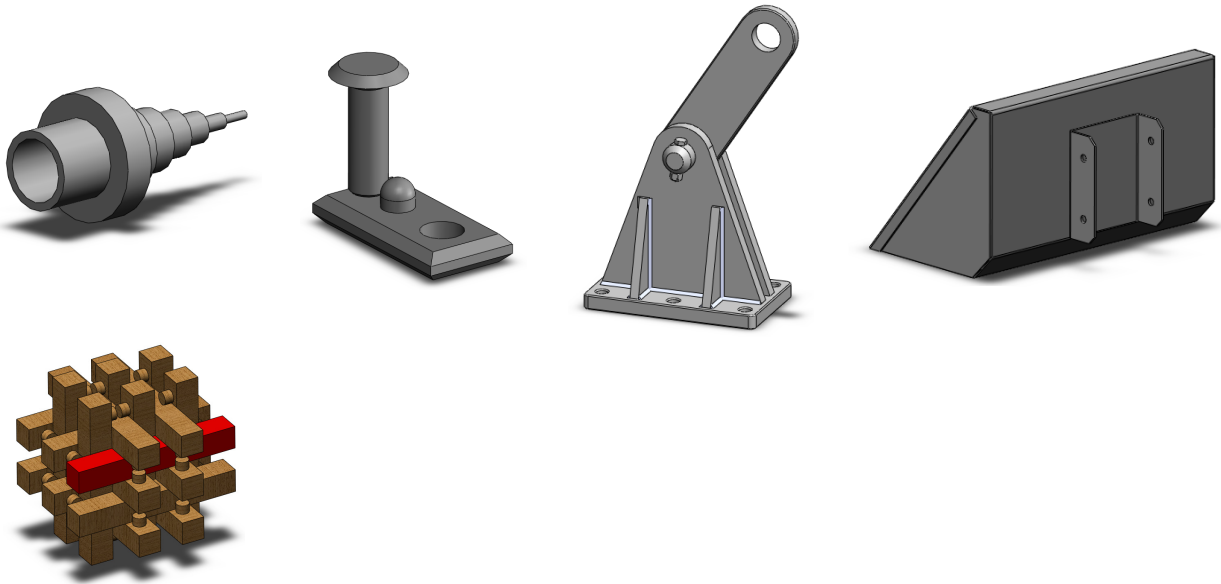


# SolidWorks® tutorials

## EXERCISES



Junior and Senior Secondary Technical Education



Voor gebruik met SolidWorks® Educational Release 2008-2009

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U.S. Patents 5,815,154; 6,219,049; 6,219,055

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Initiative: Kees Kloosterboer (SolidWorks Benelux)

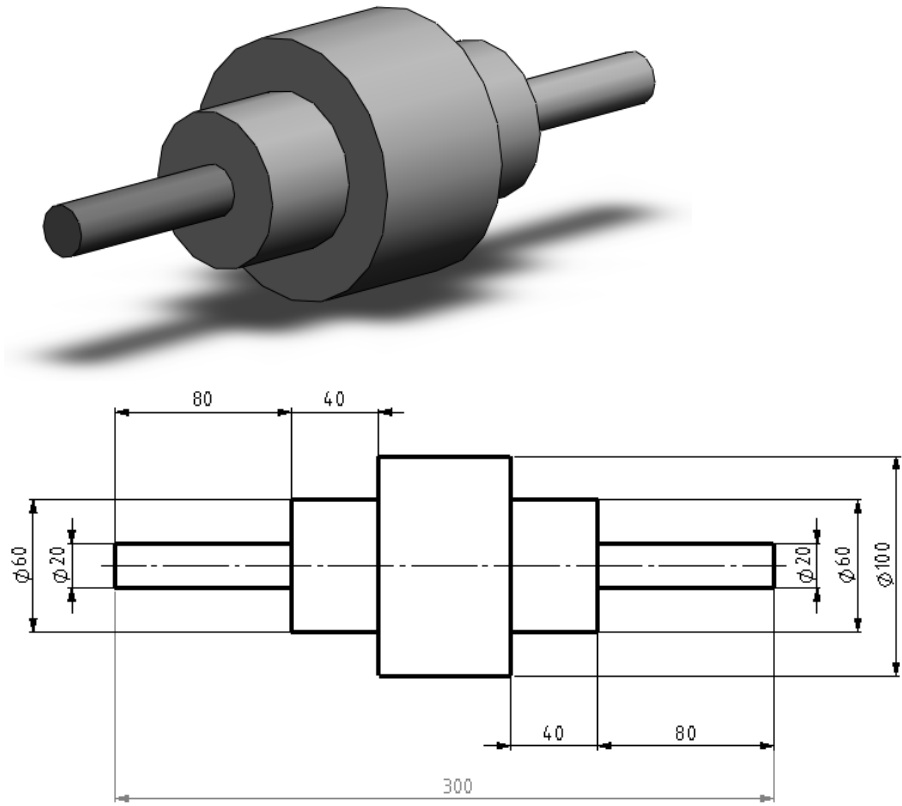
Educational coordination: Jack van den Broek (specialist class Dr. Knippenberg)

Realization: Arnoud Breedveld (PAZ Computerworks)

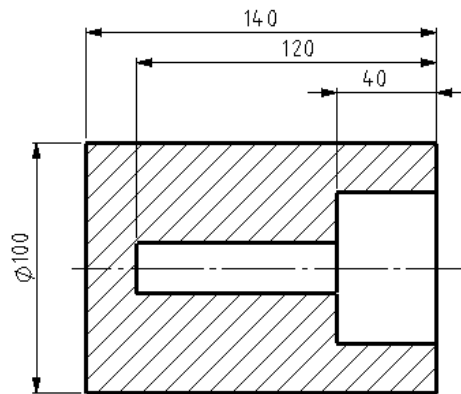
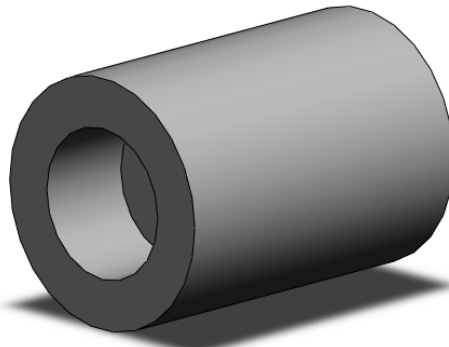
<b>Exercises</b>	
<b>Approach</b>	<p>Congratulations! You have already worked through the first few SolidWorks tutorials, so you already have some understanding of SolidWorks. In order to really get to know the program, it is important that you practice a lot! That is why we have put this bundle together. You now have a large number of exercises to help you become familiar with SolidWorks.</p> <p>The best idea is to do the relevant exercises from this bundle every time you have completed a SolidWorks tutorial. You can then put what you have learnt into practice straight away.</p> <p>The exercises you are going to do depend on each other. This means that: you may make an axle in the exercises for tutorial one, another part in tutorial 3 and then put them together in an assembly in tutorial 7. <b>It is therefore very important that you save everything that you make</b> (parts, assemblies and drawings)! Create a new folder for this purpose. You could call this folder “SolidWorks exercises”, for example. Always give the file the same name as the exercise (for example: Exercise1-1sldprt).</p> <p>Most of the exercises do not have an explanation. You should be able to model the part using the drawing.</p> <p><b>Have fun drawing and good luck!</b></p>

## Tutorial 1 Exercises

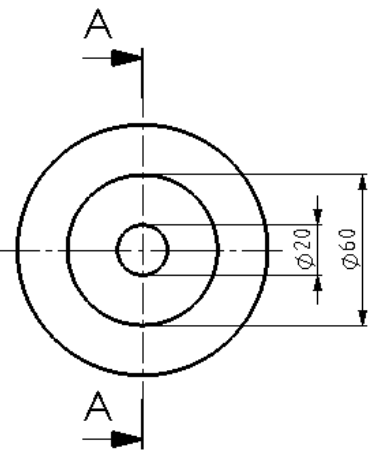
### Exercise 1-1



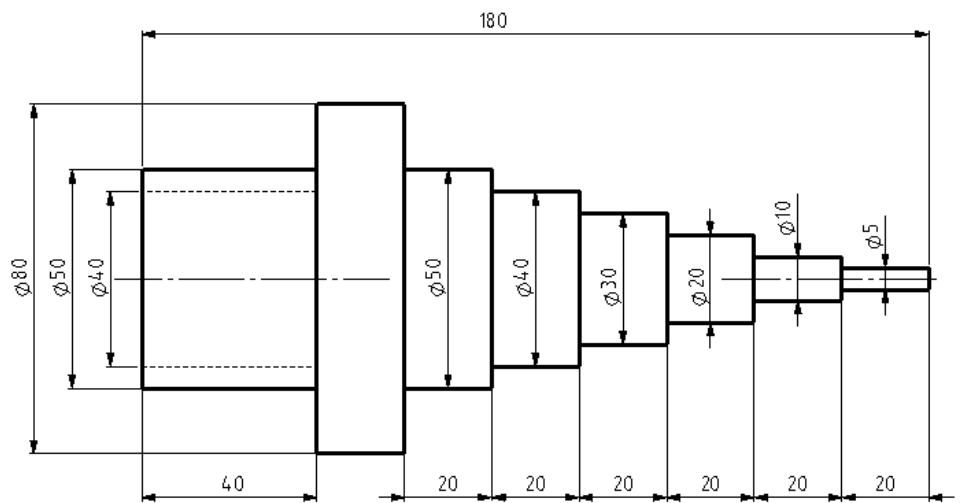
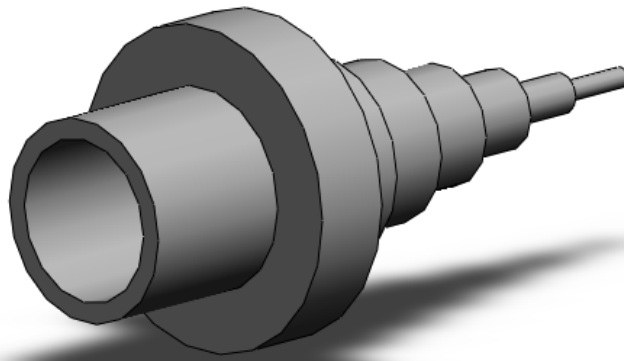
Exercise 1-2



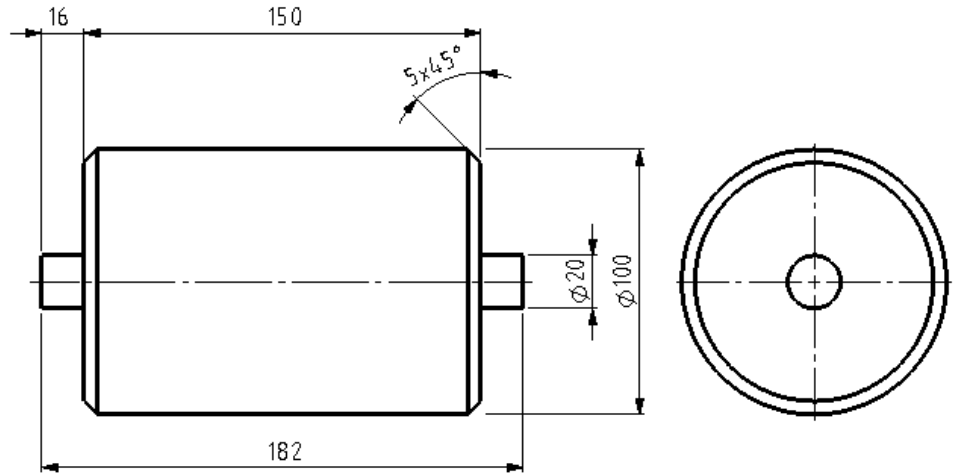
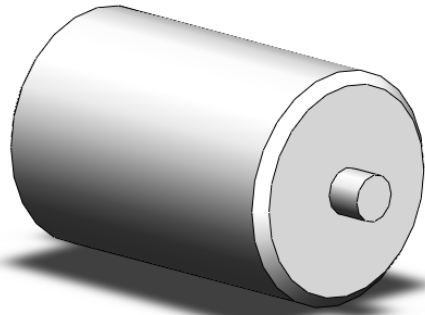
SECTION A-A



**Exercise 1-3**

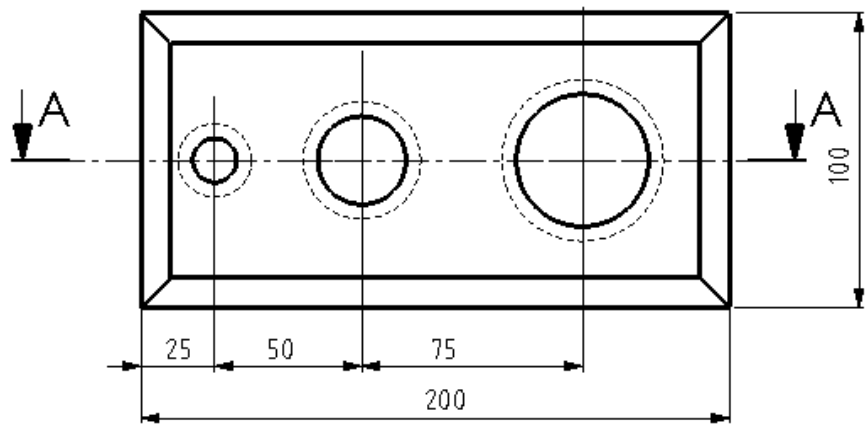
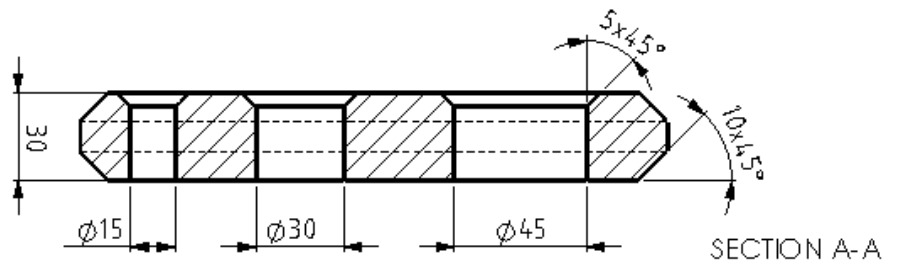
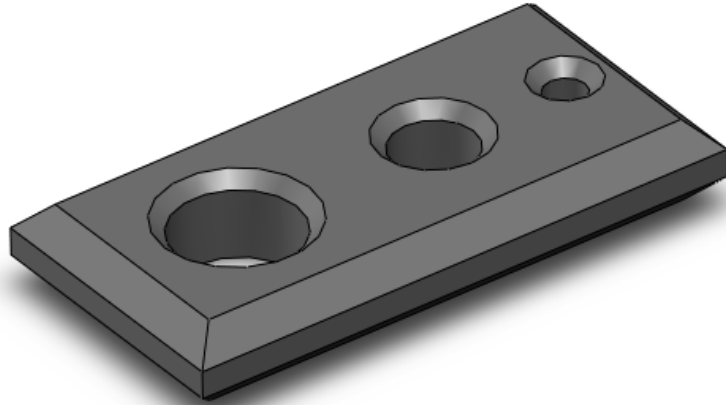


**Exercise 1-4**



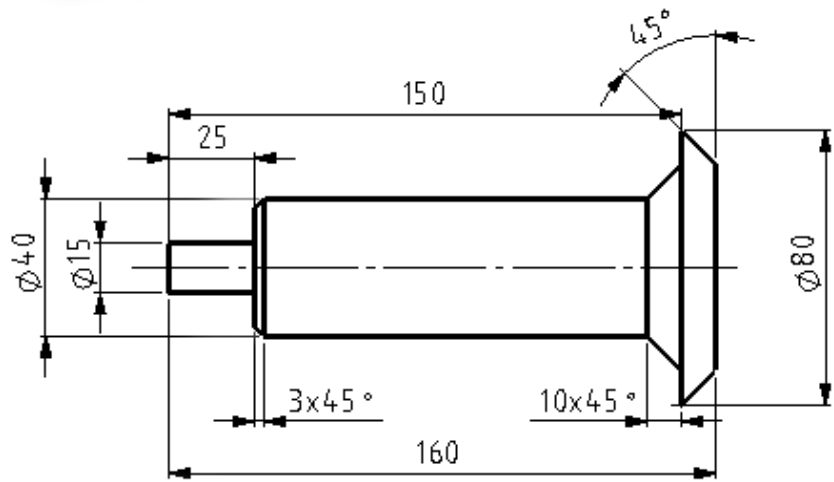
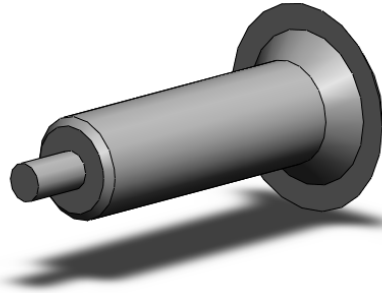
## Tutorial 2 Exercises

### Exercise 2-1

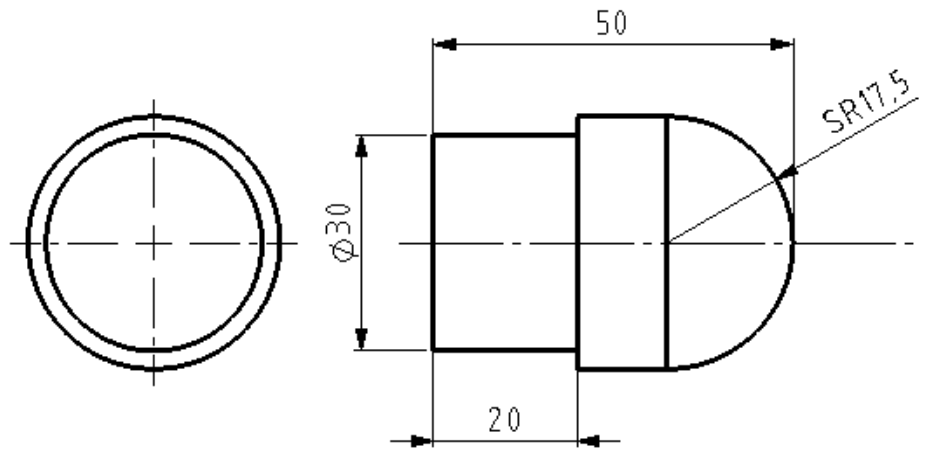
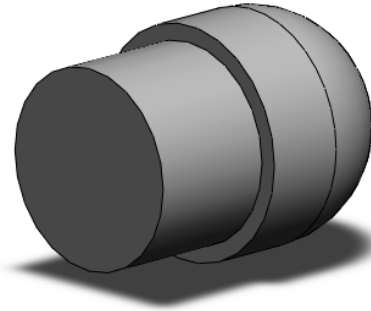




Exercise 2-2

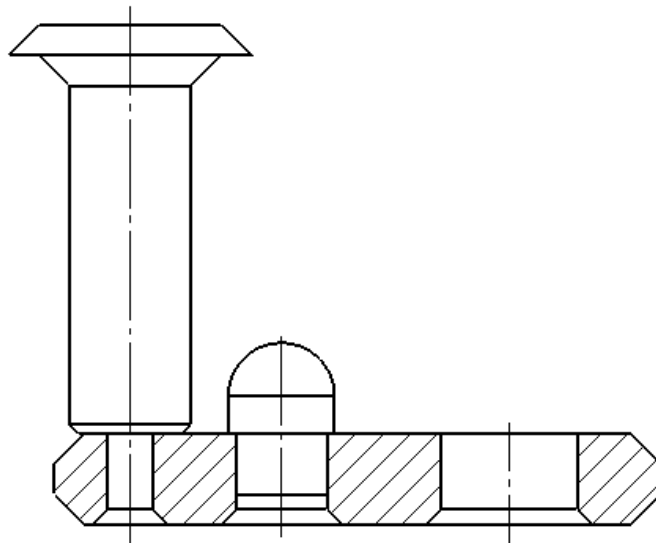
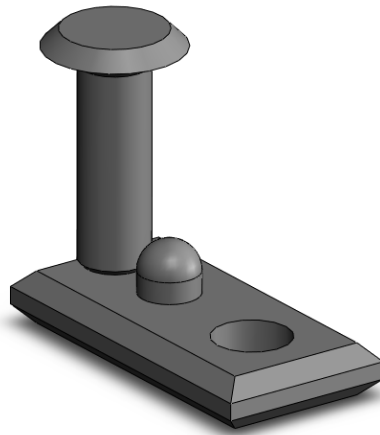


Exercise 2-3

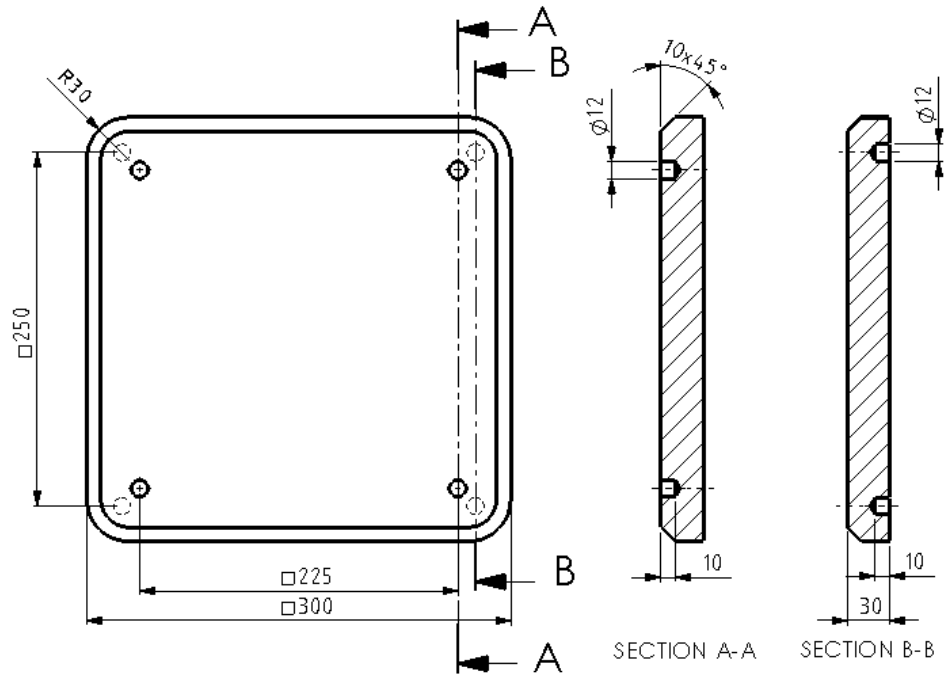
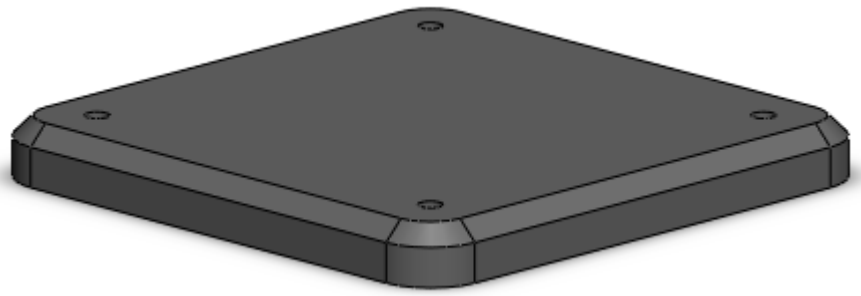


**Exercise 2-4**

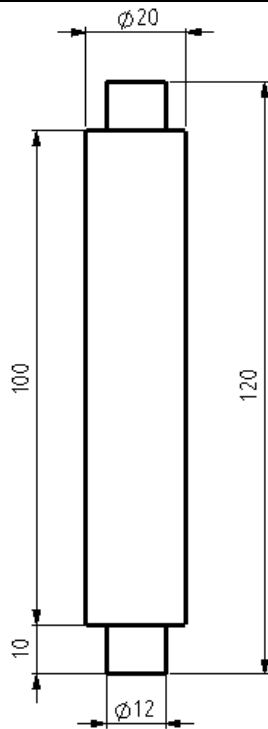
Use the parts from exercises 2-1 through 2-3 to make the assembly shown on the right.



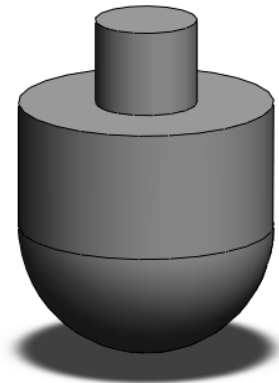
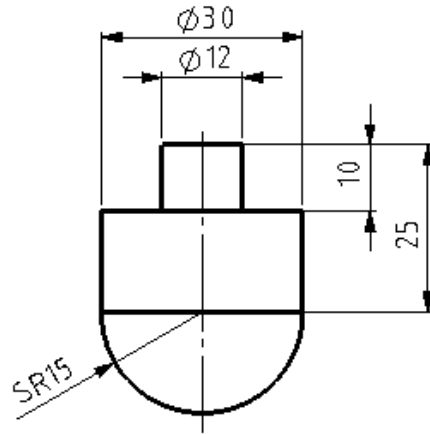
**Exercise 2-5**



**Exercise 2-6**



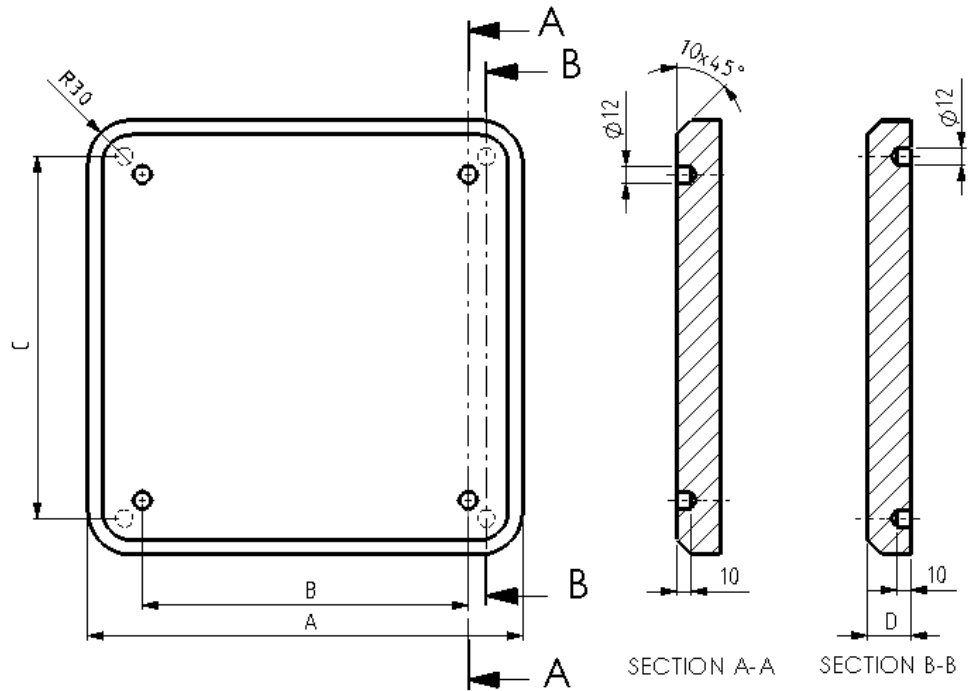
**Exercise 2-7**



## Tutorial 3 Exercises

### Exercise 3-1

Based on the model from exercise 2-5, make configurations using the dimensions in the table on the right. Configuration 1 has the same dimensions as the model in exercise 2-5, so 5 new configurations need to be made.

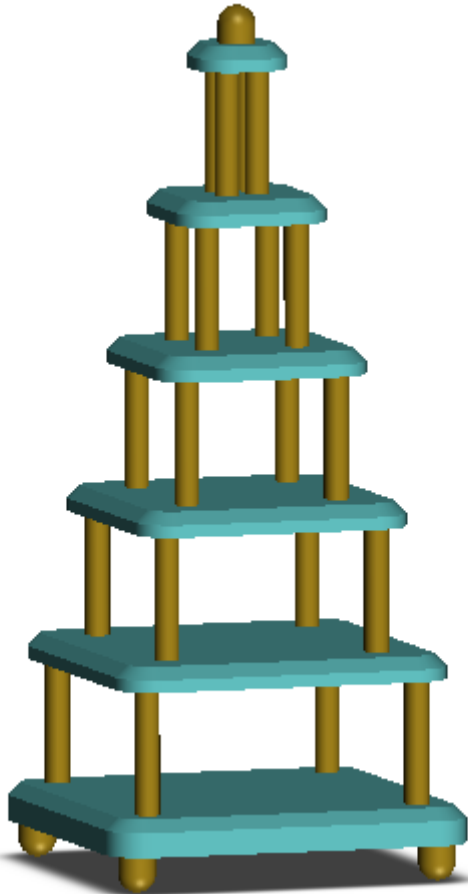


	A	B	C	D
Configuration 1	300	225	250	30
Configuration 2	275	175	225	20
Configuration 3	225	125	175	20
Configuration 4	175	75	125	20
Configuration 5	125	25	75	20
Configuration 6	75	0*	25	20

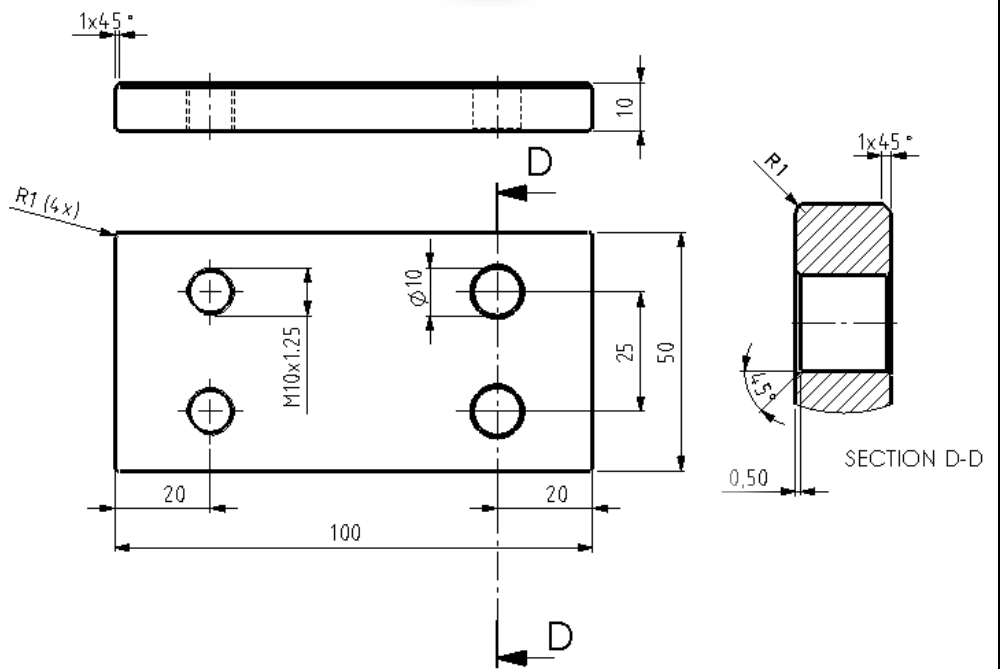
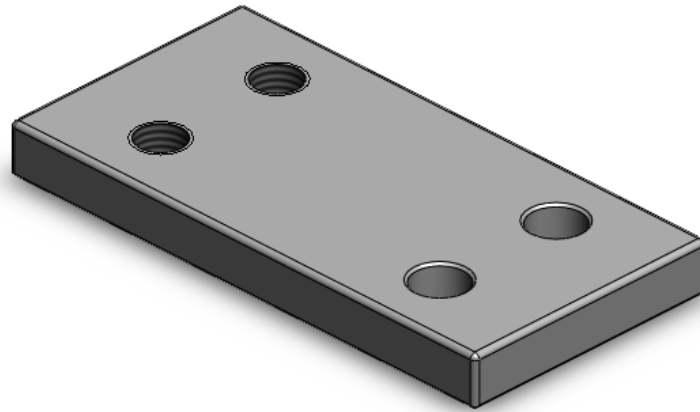
\* Configuration 6 has just one single hole in its upper surface, in the middle of the surface.

**Exercise 3-2**

Put the parts from exercises 2-5 through 3-1 together to produce an assembly as shown on the right. Use different colors for the different parts.



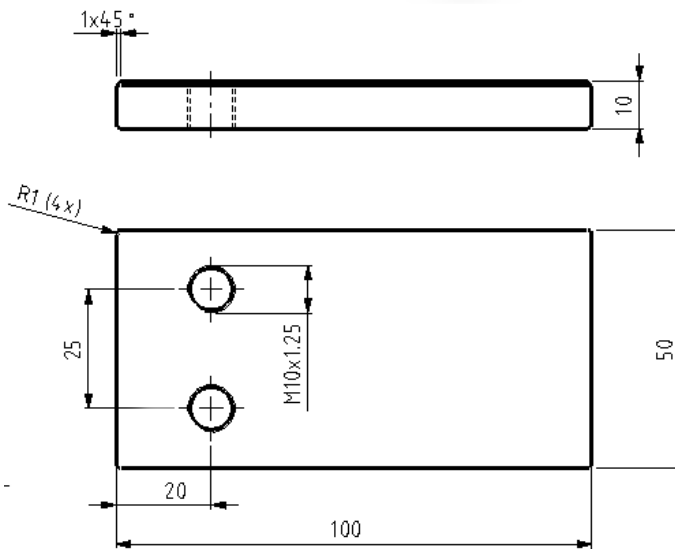
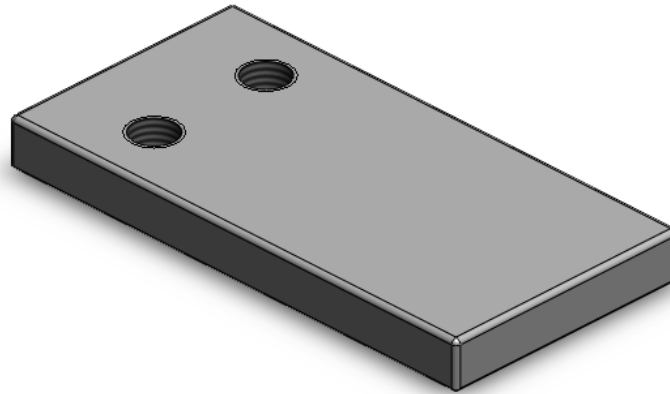
Exercise 3-3





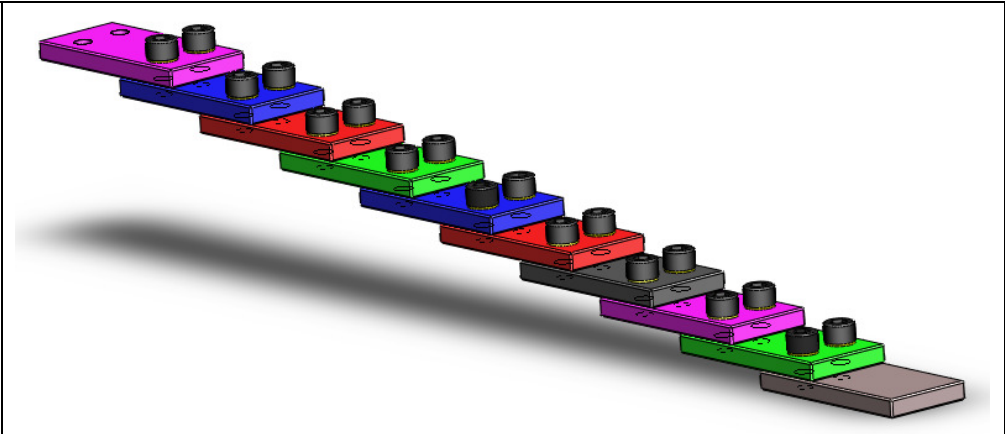
**Exercise 3-4**

Make a new configuration out of the model from exercise 3-3 as shown on the right. You need to suppress the two holes to make this configuration.

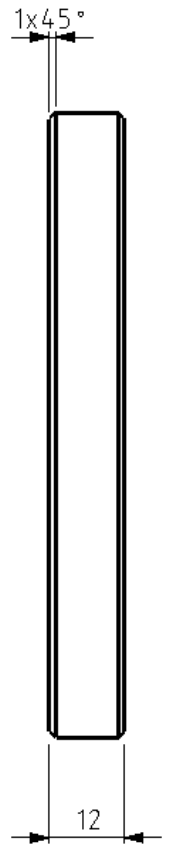
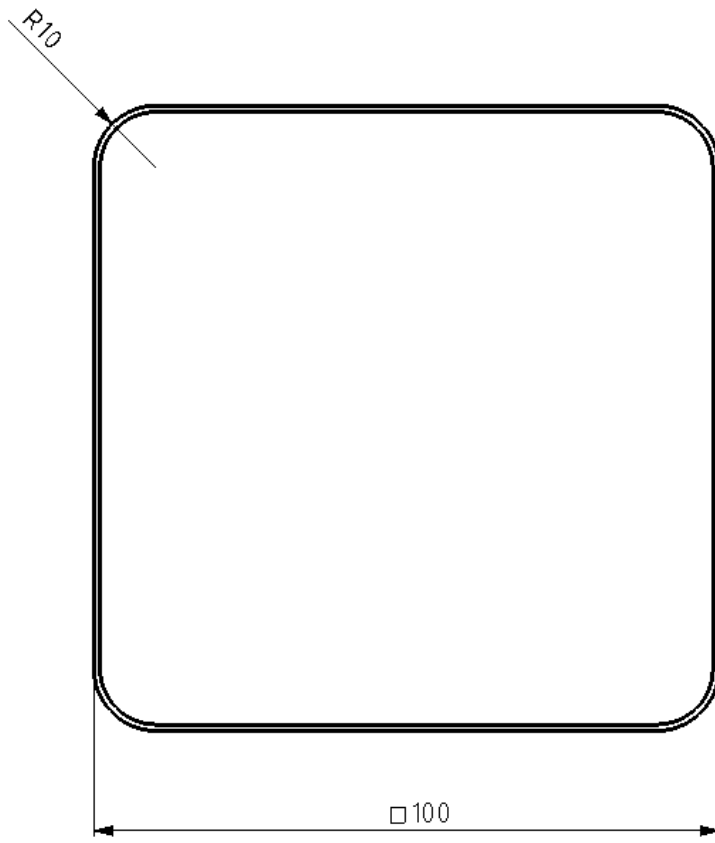


**Exercise 3-5**

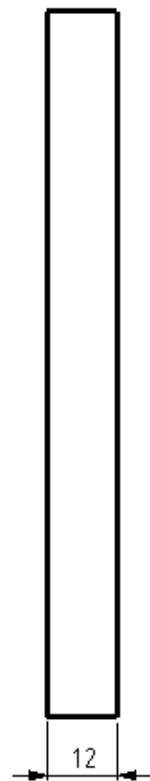
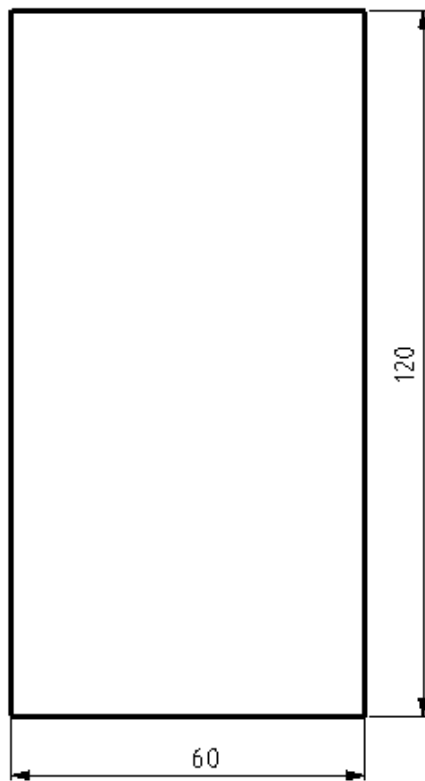
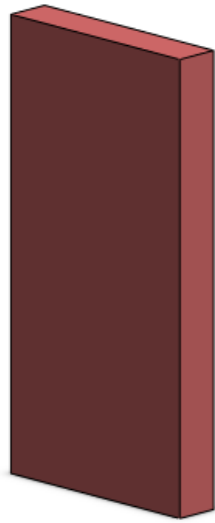
Make an assembly as shown on the right. Use the parts from exercises 3-3 and 3-4. To secure it, use a Hex Socket Head, M12x1.25x20 from the Toolbox. Place a plain washer under each screw.



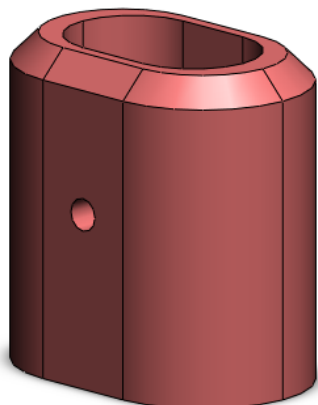
Exercise 3-6

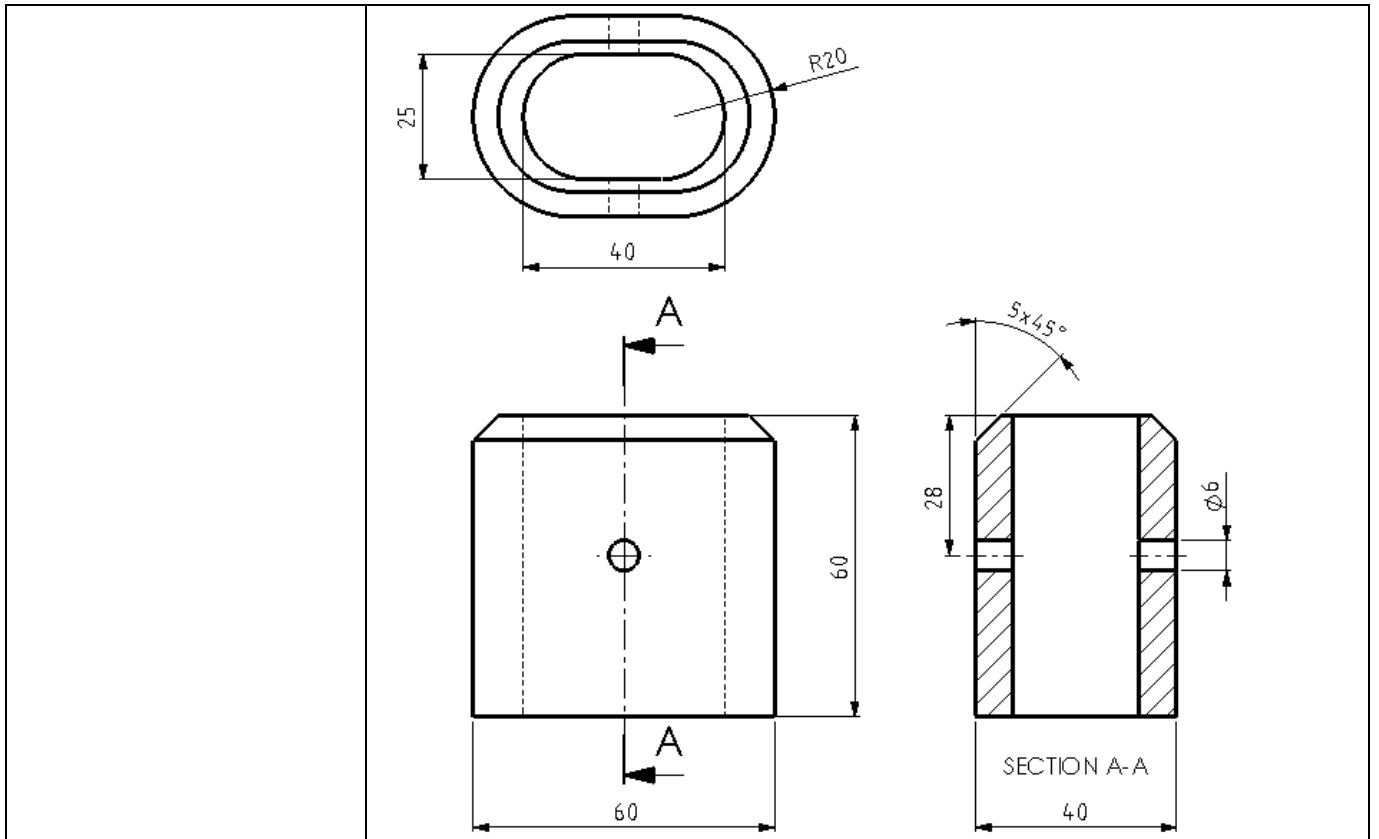


**Exercise 3-7**



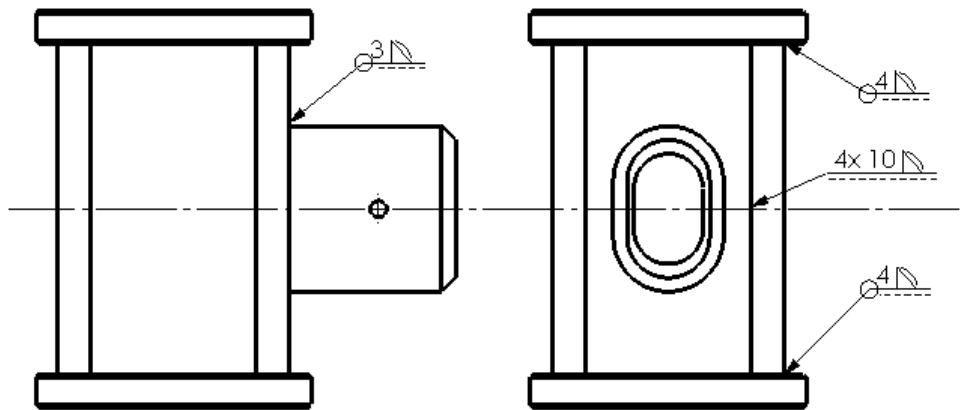
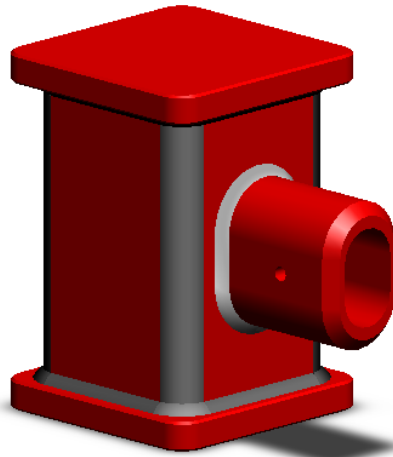
**Exercise 3-8**





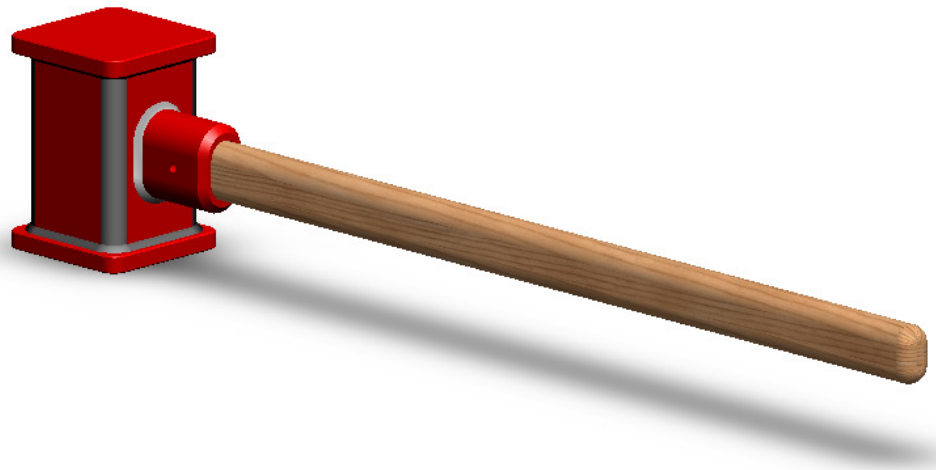
**Exercise 3-9**

Use the parts from exercises 3-6 through 3-8 to make the assembly. Add fillet welds as shown in the drawing on the right.

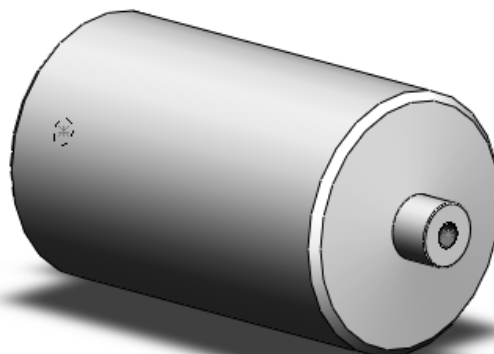


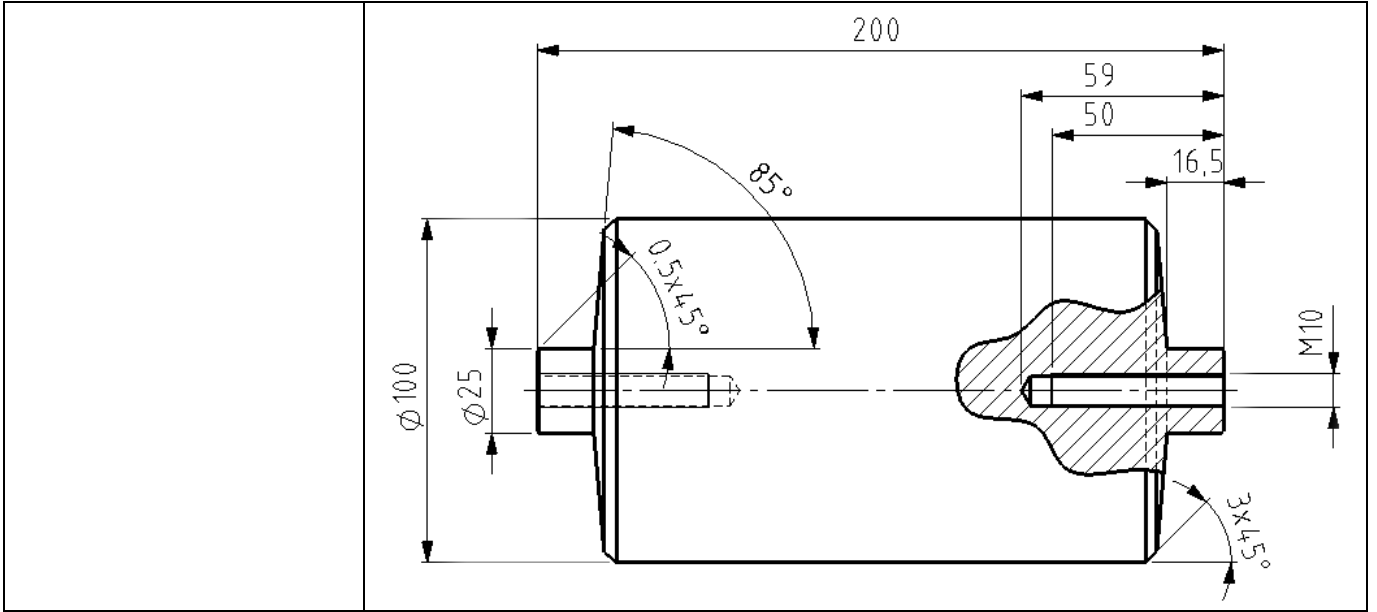
**Exercise 3-10**

Now make a 700mm long handle and add it to the assembly.

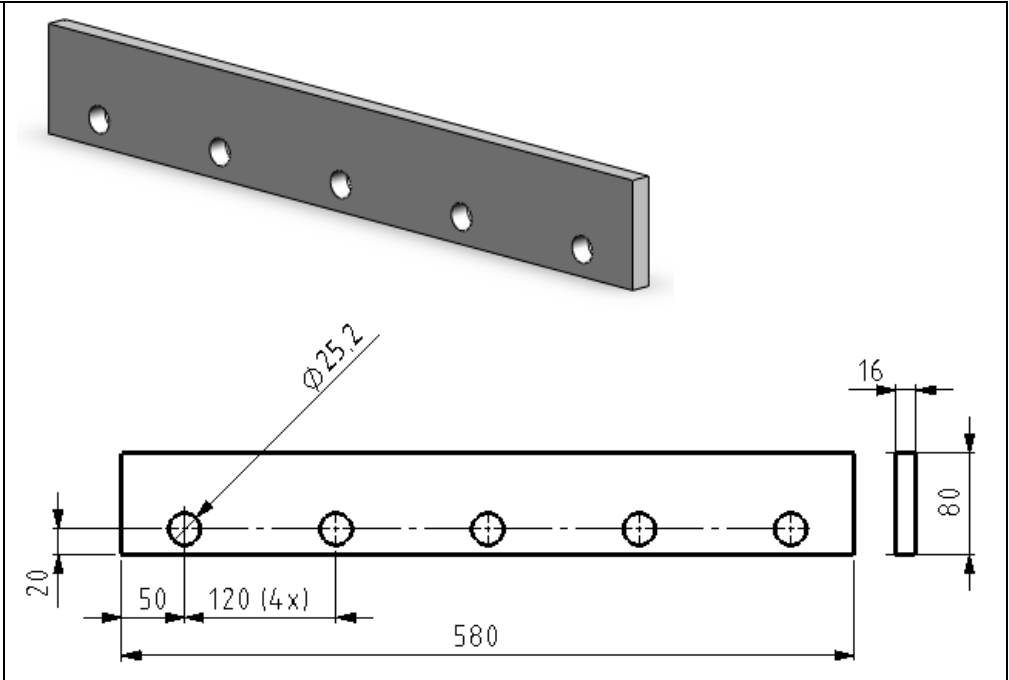


**Exercise 3-11**



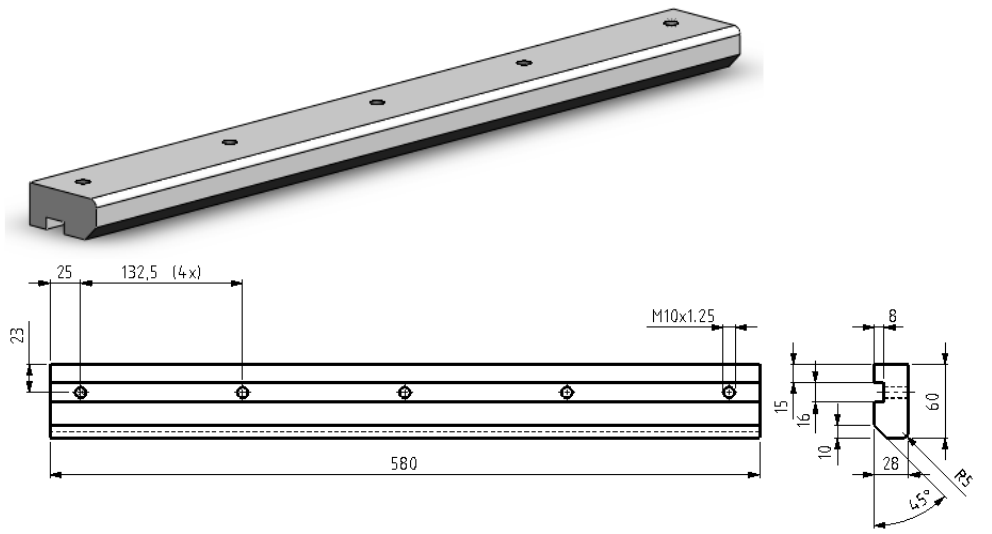


**Exercise 3-12**

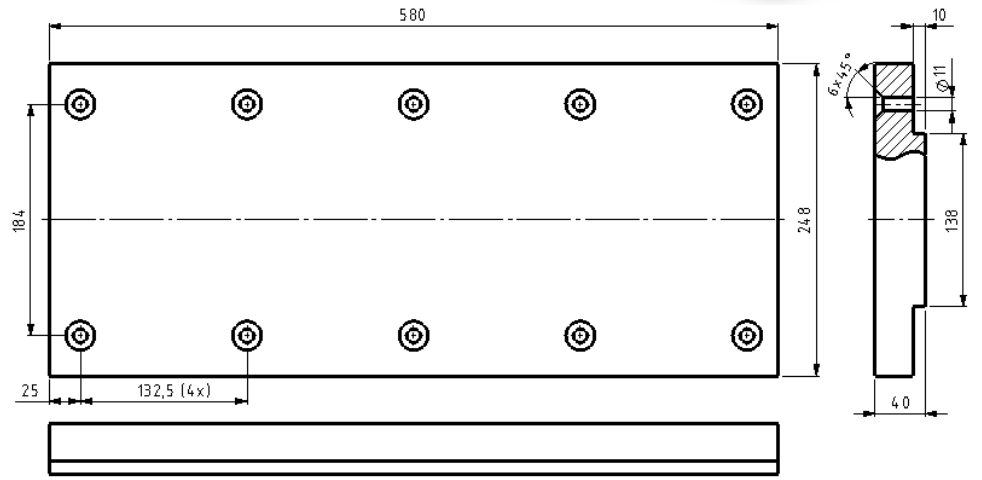
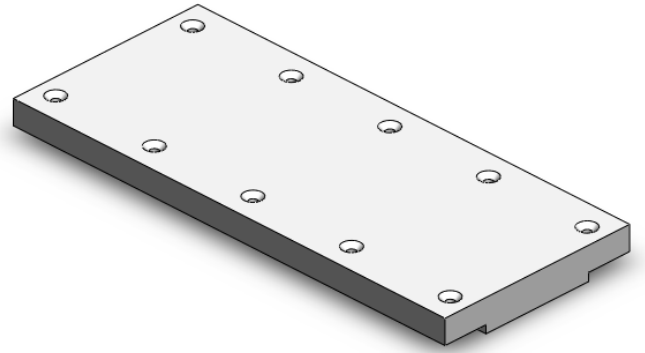




**Exercise 3-13**



**Exercise 3-14**



**Exercise 3-15**

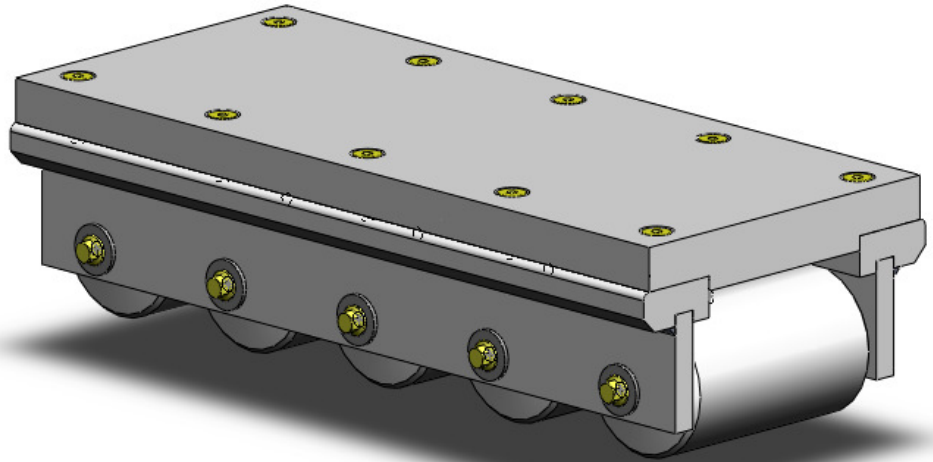
Make the assembly using parts 3-11 through 3-14.

Use 10x Socket Countersunk Head Screw (ISO 10642)

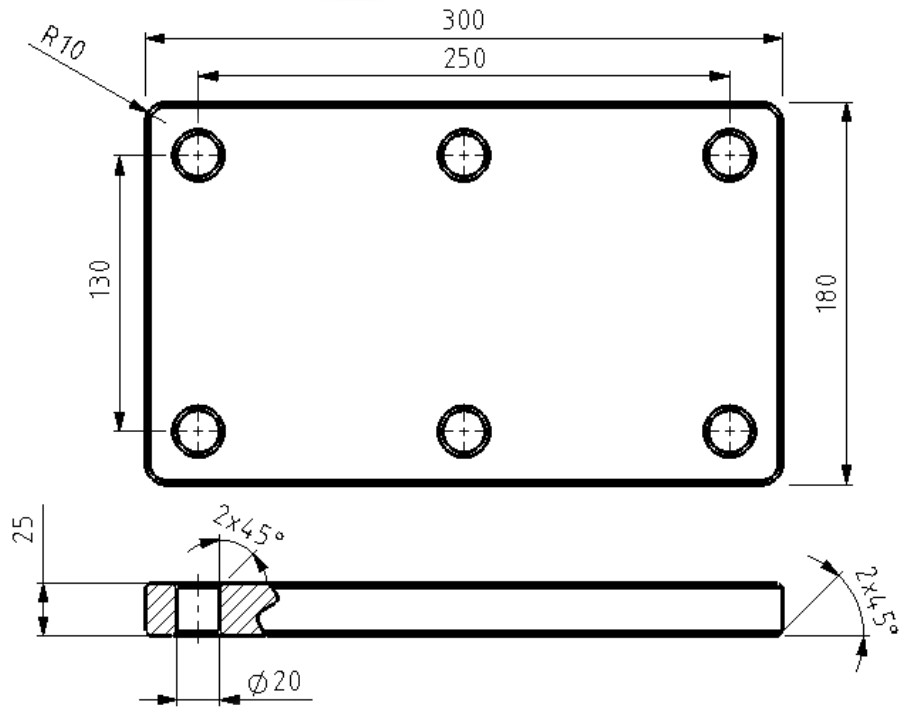
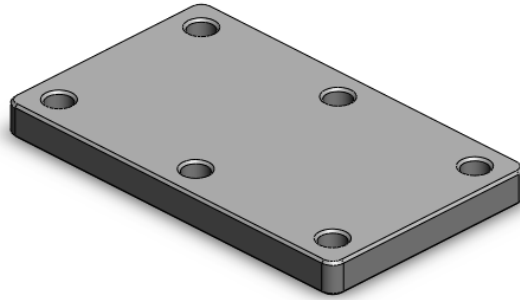
M10x45 to attach strip 3-13 to top plate 3-14.

Weld strip 3-12 to strip 3-13 using a fillet weld over the full length.

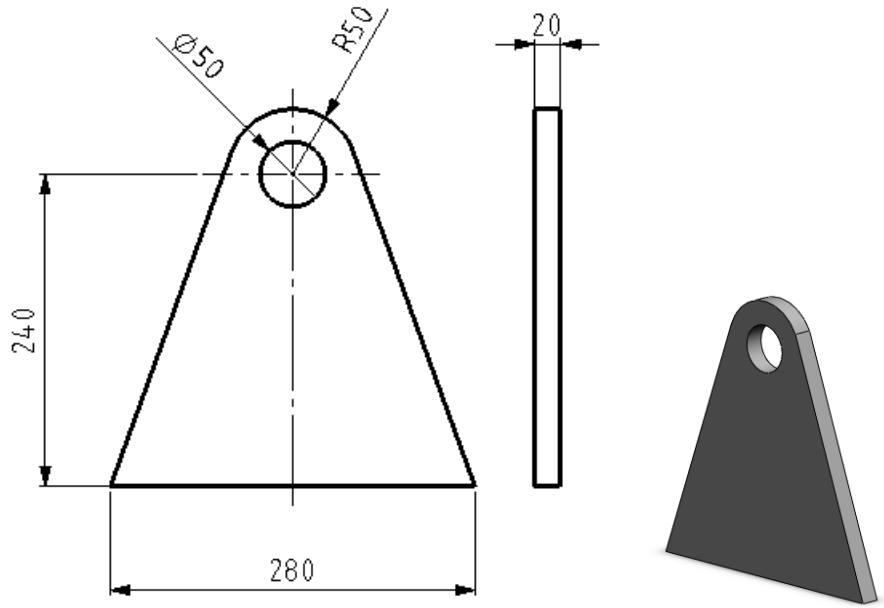
Use hex head screws M10x30 and an extra large plain washer to secure the rollers (3-11).



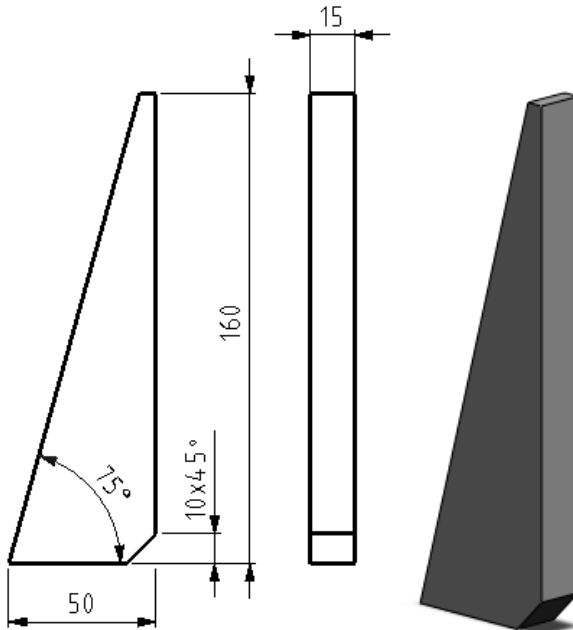
**Exercise 3-16**



**Exercise 3-17**

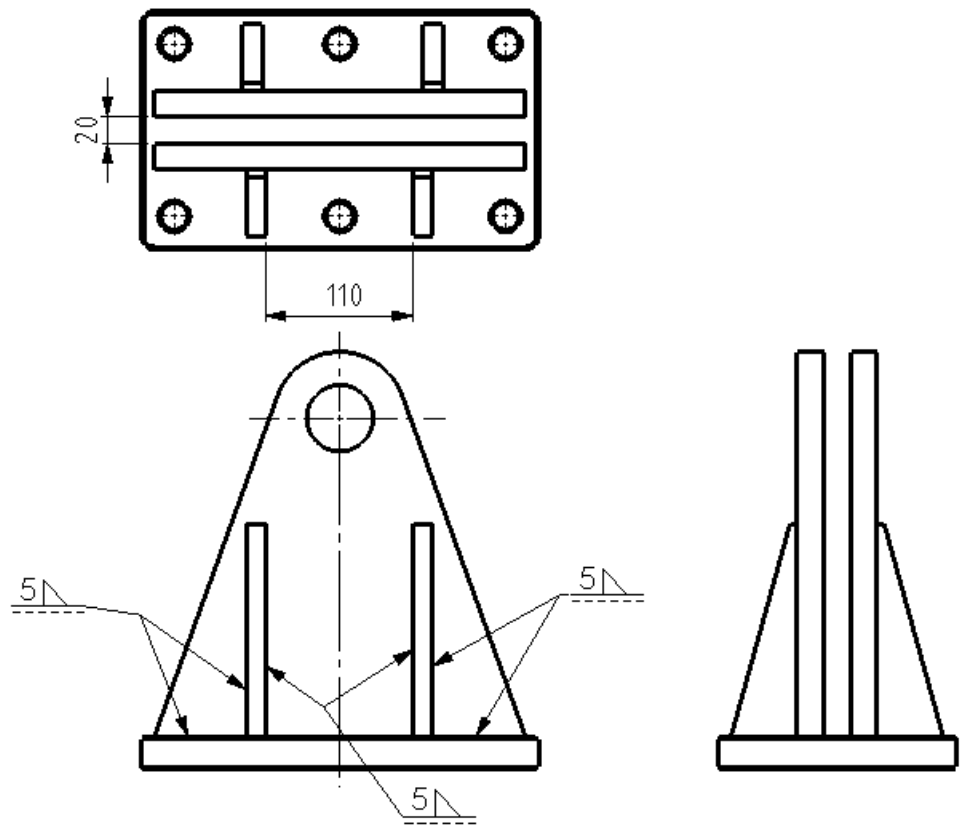
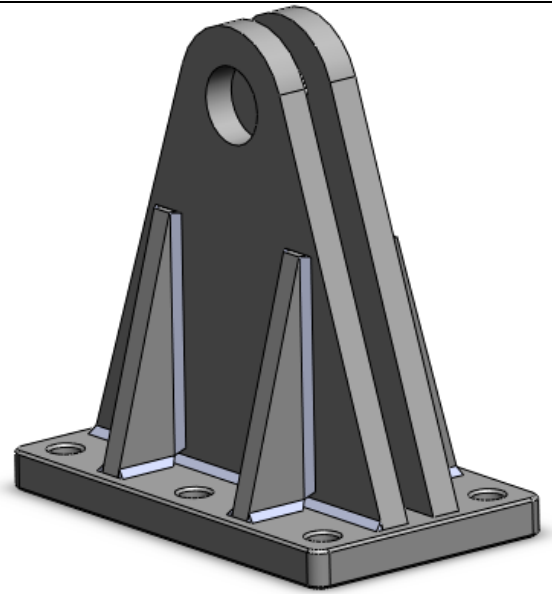


**Exercise 3-18**



**Exercise 3-19**

Make a welded assembly out of parts 3-16 through 3-18 as shown in the drawing on the right. Apply the welds as shown in the assembly drawing.



**Exercise 3-20**

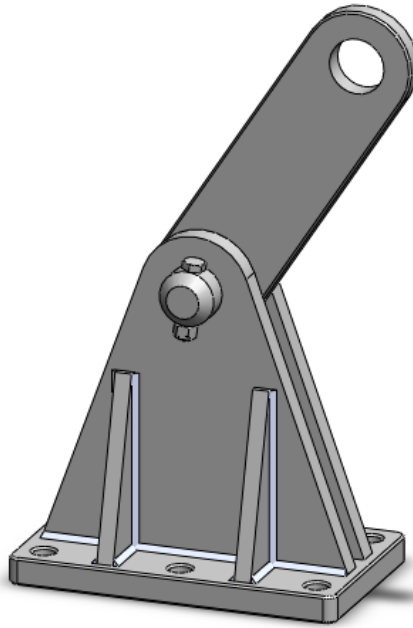
Technical drawing of a vertical link. The front view shows a total width of 100 and a height of 255. Two circular holes are positioned vertically, each with a diameter of  $\varnothing 50$ . The top corners are chamfered with a  $2 \times 45^\circ$  angle. A side view shows a thickness of 20. A 3D perspective view of the link is shown to the right.

**Exercise 3-21**

Technical drawing of a stepped shaft. The front view shows a total length of 120 and a central diameter of  $\varnothing 12$ . The length of the central section is 72. The right end has a diameter of  $\varnothing 80$  and a chamfered edge with a  $45^\circ$  angle. The left end has a chamfered edge with a  $10 \times 45^\circ$  angle. A cross-sectional view shows concentric circles with diameters of  $\varnothing 80$ ,  $\varnothing 50$ , and  $\varnothing 30$ . A 3D perspective view of the shaft is shown at the bottom right.

**Exercise 3-22**

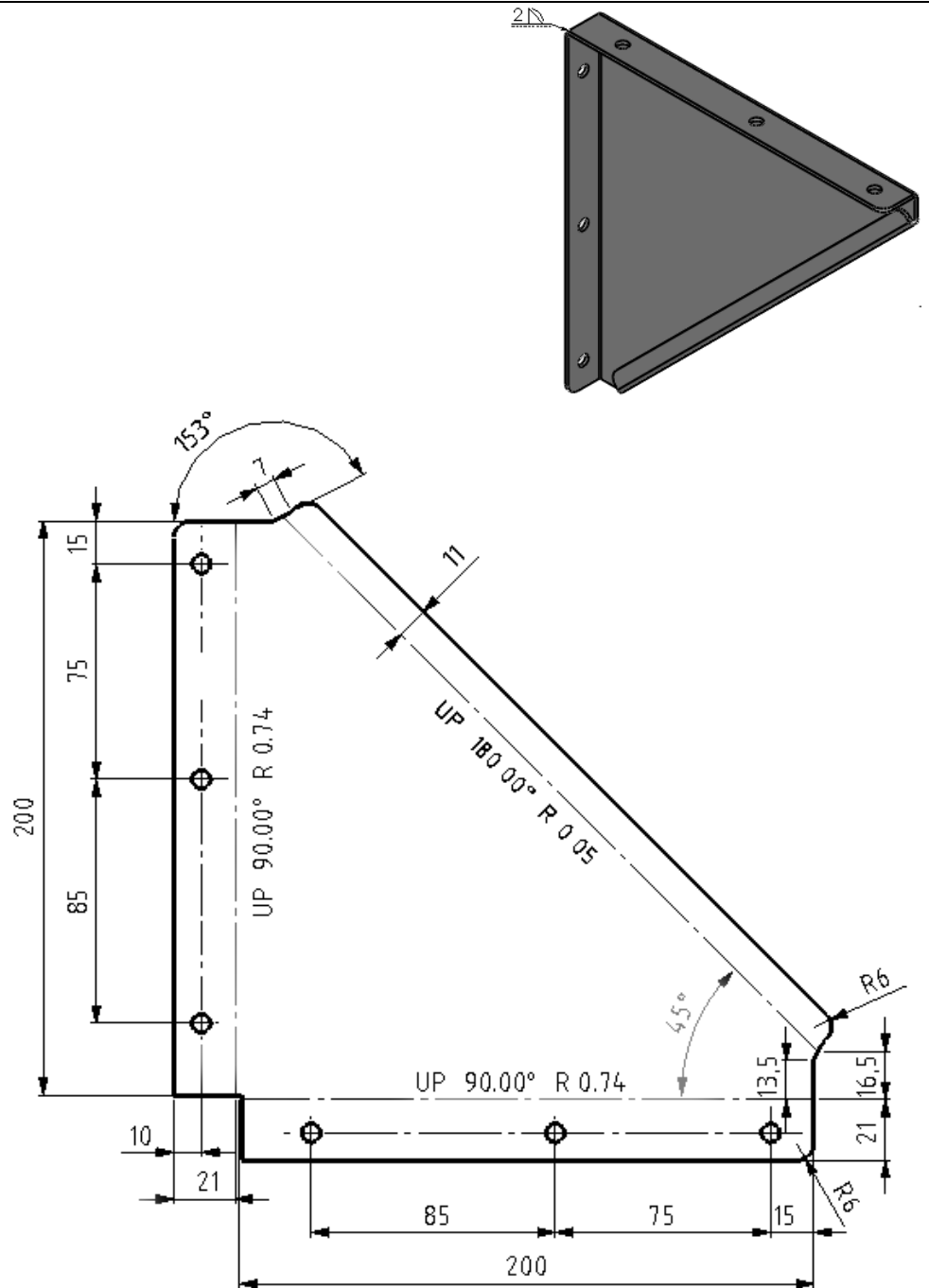
Make an assembly out of parts 3-19 through 3-21. Use a hex head screw (ISO 4014) M12x65 and a hex nut (ISO 4034) to secure the pin.



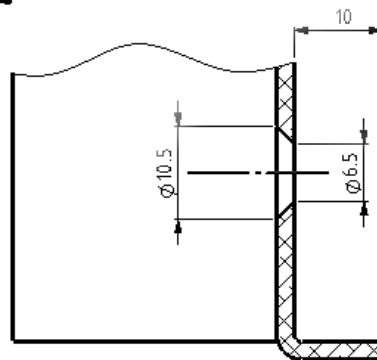
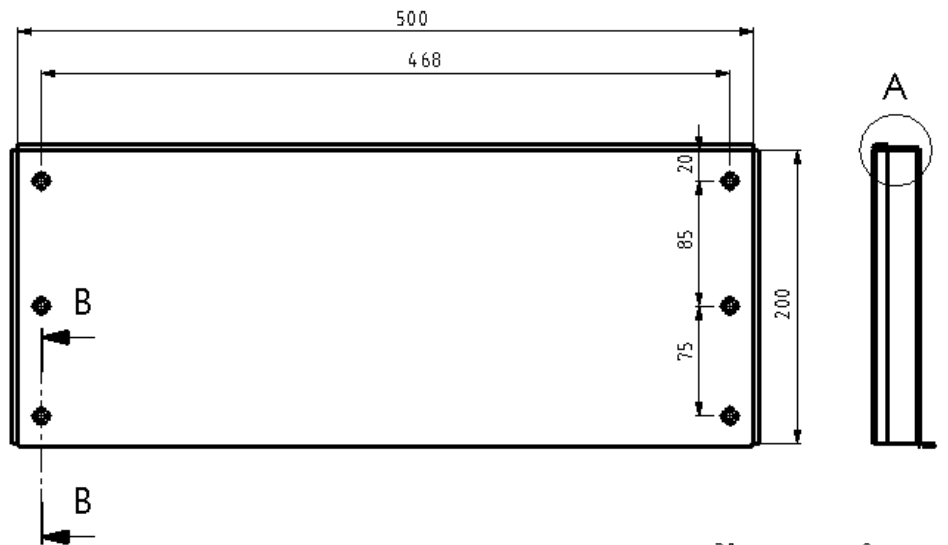
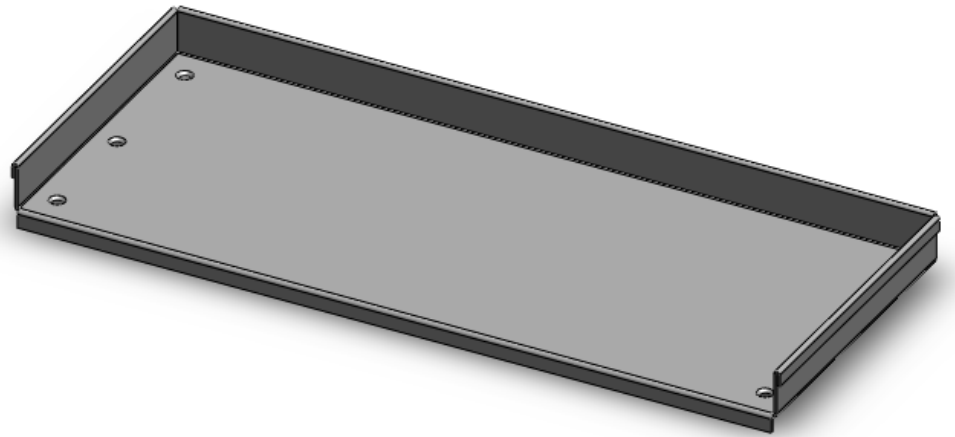


## Tutorial 4 Exercises

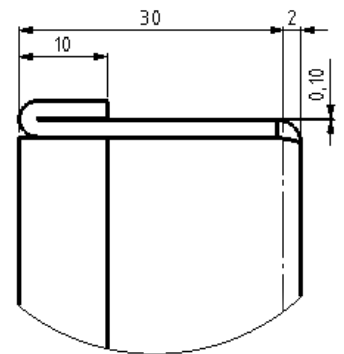
### Exercise 4-1



**Exercise 4-2**

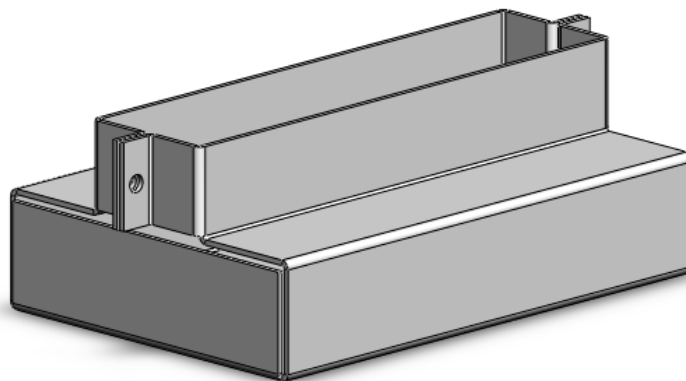


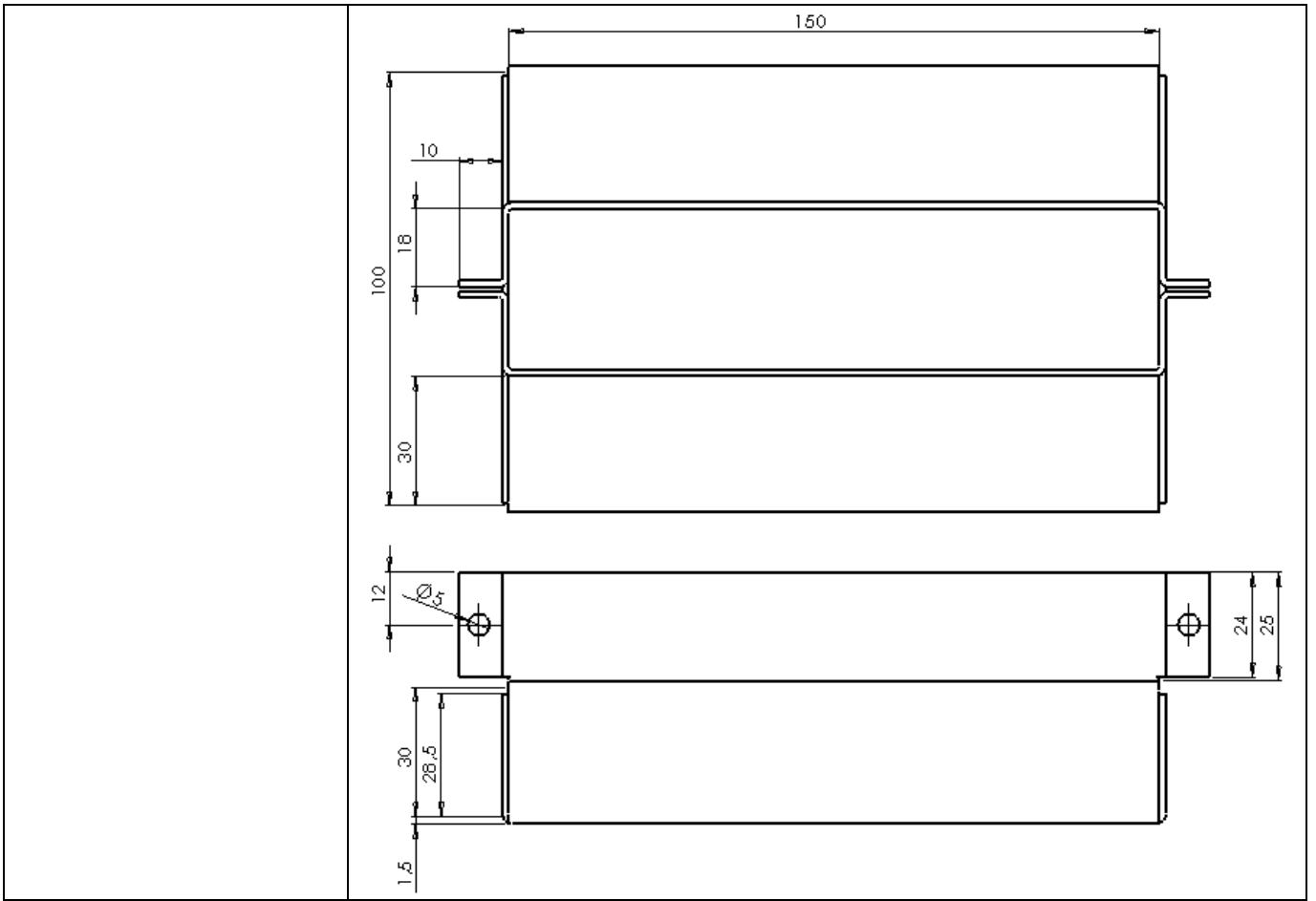
SECTION B-B  
SCALE 2 : 1



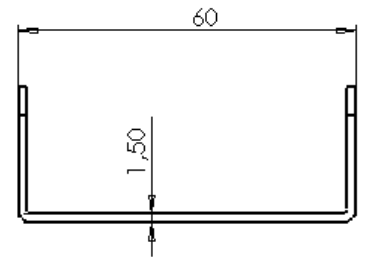
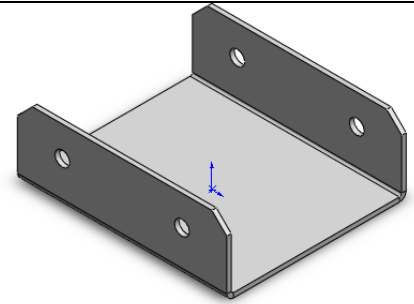
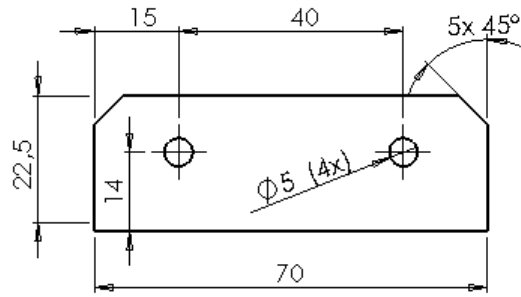
DETAIL A  
SCALE 2 : 1

**Exercise 4-3**

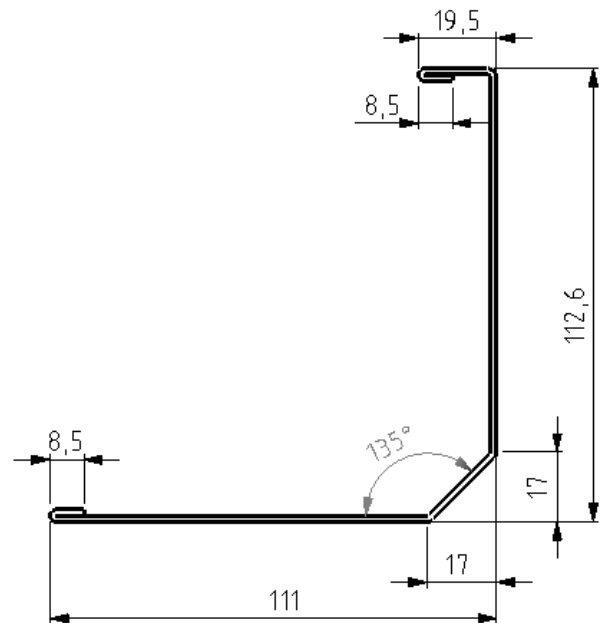
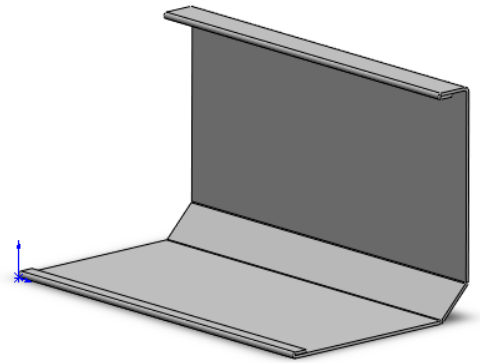
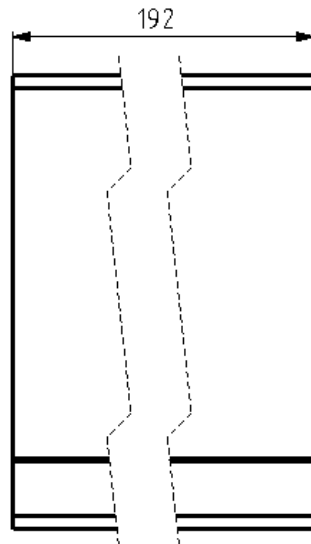




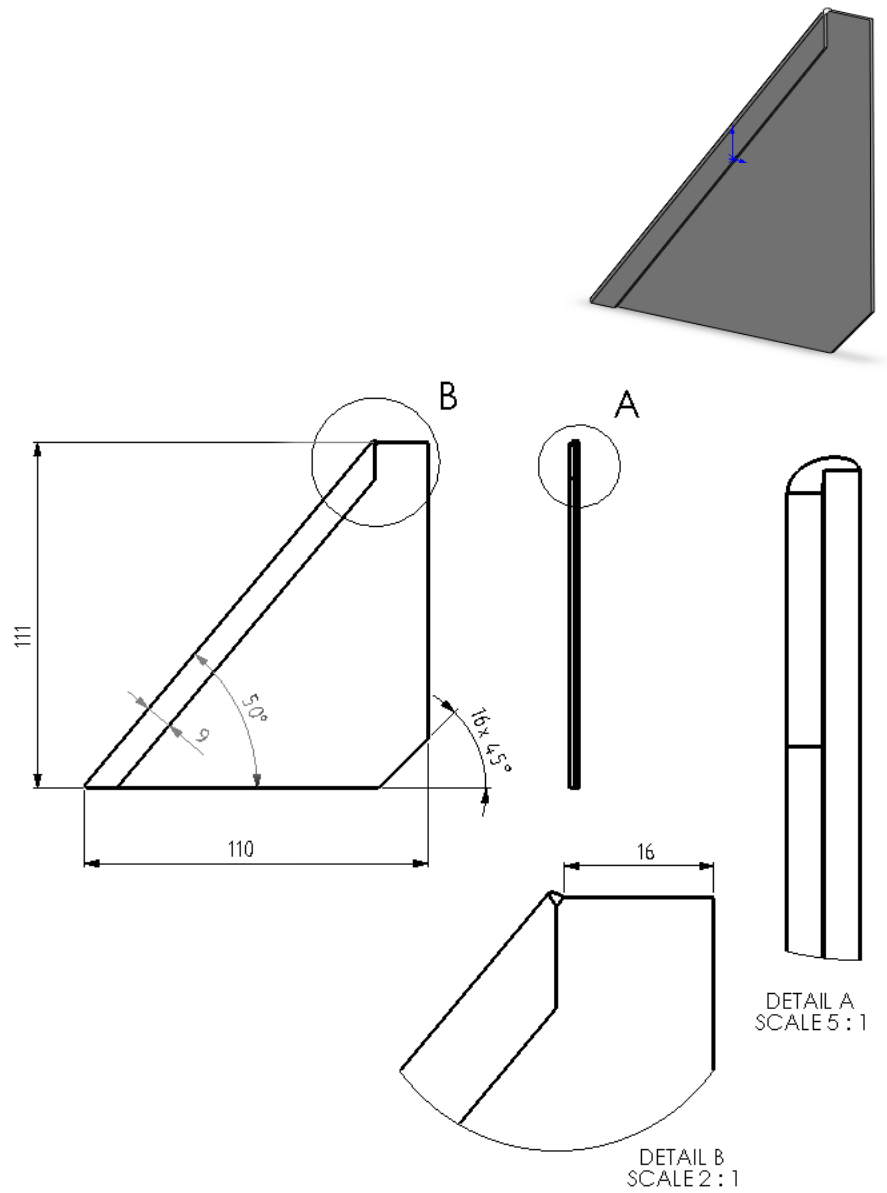
**Exercise 4-4**



**Exercise 4-5**

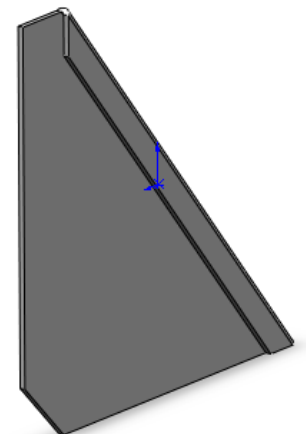


**Exercise 4-6a**



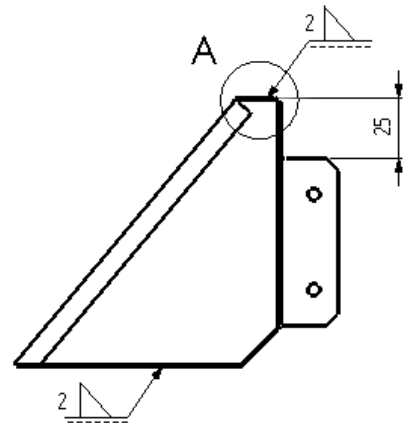
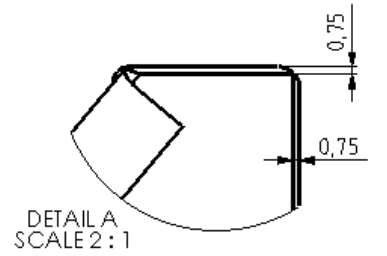
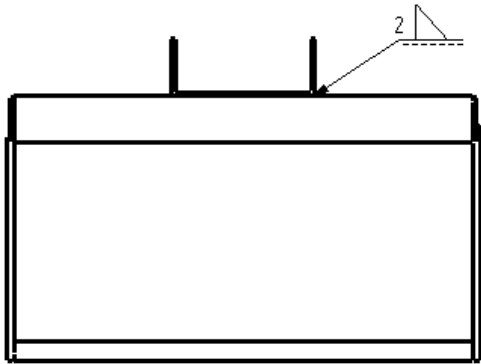
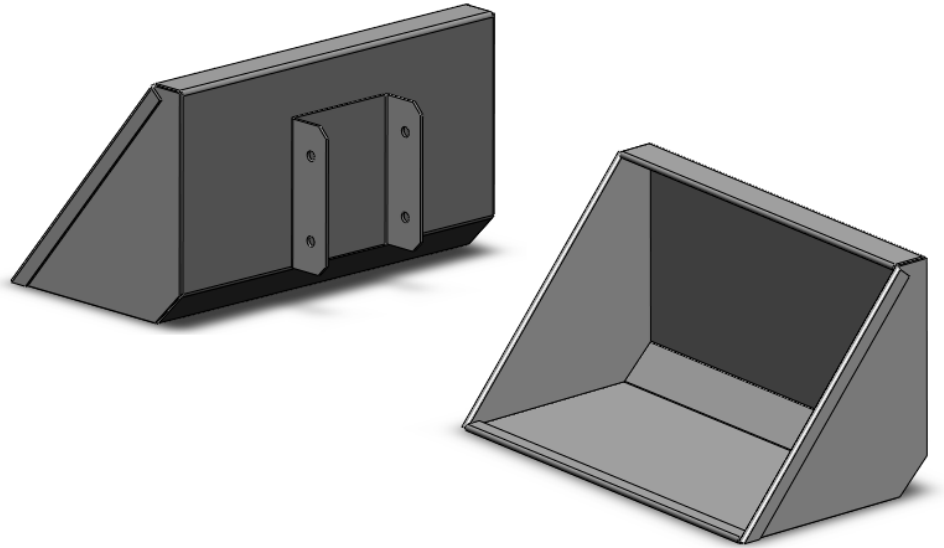
**Exercise 4-6b**

Make a mirror image version of part 6a. The folded edge is now on the other side.



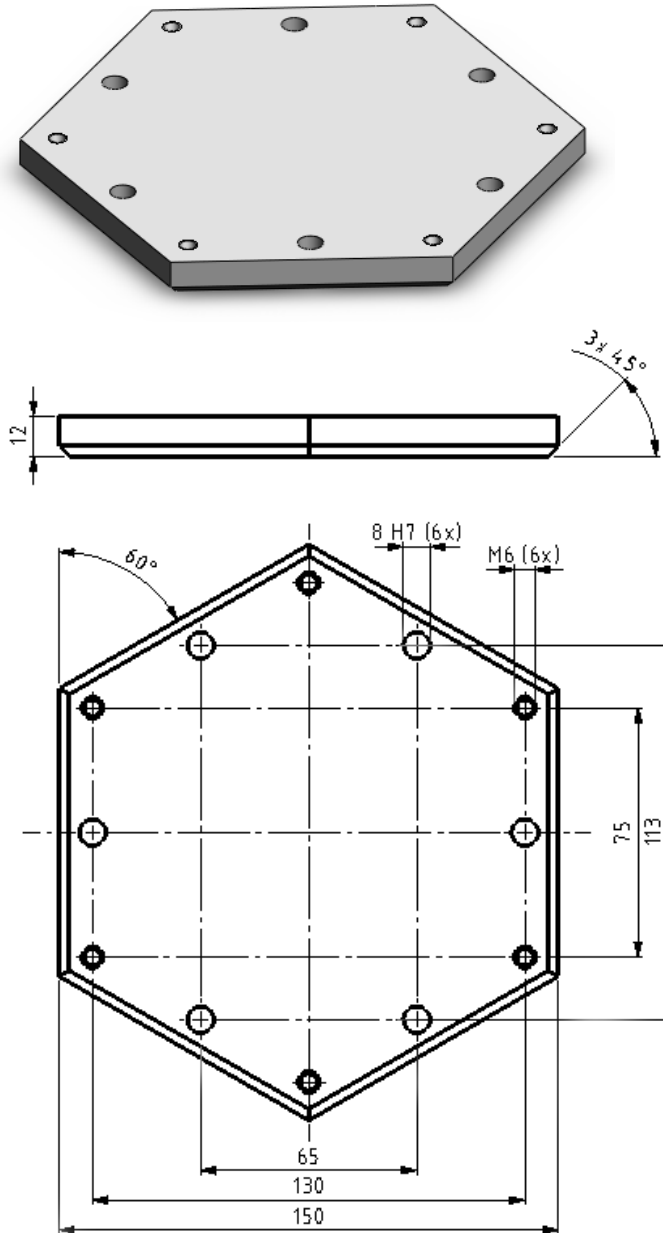
**Exercise 4-7**

Use parts 4-4, 4-5, 4-6a and 4-6b to construct the assembly as shown on the right.

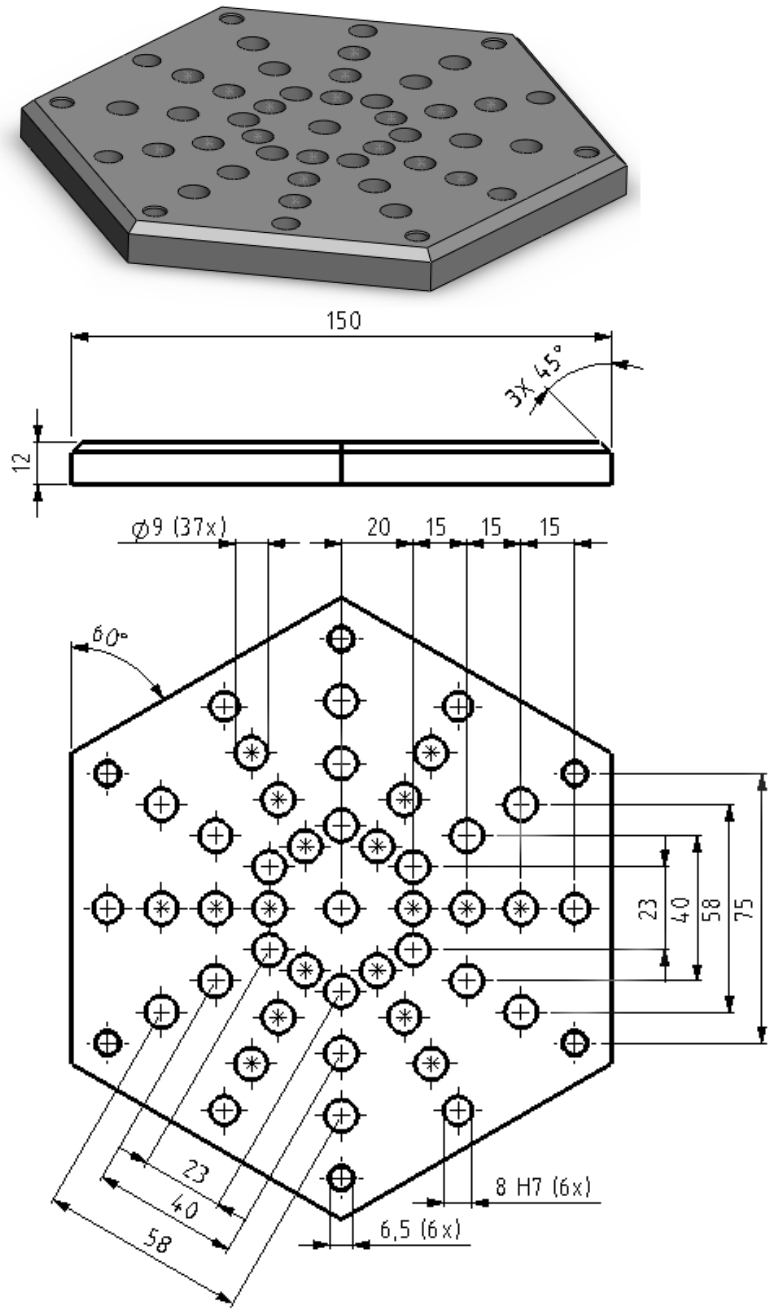


## Tutorial 5 Exercises

### Exercise 5-1



**Exercise 5-2**





**Exercise 5-3**

Technical drawing of a cylinder with a diameter of 8 d9 and a height of 24. The top edge is chamfered with a 1x45 degree chamfer. A top view shows a circle with a crosshair. To the right is a shaded 3D perspective view of the cylinder.

**Exercise 5-4**

Technical drawing of a hexagonal bolt with a total height of 30 and a hexagonal base with a width of 12. The upper cylindrical part has a diameter of 9 f9 and a height of 20. The top edge is chamfered with a 1x45 degree chamfer. The base has a fillet radius of R2 and chamfered edges with a 60 degree angle. A top view shows a hexagon with a width of 12 and a 60 degree chamfer. To the right is a shaded 3D perspective view of the bolt.

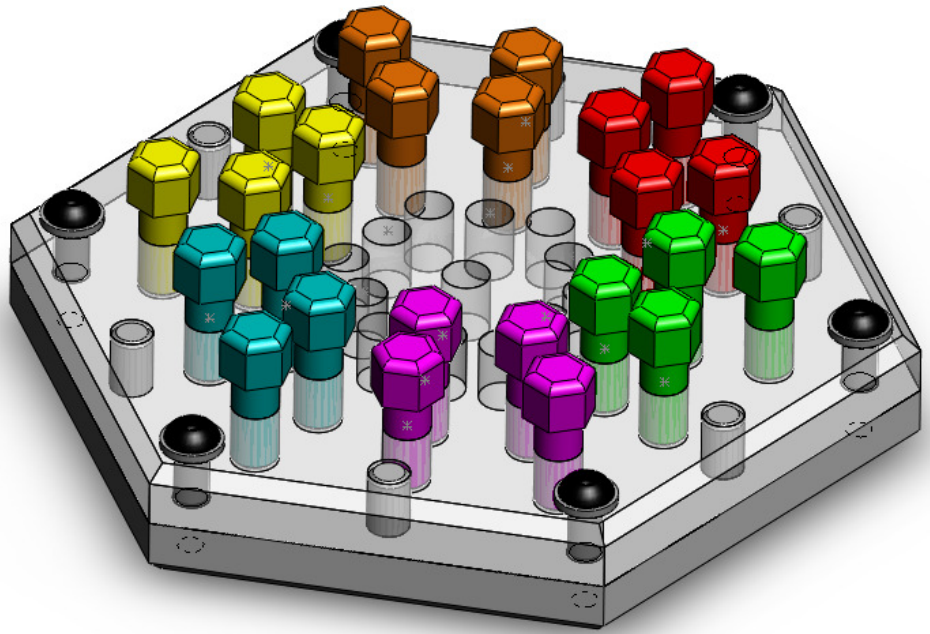
**Exercise 5-5**

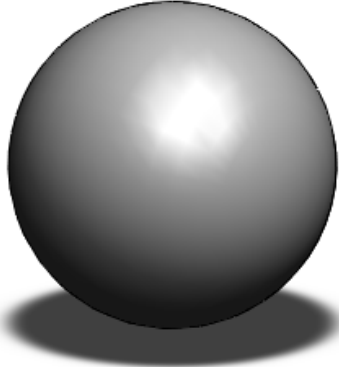
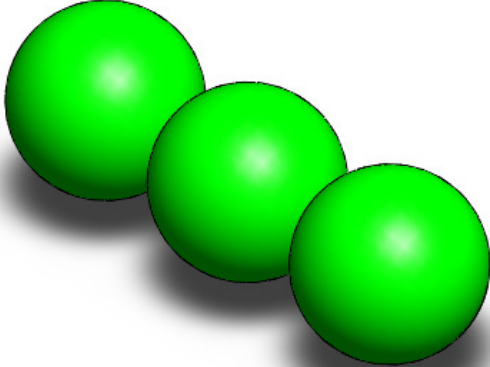
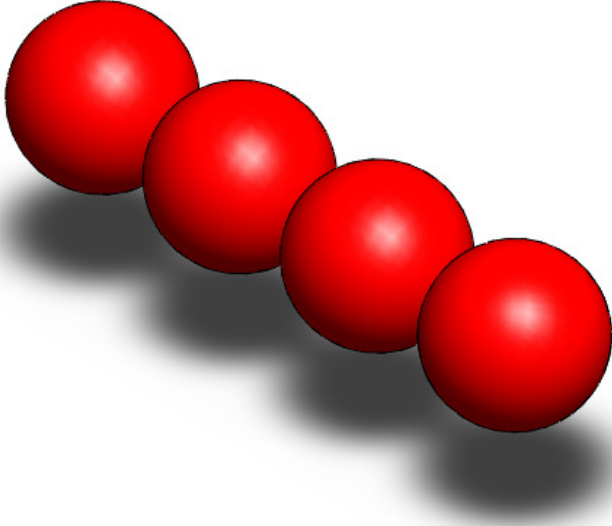
Make an assembly using parts 5-1 through 5-4 as shown on the right.

The two plates are positioned with dowel pins (5-3) and then secured using:

- Plain washer, Normal Grade A, ISO 7089-6 (M6)
- Socket Button Head Screw, ISO 7380 – M6x25

You will find both parts in the Toolbox.



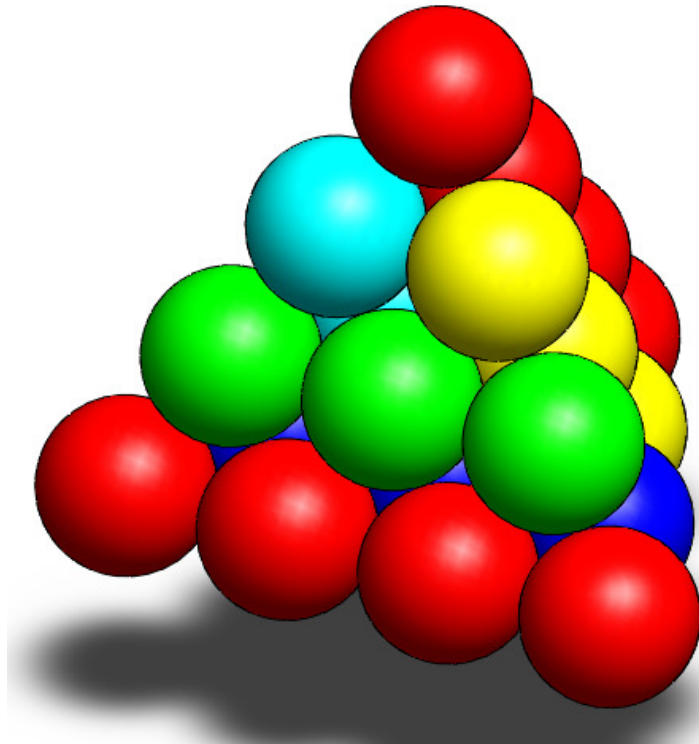
<p><b>Exercise 5-6a</b> Model a sphere with a diameter of 60mm.</p>	
<p><b>Exercise 5-6b</b> Make an assembly using the sphere from exercise 5-6a three times. The spheres should all sit in a row and touch each other.</p>	
<p><b>Exercise 5-6c</b> Make an assembly using the sphere from exercise 5-6a four times. The spheres should all sit in a row and touch each other.</p>	

**Exercise 5-7**

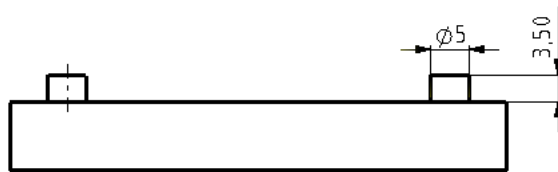
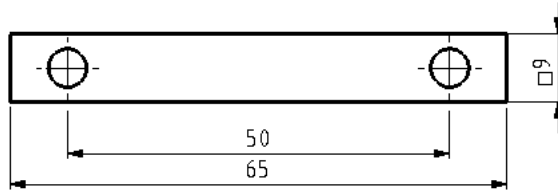
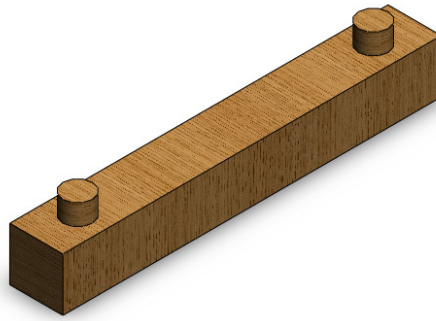
Make the assembly shown on the right. Use:

- 4 x assembly 5-6b
- 2 x assembly 5-6c

A different color is used for each occurrence of assembly 5-6b in the image on the right.

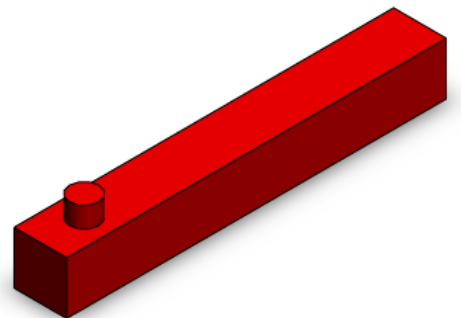


**Exercise 5-8**



**Exercise 5-9**

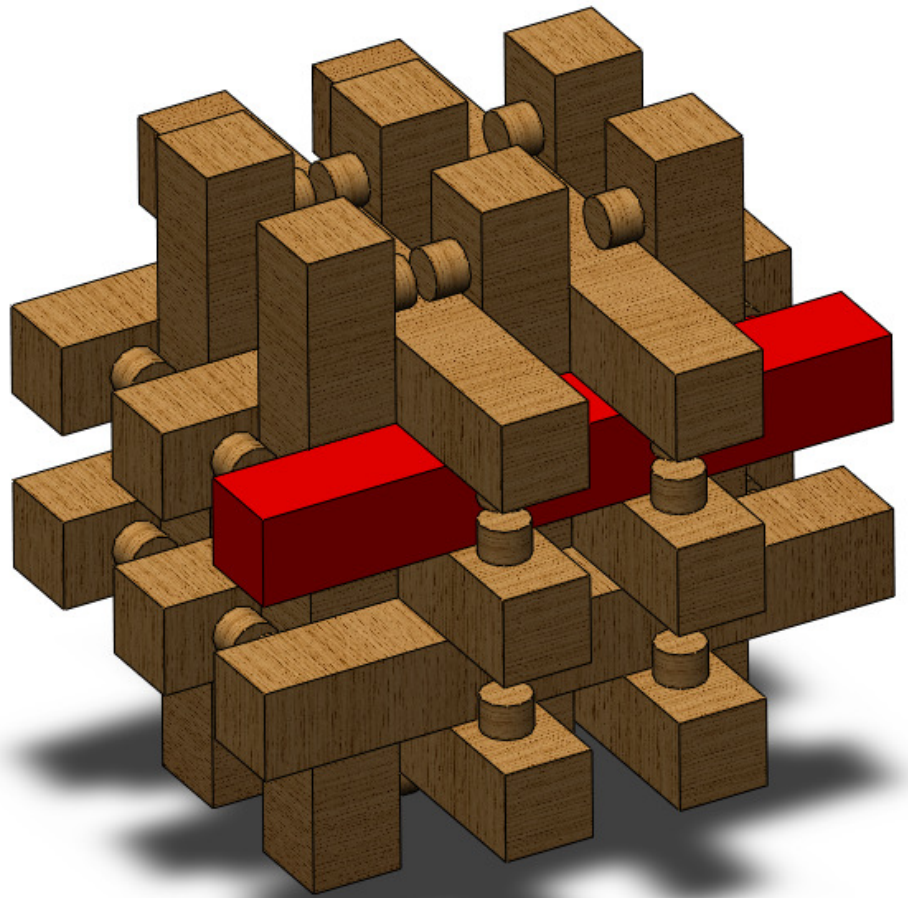
Make a new configuration for exercise 5-8 with just one pin.

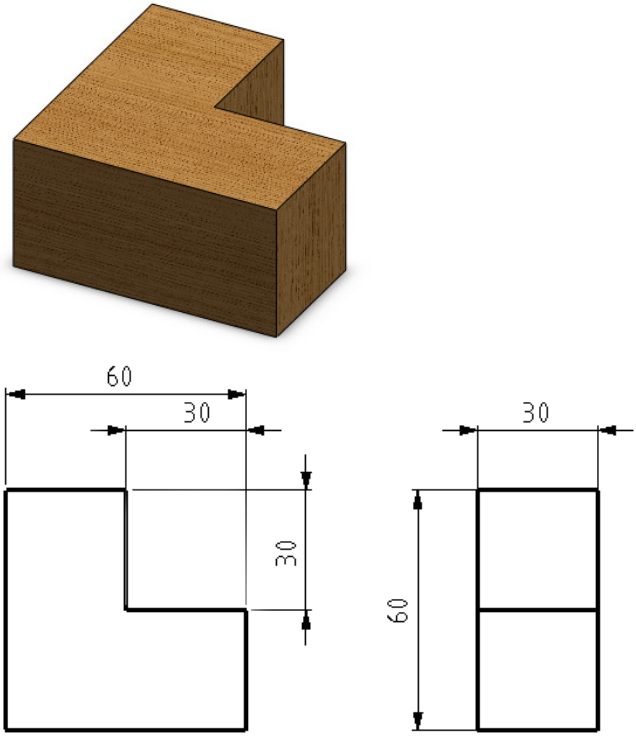
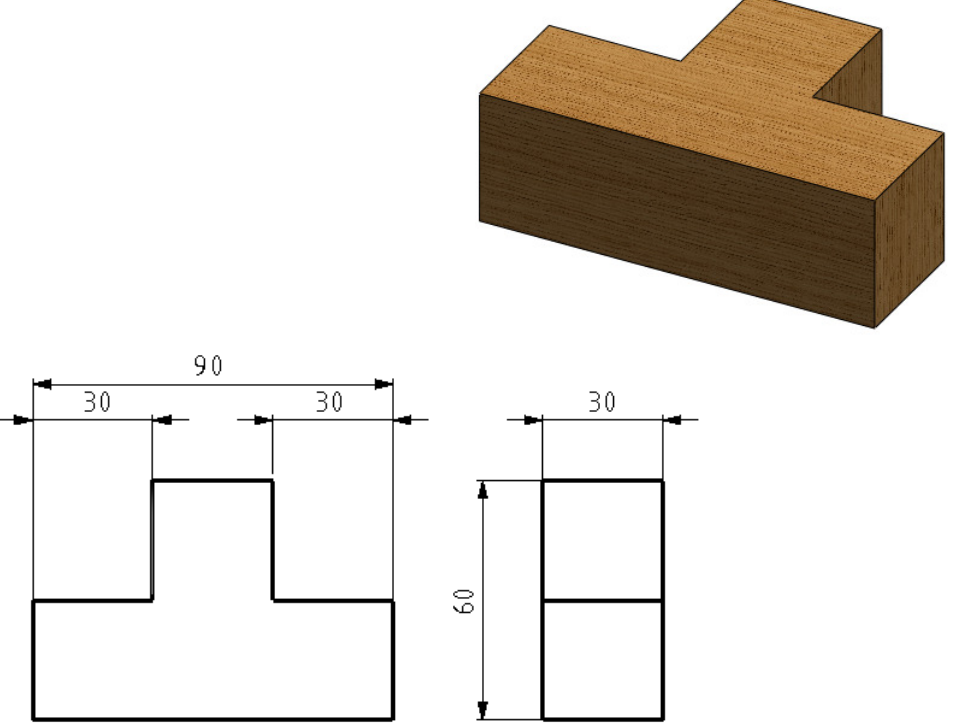
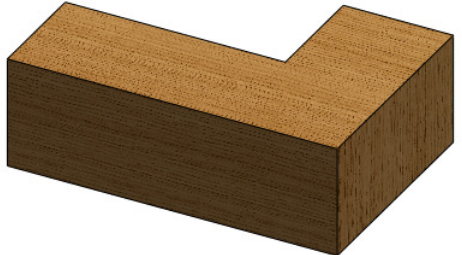


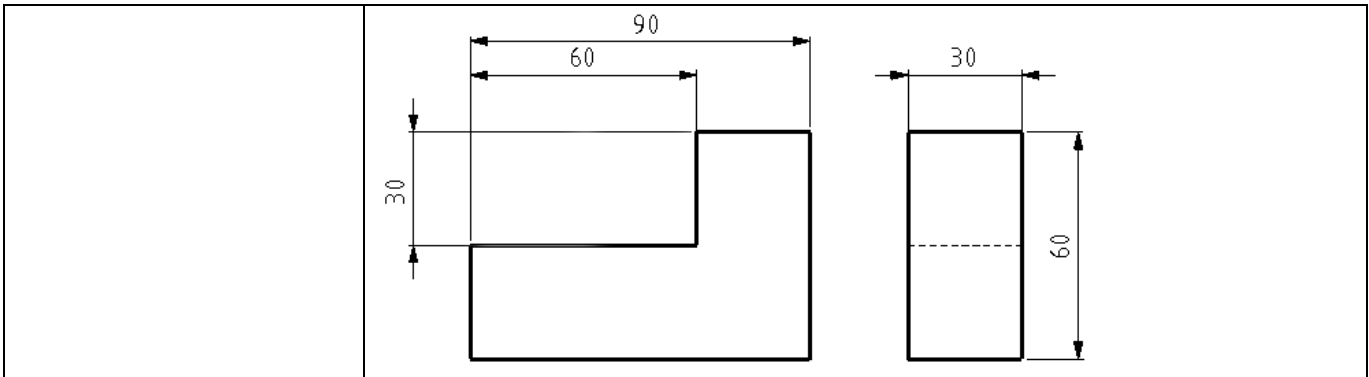
**Exercise 5-10**

Make the assembly. Use:

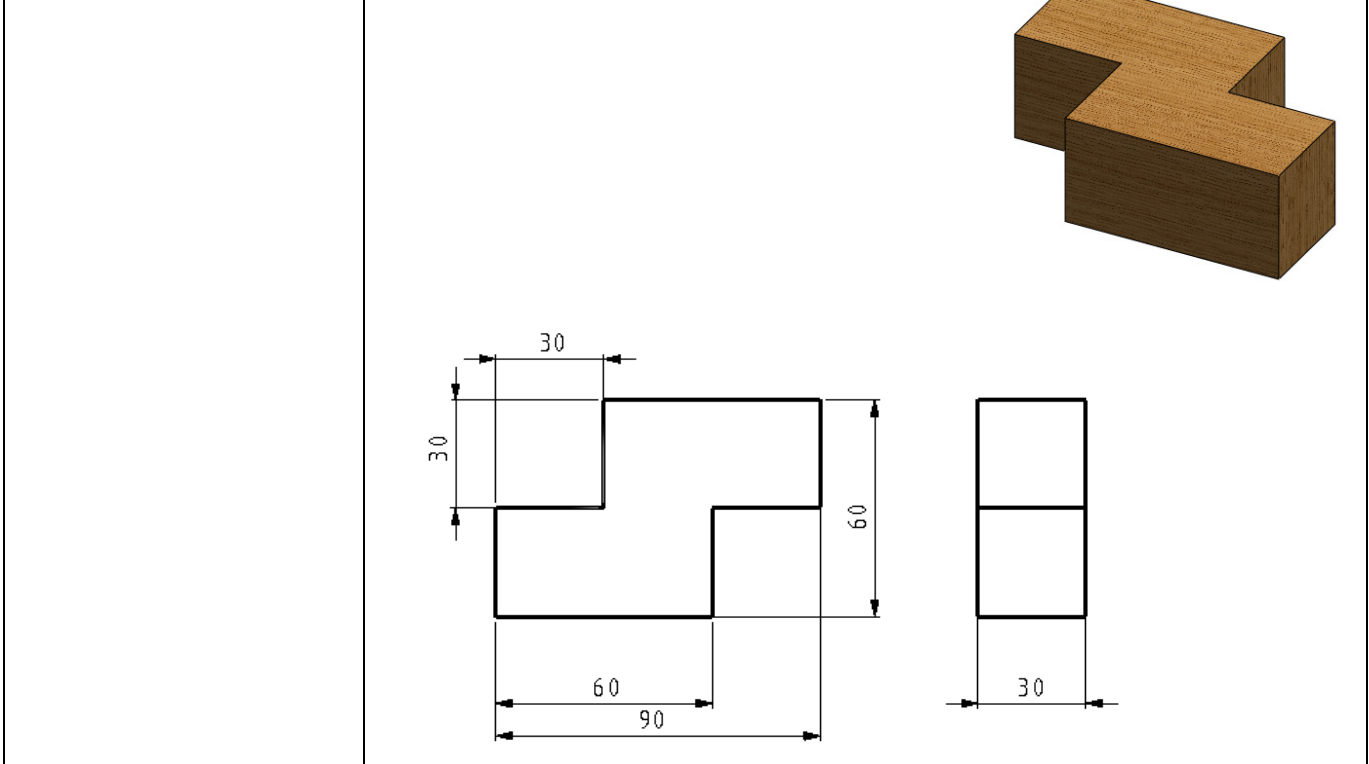
- 17 x exercise 5-8
- 1 x exercise 5-9



<p><b>Exercise 5-11</b></p>	 <p>The 3D model shows a rectangular block with a total length of 60 and a height of 60. A section of length 30 is cut out from the top, starting from the left end. The orthographic views show a front view with a total width of 60 and a height of 60, with a 30x30 rectangular cutout on the top right. The side view shows a total width of 30 and a height of 60.</p>
<p><b>Exercise 5-12</b></p>	 <p>The 3D model shows a rectangular block with a total length of 90 and a height of 60. A section of length 30 is cut out from the top, centered. The orthographic views show a front view with a total width of 90 and a height of 60, with a 30x30 rectangular cutout centered on the top. The side view shows a total width of 30 and a height of 60.</p>
<p><b>Exercise 5-13</b></p>	 <p>The 3D model shows a rectangular block with a total length of 90 and a height of 60. A section of length 30 is cut out from the top, centered.</p>



**Exercise 5-14**



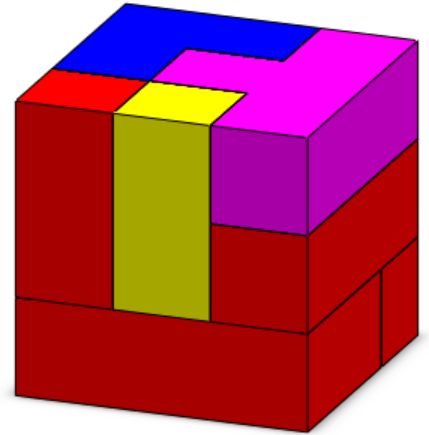
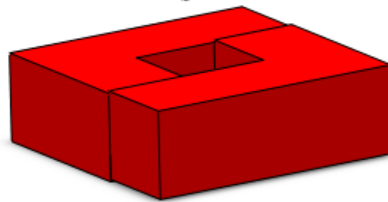
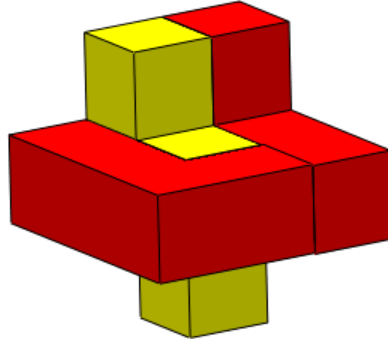
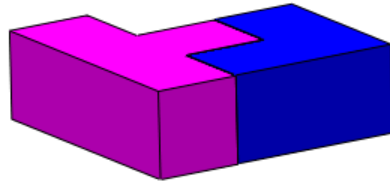


**Exercise 5-15**

Make the assembly as shown on the right. Use:

- 1 x exercise 5-11
- 1 x exercise 5-12
- 4 x exercise 5-13
- 1 x exercise 5-14

For the sake of clarity, a different color has been used for the various components shown on the right.



## SolidWorks is active in education

3D CAD is an indispensable part of today's technical world. Whether you specialize in the field of mechanical engineering, metal, metal electronics, industrial product design or car engineering: 3D CAD is *the* tool for today's designers and engineers.

In the Benelux countries, SolidWorks is the most popular 3D CAD software on the market. This is thanks to a unique combination of characteristics: great ease of use, wide usability and outstanding support. User requirements are always included in the software in the annual updates, which leads to an annual extension of functionality, but also to optimization of existing functions in the software.

### Education

A large number of educational organizations, ranging from organizations for secondary technical education to technical universities, have already chosen SolidWorks. Why?

For the **lecturer**, SolidWorks means choosing a user-friendly piece of software that pupils or students can master quickly. SolidWorks is therefore excellently suited to problem-driven education or competence-oriented education. Free tutorials in Dutch are available for various levels of education, such as a series of tutorials for junior and senior secondary technical education, which explain the basic principles of SolidWorks, or the Advanced Modeling tutorial, which deals with more complex topics, such as modeling complex double-curved surfaces. All of the tutorials are in Dutch and can be downloaded for free from [www.solidworks.nl](http://www.solidworks.nl).

For the **pupil or student**, learning SolidWorks is above all fun and challenging. SolidWorks makes technology much clearer and more accessible, so that working on tasks and projects is much more realistic and fun. Furthermore, every pupil or student knows that their employment prospects increase significantly if they can put SolidWorks, the most commonly used piece of 3D CAD software in the Benelux countries, on their CV. A large number of the vacancies and internships advertised at [www.cadjobs.nl](http://www.cadjobs.nl), for example, require knowledge of SolidWorks. This increases the motivation to learn how to use SolidWorks even more.

A Student Kit is available to make SolidWorks even easier to use. All students following a course that uses SolidWorks can download the Student Kit **for free**. The Student Kit is a full version of SolidWorks that may only be used for educational purposes. Your lecturer will provide you with the information needed to download the Student Kit.

For the **IT department**, choosing SolidWorks means that investments in new computers can be post-

poned, as SolidWorks has relatively modest hardware requirements. SolidWorks is very simple to install and manage in a network environment, thanks to the use of network licenses, among other things. Should you still experience problems, our qualified help desk team will get you back on track quickly.

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When you have mastered SolidWorks sufficiently, you can take the CSWA exam. CSWA is short for Certified SolidWorks Associate. On successful completion of the exam, you will be awarded a certificate, which you can use to show that you have mastered SolidWorks sufficiently. This is useful when applying for jobs or internships.

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### Contact

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Further information about SolidWorks is available on the website: <http://www.solidworks.nl>

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