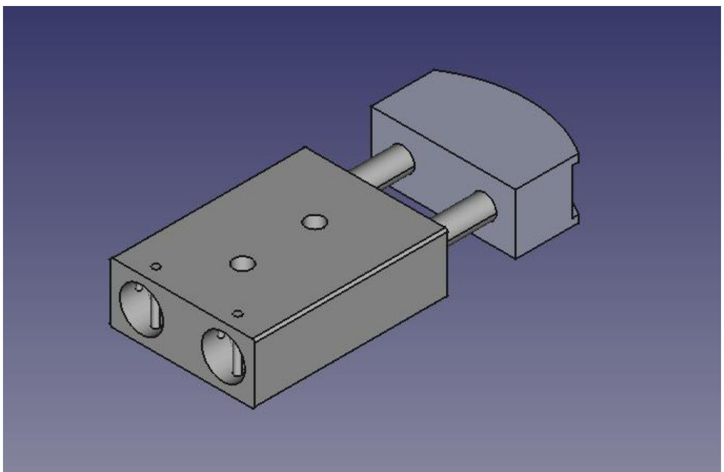

Figure 24. Chain tightener assembly in FreeCAD

During the process the biggest challenge was to mate the springs with the press caps. All types of mating were checked, but no one was the suitable for these objects, so the objects were placed inside using the coordinates system, the coordinates were adjusted for all 3 axes in the way, that the spring was placed in the appropriate place.

3.7 Drawings

The last feature that is compared in this thesis is “Drawings”. The process of creating technical drawings is a very important stage in 3D modelling. A technical drawing is the guide for an object which has to be built or manufactured. It gives the detailed visual representation of the object. The drawings include all needed information for manufacturers, shapes, dimensions, materials which are, the quantity of each part and other important features of object. (http://www. indovance.com/importance-technical-drawings/)

In this chapter the process of creating the drawings is described.

3.7.1 Drawings in Solidworks

The feature of creating the drawings in Solidworks is done very conveniently and organically.

When the part is ready, it is necessary to choose “New” menu and then click “Make Drawing from Part/Assembly”. Then the designer should choose the sheet format, this format depends on the drawing. In case of chain tightener, for the parts and assembly drawings A4 was the optimal format.

After choosing the format, the designer should take the needed views of object from “View Palette”. For simple parts one or two different views are usually enough, front view and top view for example, but for a complex object sometimes even three views or more are needed. For complex objects with inner details and shapes, the designer can use “Broken- out section” feature, which helps to show all the inner details by cutting off the needed quantity of material. During the process of making the drawings for chain tightener this feature was very useful for such parts, as the body, shaft, plastic slide and press cap. All these objects have the inner geometry inside, so it was not possible to create the right drawings without this feature.

The next step was to put all the needed centerlines and center marks, following the standards of technical drawings. Then the turn of dimensions, they should be put by “Smart dimension” feature.

The last step was to fill the documentation. At this point the information about the person who made the drawing should be mentioned. Also it is possible to add the data about the weight of object, material, drawing number, page format and scale of drawing.

After this stage, the drawings were ready. The process of creating the drawing from part was very logical and clear in Solidworks. The program gives positive experience of use and it is easy to learn this process in short period of time in order to create correct drawings.

Finally, it is necessary to create an assembly drawing. The assembly drawing includes the drawing of the assembled mechanism, showing the main dimensions and geometric features. Also the drawing should include the bill of materials, that is a list of parts and quantities of these parts, which are used during the manufacturing process in order to get the product.

The process of creating the assembly drawing was similar to the process of creating the drawing for the part. From “view pallet” two optimal views of chain tightener were chosen. For both views it was decided to use “Broken- out section” in order to show all the inner components, because almost all components are located inside of the body and the plastic slide. Then all the needed dimensions were inserted.

The process of creating a bill of materials begins with inserting special balloons. These balloons help to mark the parts in assembly and assign each part with own item number, which is used in the bill of materials. The feature can be applied automatically, using “Auto balloons” feature or manually by “Balloons” clicking on the parts of object with the mouse. For chain tightener it was decided to use “Balloons” feature in order to get a more accurate drawing and place the balloons in the best places. Then the bill of materials was added by choosing “Insert” menu and then “Tables”- “Bill of Materials”. The table was edited automatically and included information about the item number, which was taken from the balloons, part name, description and quantity of each parts. The last step was to fill the needed information in the documentation block.

At this stage all the drawings in Solidworks were ready. Soliworks has the perfect model for creating all kinds of drawings, having intuitive interface, suitable features and tools for students at Saimaa University of Applied Sciences in order to satisfy their requirements and needs.

The drawings are attached in Appendix 1.

3.7.2 Drawings in FreeCad

As for standard workbench ”Drawing” which is available in FreeCAD by default, it is not suitable for making technical drawings, because the tools which are available in this workbench do not allow to create good technical drawings.

In order to solve this problem, the research of all available workbenches was done. It was defined that there is no workbench with the assembly drawings feature at the moment. The only possibility was to do the drawings for the parts of the chain tightener. The process of creating the drawings is described below.

First of all, after the research it was decided to install “Drawing dimensioning” workbench. This workbench has more features for the drawings, compared to the standard one.

After installing this add-on, the user should open the needed part and switch FreeCAD to “Drawing dimensioning” workbench and then in the toolbox it is necessary to press “create new drawing” and choose format of page. For drawings of parts of chain tightener “A4 (ISO7200)” page format was taken. ISO7200 is an international technical standard which is used for technical drawings.

Then in the dialog box the drawing has to be set. In the dialog box the designer can choose the angle of projection, secondary views, scale of drawings and some other minor settings. In the begging, it was decided to do the drawing of the body. It was enough to use four different views of the object in order to show all important features.

At the moment one more drawback of FreeCAD was discovered. There is no possibility to do a section view of object. This is a critical point while making a drawing for a body of chain tightener, because this part has the hole which should be displayed in the drawing in a special way. So it is not possible to do the correct drawings in FreeCAD for the objects with complex shapes.

After setting the views it is necessary to put all the needed dimensions, center marks and centerlines choosing the measuring tools which are available. This feature realized in the right way and it is easy to put all the dimensions. This step was done fast.

The last step was to put the information about the author of the design, part number, date, scale and supplementary information on the paper. This was the easiest part, because the information can be edited in the special window.

Summing up, the feature of drawings is developed poorly. FreeCAD does not give the needed experience and it is not possible to get quality drawings using available workbenches.

The example of drawing is attached in Appendix 2.

3.7.3 Drawings in Kompas-3D

The process of creating drawings leaves mixed feelings. On the one hand, Kompas-3D has enough features to create good drawings, but on the other hand, this process is quite complicated, some features remain obscure even after viewing the guides and tutorials on the Internet.

The process starts after creation of the part. It is necessary to create a new file and choose “Drawing” from the list of options.

After this the user is transferred to the workspace with sheet for drawing with format A4, this sheet has the same template as it is in Solidworks. Then the user should insert the views of object by choosing this feature from toolbox. The next step is to set them, choosing the needed views from “View Pattern”. Also it is possible to set the scale, choose the main view orientation and color of views. When the views are set, the user can put the dimensions, using “dimensions” tool bar.

As for simple parts, the procedure is almost similar to Solidworks. But the story is totally different, when it is dealing with complex parts with inner details. There is a special feature “Detail Section”, but the principle of work was very difficult to understand. After several hours of useless work, it was decided to find other ways in order to create the needed views, on which it is possible to show the inner details of the part. Then the solution was found on the forum of Kompas-3D users.

For this purpose, files with the parts which have inner geometry were copied and then modified. The idea was to cut a part into two symmetrical pieces and remove one of the pieces, then insert the view of cut part in the drawing.

To realize this idea, it was needed to create a new sketch on the surface of part and to draw a cut line. For this step “Polyline” tool was taken from “Geometry” toolbox. After this the feature “trim” from “Operation” toolbox was used. This feature helps to cut off extra material using the sketch which was done in the last step as a cutting line.

Then the desired “section” view can be added in the current drawing, using the same procedure, which was described above. This trick was used in the processes of creating the assembly drawing, body and plastic slide.

The next step was to arrange the drawings properly, by adding the needed centerlines, center marks and hatch. For these needs Kompas- 3D has special tools, which makes this process simple. As for centerlines and center marks, these tools can be chosen from “Designations” tool bar, and hatching feature can be applied by choosing this tool in “Tool” menu.

The last step was to edit the documentation block in the bottom of sheet. This operation can be easily done by two left clicking the mouse in the needed block.

Summing up, the process of creating the drawings from part is almost the same as it is in Solidworks, otherwise it is quite difficult for parts with complex shapes, which need additional views and features.

The final step was to do the assembly drawing. This drawing required two special views with bisected chain tightener. For this purpose, the operation with changing 3D model of assembly by cutting off the half object was applied two times in order to get the needed views.

The procedure was the same, as it was for drawings of parts by the last step. At the last stage it was necessary to do the bill of materials. The program has a special tool for this feature.

First of all, it was necessary to create a new document, choosing “BOM” type. Then it was needed to take “Assembly management” feature in order to connect the bill of materials and drawing. The needed drawing was chosen, after this step the created file was connected with the assembly drawing.

The next step was to create references numbers for each item of assembly, this tool was taken from “Designations” tool bar. This feature is similar to “Balloons” tool in Solidworks. The circles with numbers and arrows, pointing to the parts appeared. At this stage it was necessary to set every reference number by clicking the right mouse button and choose “Add BOM item”. After this the task was to set the item, by choosing its type, in case of chain tightener it was “Part”, then the information about the item should be filled. It was possible to choose the quantity of item, name and some description. After saving these settings, file with the BOM was modifying automatically and all parts were indicated in it.

The process of creating the assembly drawing was quite simple but a little bit dreary, because in order to get the final drawing many steps were done. So the developers of Kompas- 3D should take the steps in order to simplify this process and do it more clear and user- friendly.

The examples of drawings for chain tightener in Kompas- 3D are attached in Appendix 3.

4 Evaluation of obtained data

It is obvious that each program has both advantages and disadvantages. In case of Solidworks, FreeCAD and Kompas-3D this statement was confirmed. During the work on the thesis it was possible to reveal the features of each of the programs by the example of creating a 3d model of the chain tightener. Such features as part modelling, special parts modelling, assembling and drawings were discovered in the appropriate way. In this chapter all received information is arranged and compiled in a comparative table. All features were evaluated on a five-point scale, from 0 to 5, where:

0 – feature is missing

1 – poor

2 – satisfactory

3 – good

4 – very good

5 – perfect

All the marks were exposed based on the feelings and experience gained during the use of these programs, depending on ease of use, time spent for operations and level of implementation of the function.

Also the table includes such features as multiplatform of programs, availability of training materials on the Internet and stability of running programs. The last point relates to the number of bugs and the forced closure of the programs. These features were evaluated from 0 to 2, as it is the minor features, where 0 is the lowest grade for feature, 2 is the highest. The results are displayed in Table 1.

	Grade		
Feature	Solidworks	FreeCAD	Kompas-3D
Parts modelling	5	4	4
Special parts modelling	4	5	3
Assembling	5	4	5
Drawings of parts	5	2	3
Assembly drawings	5	0	3
Multiplatform	0	2	0
Training materials and tutorials	2	1	1
Program stability	1	1	2
Total:	27	19	21

Table 1. Comparison of features.

Summing up the grades from the table the results are the following, Solidworks has 27 points out of 31, FreeCAD has 19 and Kompas–3D has 21 points.

5 Summary

The goal of the study was to do a research and get knowledge about three different 3D modeling Software and to figure out the best option for Mechanical Engineering student at Saimaa University of Applied Sciences and also identify the advantages and disadvantages of each program. The purpose was reached and the summary is below.

Solidworks, FreeCAD and Kompas- 3D are the programs with approximately one set of basic features needed for students of Saimaa University of Applied Sciences. Each of them has the right to exist, having both advantages and disadvantages. But one fact is obvious, Solidworks is the most balanced software among the competitors in this study. During the modelling process of chain tightener, Solidworks passed the test, while the creation of simple basic parts, and components of increased complexity with complex geometry and shapes. During the assembling process the program was stable, providing with all needed features for mating and placing the components. At the final stage of creating the drawings Solidworks showed itself in the best way and showed superiority over competitors, giving the designer a complete set of necessary features for creating drawings of parts and assembly drawings.

On the other hand, FreeCAD and Kompas-3D were also quite good. Each of them has its own advantages, in some aspects these programs were even better and more stable than Solidworks.

In case of FreeCAD, the main advantage of the program is the availability of different additional workbenches, which can be downloaded by the user and have special features and possibilities and at the same time it is a free software. Another strong point of FreeCAD is multi- platform, which gives an opportunity to work on the project, using Windows, Linux and macOS platforms. FreeCAD has a nice and convenient interface. But at the same time, the workbench with drawing is developed poorly at the moment, so it was not possible to do good drawings, assembly drawing feature is not available at the moment. This is a critical point of the program, that crosses out all the merits of this program and makes its use impossible for the students at Saimaa University of Applied Sciences at the moment.

As for Kompas- 3D, this program has a similar interface to Solidworks. The features are realized almost in the same ways. The only problem during the use was the absence of library with the fasteners, but maybe this problem is urgent only for the English version of the program, because in the first place the program is oriented for Russian users. Also during the process of creating the drawings some problems were associated with the incomprehensibility of the principle of work of some features. So it was needed to find other ways in order to solve this or that problem. The appearance of the program looks quite old and requires some improvements. The advantage of the program is that it can be used for free by students. Summing up, it is a good option for designers who are looking for a reliable program with a basic set of functions and capabilities. The program can be quite popular if the developers make an effort to reach more users around the World and develop the international version, by adding the necessary functions, improving the appearance and simplifying some features.

So Solidwrorks takes the first place in this test, the program confirms its leading position in the market.

6 Conclusion

Nowadays there are more and more programs for 3d modeling, which have their own special functions and features. This process favorably influences already existing and popular programs, forcing developers to improve their products. Also the development of mobile technologies opens for developers good prospects for the implementation of their programs in mobile platforms. All this makes 3d modeling better, by simplifying and improving the process. A tremendous push can occur after the complete mastering of virtual reality technologies by people. The 21st century is a time of great technological changes and the current generation is fortunate to witness these changes. It remains to be hoped that new technologies will help people reach a new level of development and make a huge step forward in the development of new branches of science and technology.

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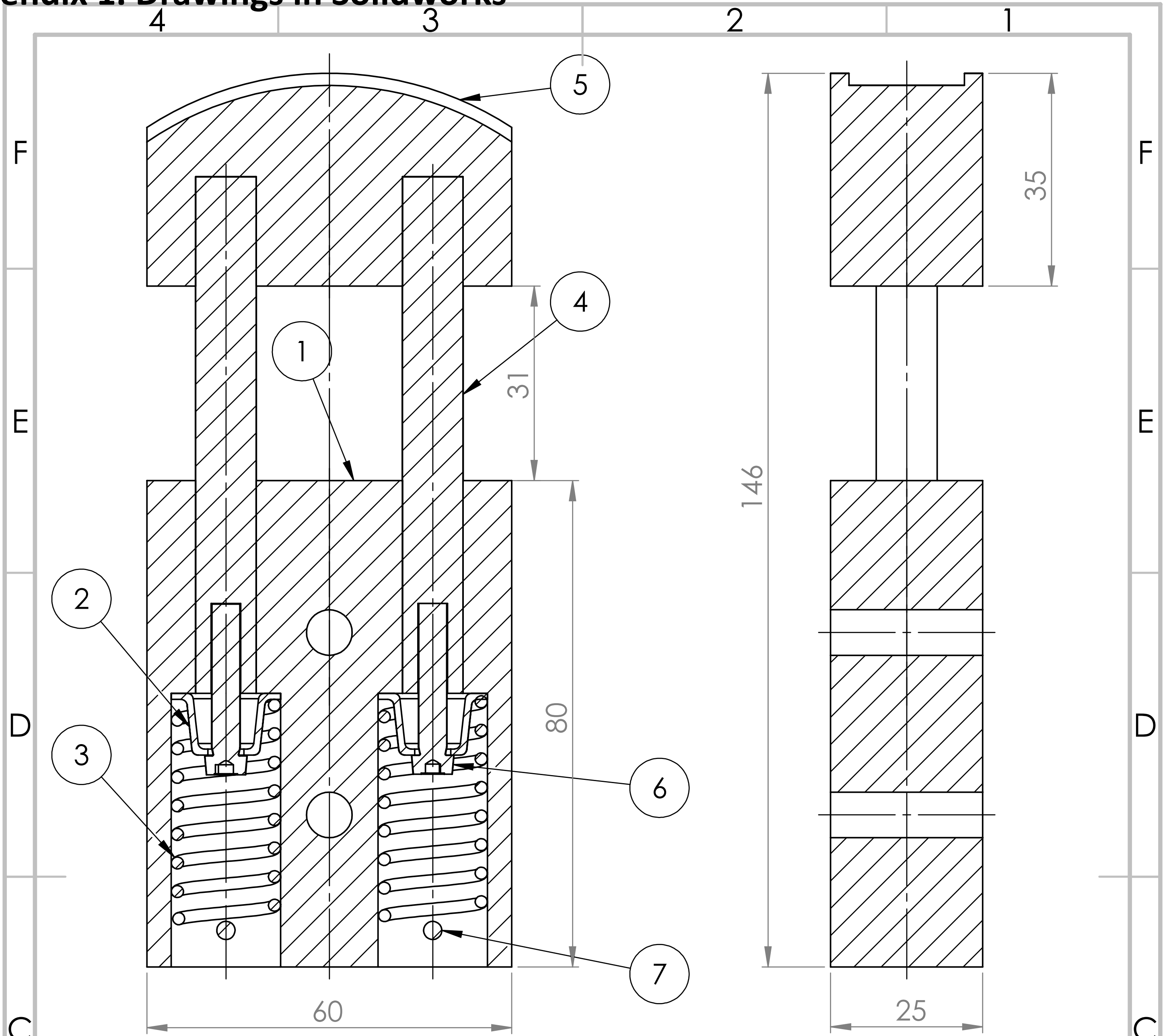
List of Figures

List of Tables

Appendix 1. Drawings in Solidworks



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Body		1
2	Press Cap		2
3	Spring		2
4	Shaft		2
5	Plastic Slide		1
6	ISO 14580 - #4 x 25 x 23.6 - 4.8-N		2
7	Pin		2

UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:

FINISH:

DEBURR AND BREAK SHARP EDGES

DO NOT SCALE DRAWING

REVISION

	NAME	SIGNATURE	DATE
DRAWN	Denis Bobylev		
CHK'D			
APPV'D			
MFG			
Q.A			

TITLE:

MATERIAL:

DWG NO.

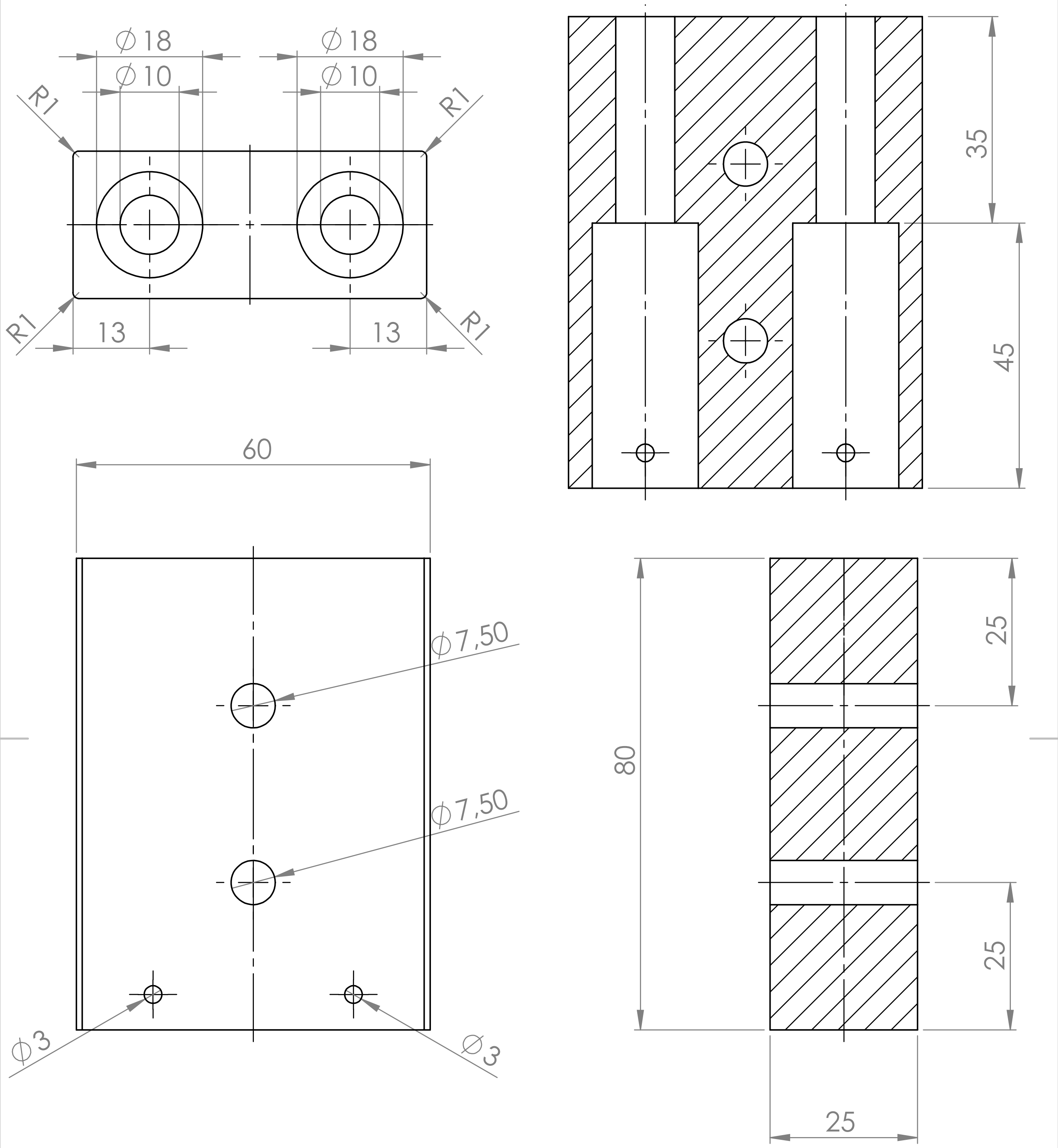
Assembly

A4

WEIGHT:

SCALE:1:2

SHEET 1 OF 1



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TOLERANCES:
LINEAR:
ANGULAR:

FINISH:

DEBURR AND BREAK SHARP EDGES

DO NOT SCALE DRAWING

REVISION

	NAME	SIGNATURE	DATE
DRAWN	Denis Bobylev		
CHK'D			
APPV'D			
MFG			
Q.A			

TITLE:

MATERIAL:

DWG NO.

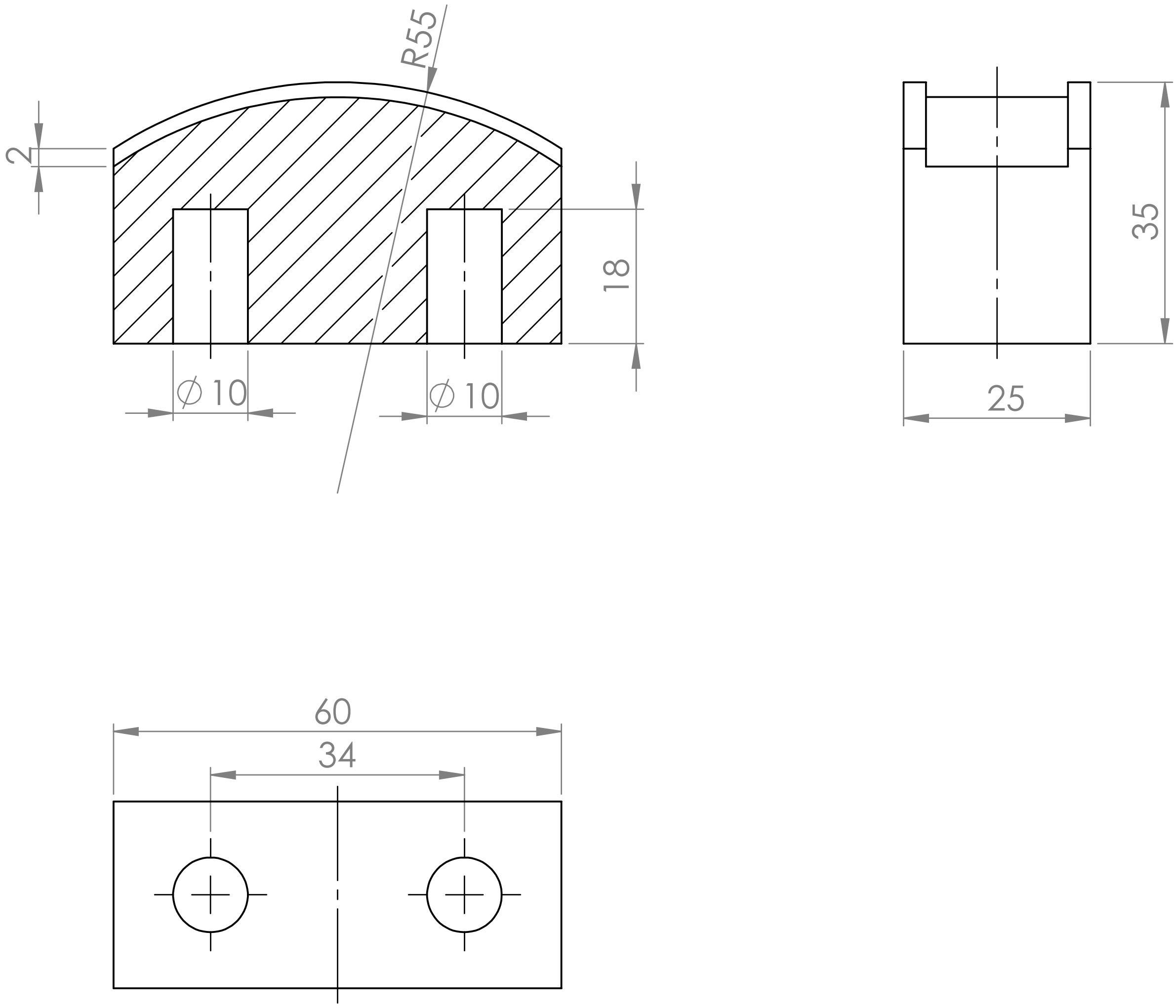
BODY

A4

WEIGHT:

SCALE:1:1

SHEET 1 OF 1



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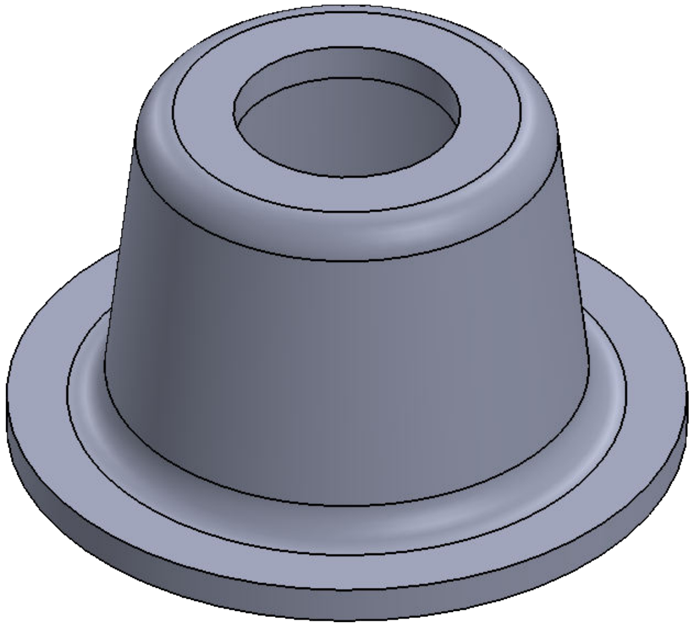
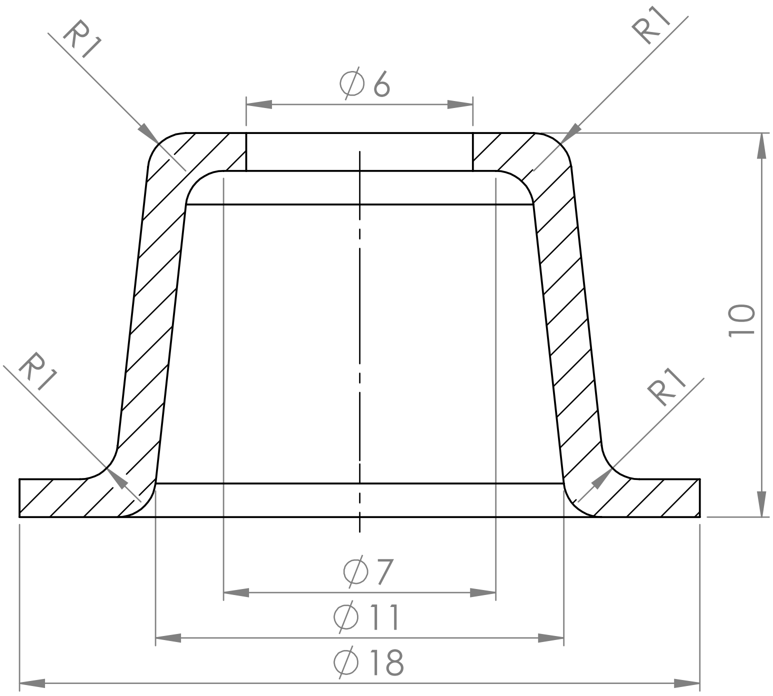
	NAME	SIGNATURE	DATE		TITLE:	
DRAWN	Denis Bobylev					
CHK'D						
APPV'D						
MFG						
Q.A				MATERIAL:	DWG NO. Plastic Slide	A4
				WEIGHT:	SCALE:1:1	SHEET 1 OF 1





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TOLERANCES:
LINEAR:
ANGULAR:

FINISH:

DEBURR AND BREAK SHARP EDGES

DO NOT SCALE DRAWING

REVISION

	NAME	SIGNATURE	DATE
DRAWN	Denis Bobylev		
CHK'D			
APPV'D			
MFG			
Q.A			

TITLE:

MATERIAL:

DWG NO.

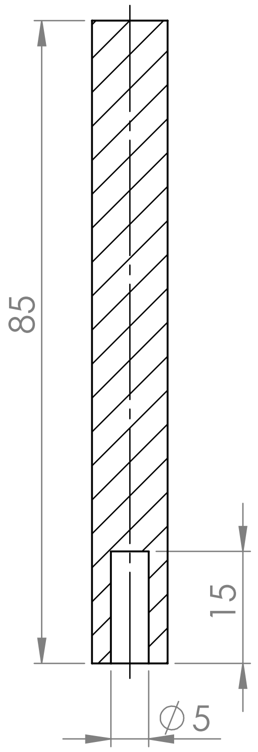
Press Cup

A4

WEIGHT:

SCALE:2:1

SHEET 1 OF 1



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	NAME	SIGNATURE	DATE
DRAWN	Denis Bobylev		
CHK'D			
APPV'D			
MFG			
Q.A			

TITLE:

MATERIAL:

DWG NO. shaft

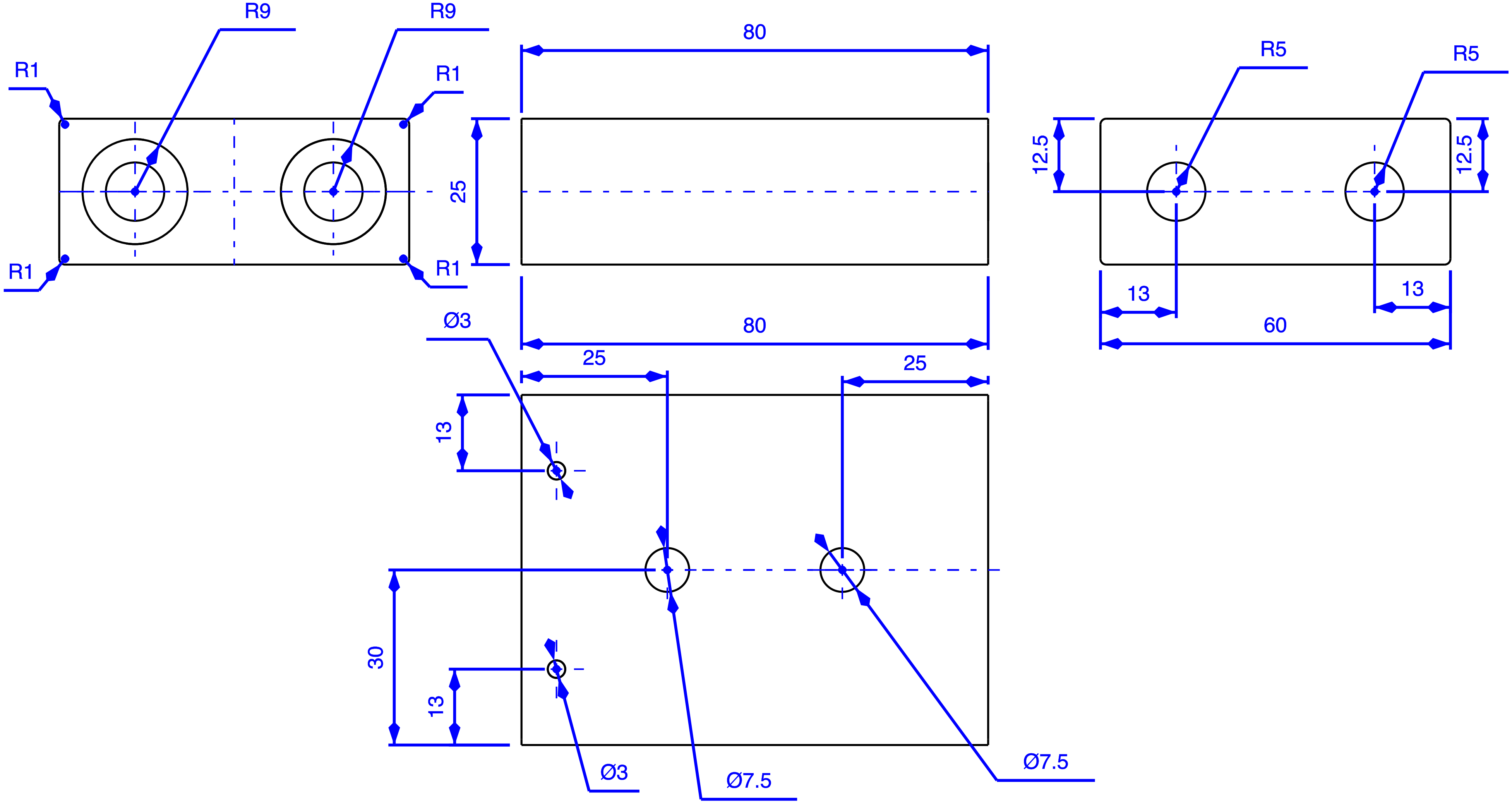
A4

WEIGHT:

SCALE:1:1

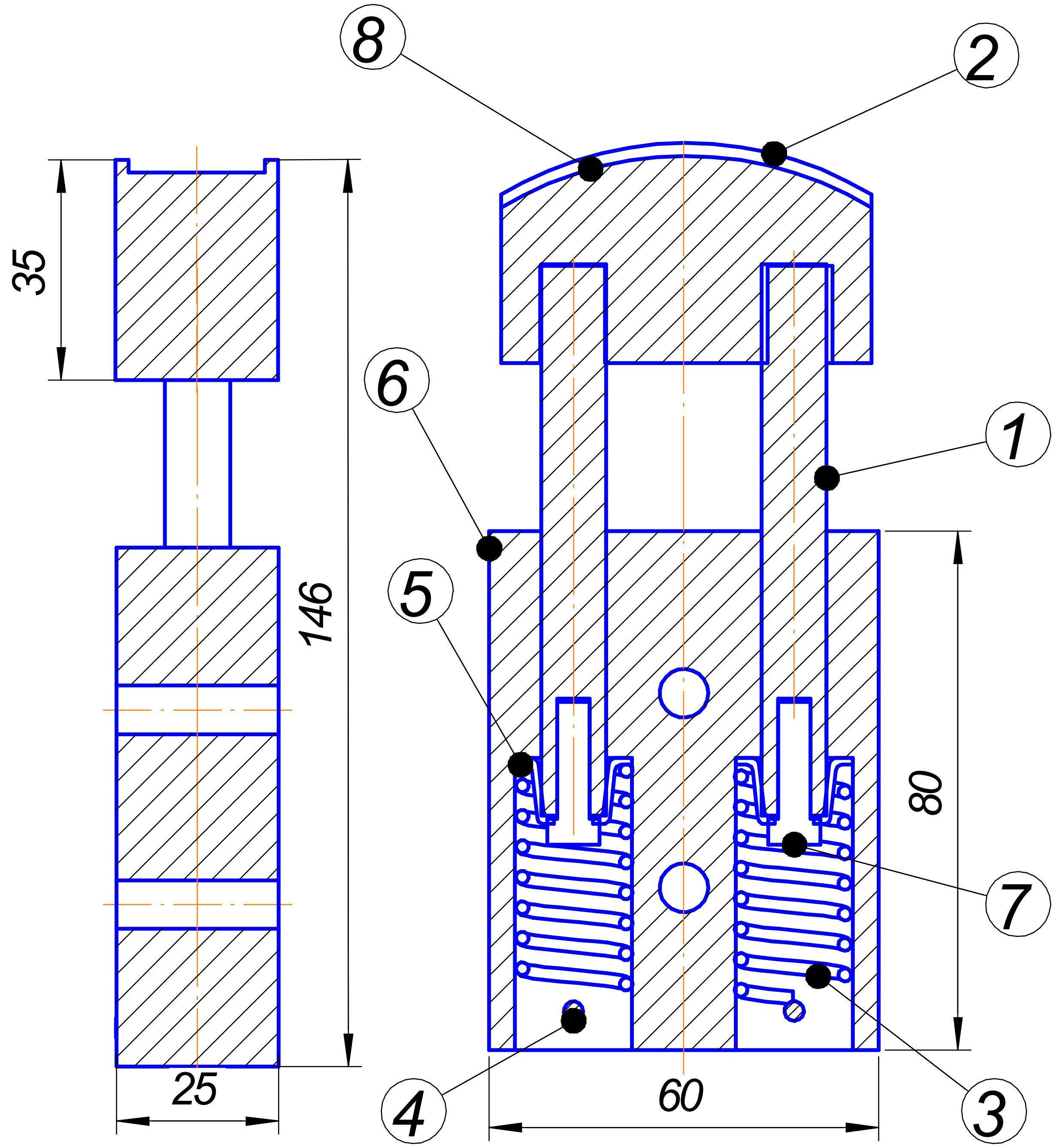
SHEET 1 OF 1

Appendix 2. Drawing in FreCAD



Created by: Denis Bobylev	Title: Body	Size:	Sheet:	Scale: 1:1
Supplementary information: FreeCAD DRAWING		Part number: 1		
		Drawing no.:		
		Date: 28/04/2017		Revision:

Appendix 3. Drawings in Kompas- 3D



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DRAWN	Denis Bobylev						TITLE: Assembly
CHK'D							
APPV'D							
MFG							
Q.A				MATERIAL:			DWG.NO / A4 / 1
				WEIGHT:			SCALE: 1:1 / SHEET

ITEM NO.	QTY.	PART NO.	NAME	WEIGHT	STOCK SIZE	MATERIAL	DESCRIPTION
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2	1		Plastic Slide				
3	2		Spring				
4	2		Pin				
5	2		Press Cap				
6	1		Body				
7	2		ISO 14580-#4 x 25				
			x 23.6 - 4.8-N				

UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS

SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:

FINISH:

DEBUR AND BREAK SHARP EDGES

DO NOT SCALE DRAWING

REVISION

	NAME	SIGNATURE	DATE
DRAWN			
CHK'D			
APPV'D			
MFG			
Q.A			

TITLE: Assembly

MATERIAL:

DWG.NO

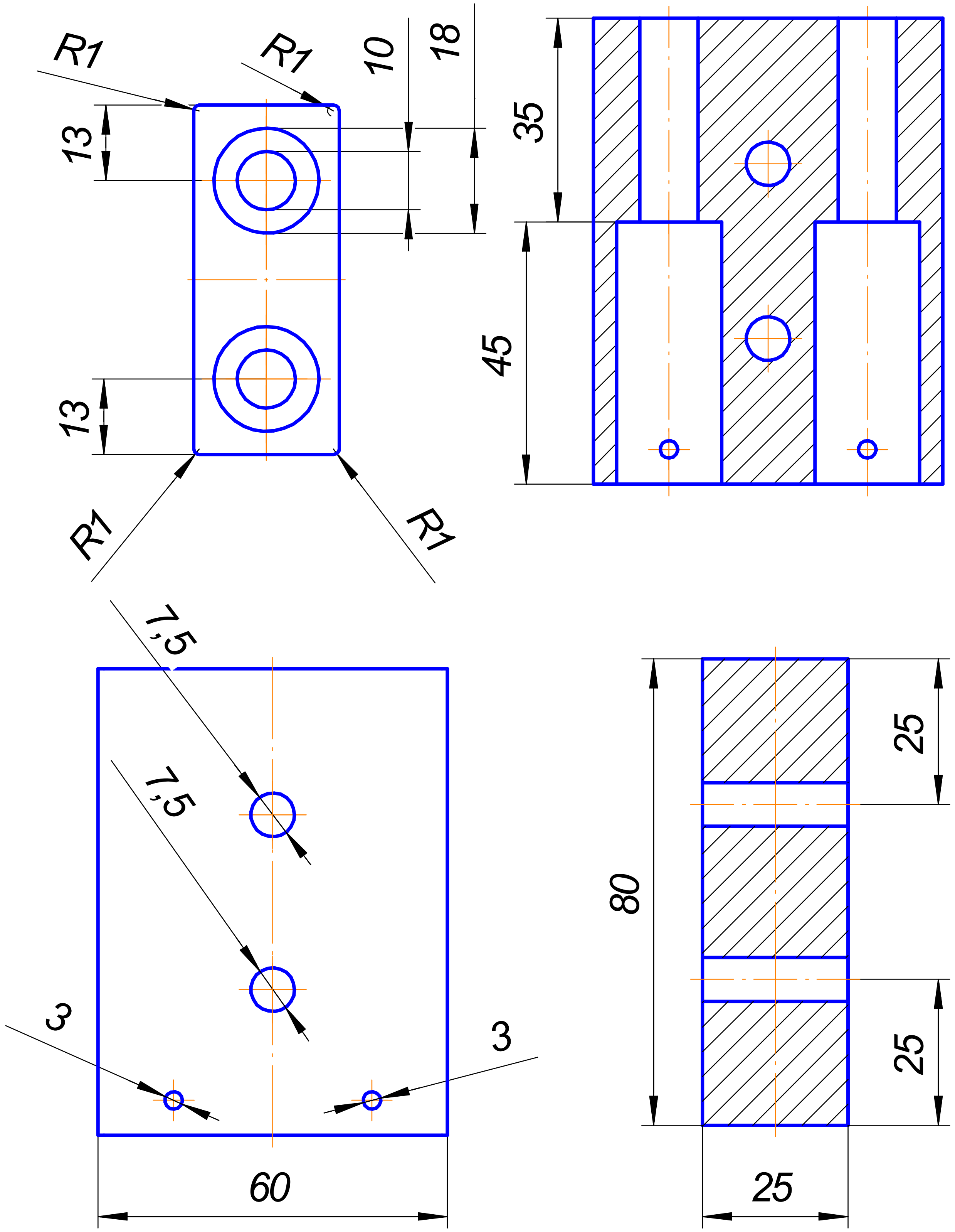
A4

1

WEIGHT:

SCALE:

SHEET



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TOLERANCES:
LINEAR:
ANGULAR:

FINISH:

DEBUR
AND BREAK
SHARP EDGES

DO NOT SCALE DRAWING

REVISION

	NAME	SIGNATURE	DATE
DRAWN	Denis Bobylev		
CHK'D			
APPV'D			
MFG			
Q.A			

TITLE: Body

MATERIAL: Steel St35 DIN 1629/3

DWG.NO

A4

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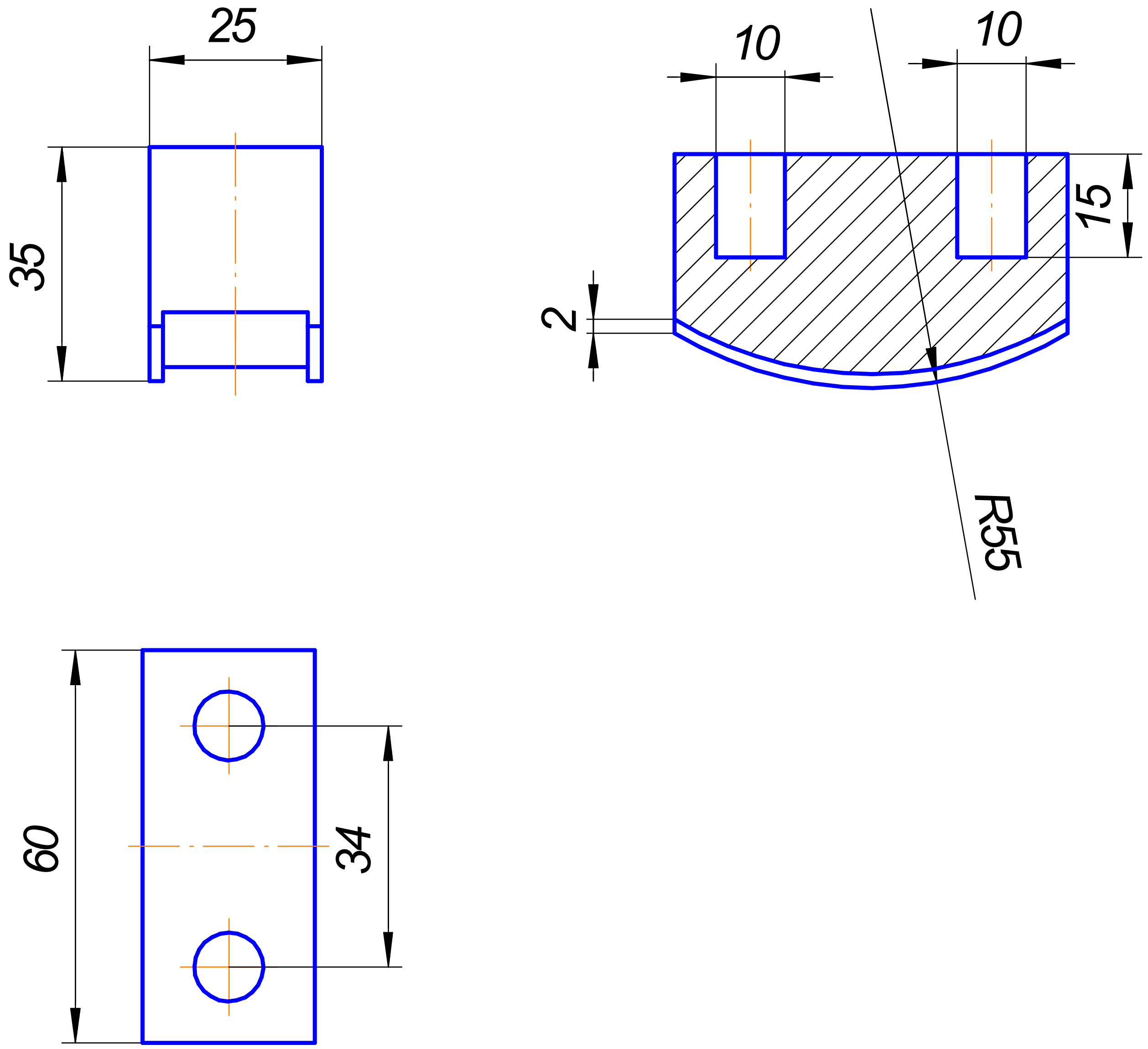
WEIGHT: 0,7

SCALE: 1:1

SHEET



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LINEAR:
ANGULAR:

FINISH:

DEBUR AND BREAK SHARP EDGES

DO NOT SCALE DRAWING

REVISION

	NAME	SIGNATURE	DATE
DRAWN	Denis Bobylev		
CHK'D			
APPV'D			
MFG			
Q.A			

TITLE: Plastic Slide

MATERIAL: Steel St35 DIN 1629/3

DWG.NO

A4

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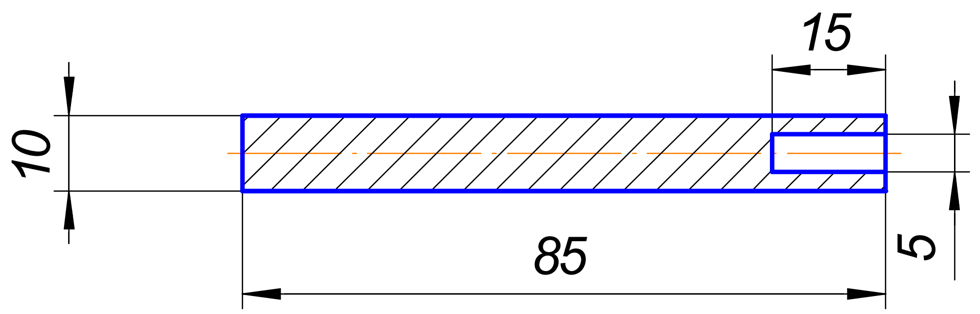
WEIGHT: 0,31

SCALE: 1:1

SHEET



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TOLERANCES:
LINEAR:
ANGULAR:

FINISH:

DEBUR
AND BREAK
SHARP EDGES

DO NOT SCALE DRAWING

REVISION

	NAME	SIGNATURE	DATE
DRAWN	Denis Bobylev		
CHK'D			
APPV'D			
MFG			
Q.A			

TITLE: Shaft

MATERIAL: Steel St35 DIN 1629/3

DWG.NO

A4

1

WEIGHT: 0

SCALE: 1:1

SHEET