Solidworks tutorial

Loft project

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SW.
Advisor

This is the process of designing a simple buckle to exercise solidowrks loft feature:
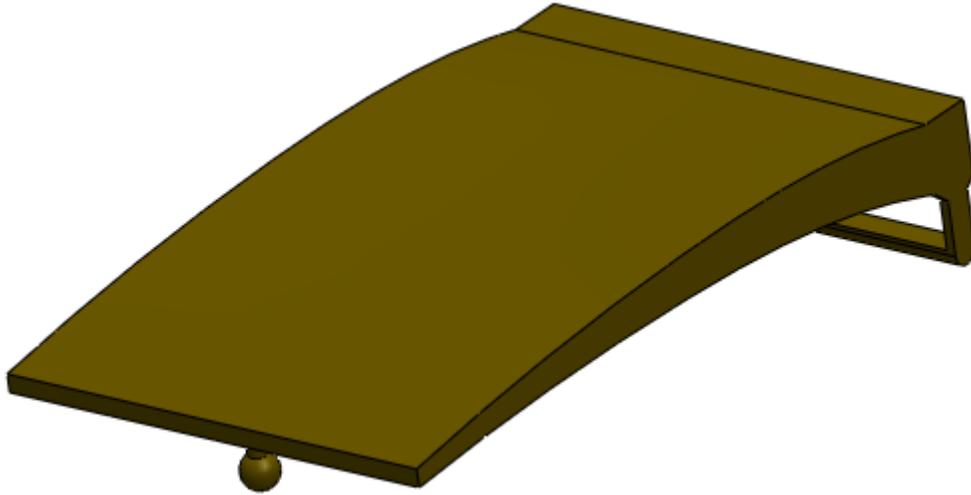
In this tutorial I assume that you know the basic of solidworks so you should know how to interactively work with the software . this means that you should know how to open a sketch , define a plane and … . but if you don't know the bascis I recommend you to visit the solidowrks beginner tutorials first.
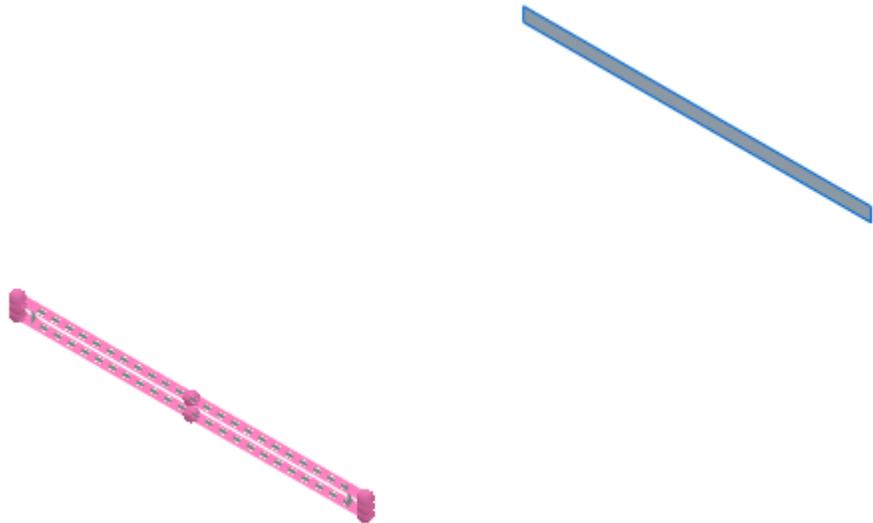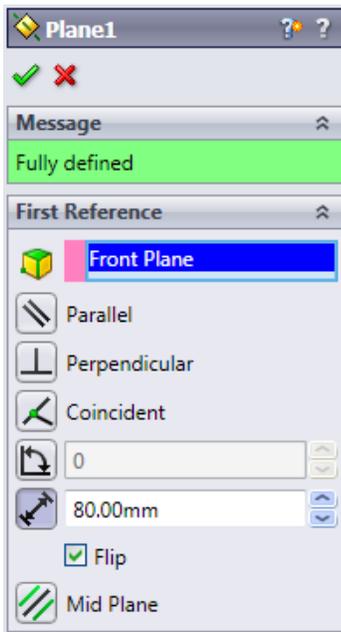
What we use to accomplish this model :

- Plane
- Sketch
- Line
- Circle
- Spline
- Extruded boss/base
- Revolved boss/base
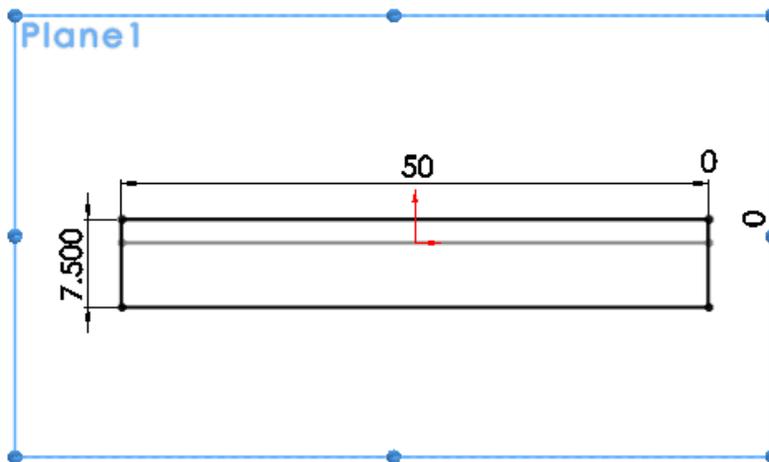- Extruded cut
- Loft
- Add relation

I guess everyone is familiar with the buckle and that's why I choose it as a model . first of all we are going to make the main body of the model which will be created by loft . to do so we need at least two sketches . the first sketch is on the front plane so open a sketch on the front plane and draw a rectangle like the following picture:

50

2
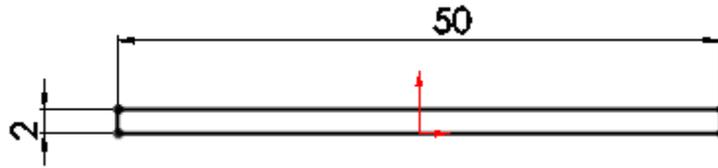
Exit the sketch . for the second sketch we need a plane so define a plane parallel to the front plane at the distance of 80mm . click flip if necessary to get the right direction .

Plane1

Message

Fully defined

First Reference

Front Plane

Parallel

Perpendicular

Coincident

0

80.00mm

☑ Flip

Mid Plane

Open a new sketch on plane1 and draw another rectangle just like this :
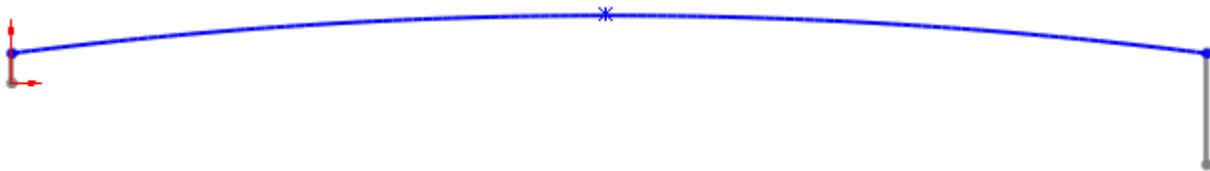
Plane1

50   0

7.500   0

Exit the sketch and go to isometric view . now you have two profiles to join by loft feature . hide the plane1 and click on the " lofted boss/base" in the features tab . now you are ready to select your profiles . click on the sketch1 and sketch2 respectively .
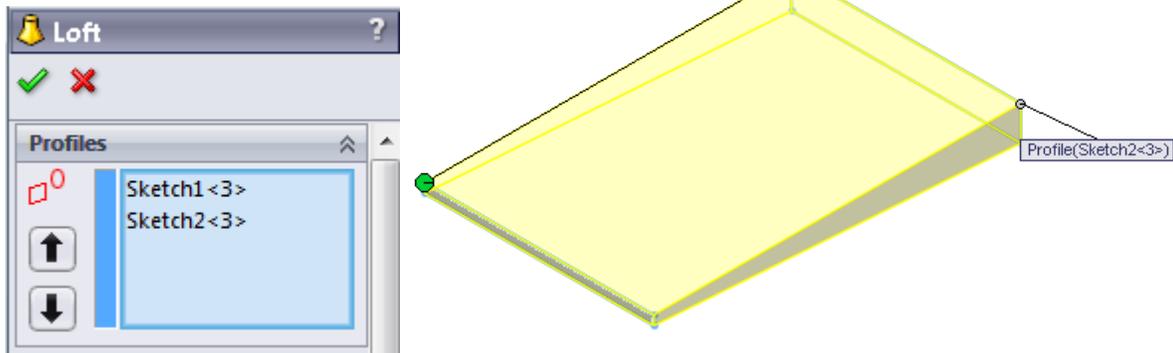
as you can see the preview , solidworks will join two profiles but this is not what we really want . we need some curvature on the surface of the body so what should we do?! The answer are guide curves .guide curves will tell the software how to join the profiles . you can draw any guide curve you want to get the desired shape just keep in mind that they should be coincident with profiles . sometimes because of tangency difficulties you cant use any curve you want and when it happens just change the curvature of the guide or play with green points when you are running loft feature .

making guide curves in this model that we are working on is easy . for the first guide curve open a sketch on the right plane and draw an arc just like the picture . I didn't fully defined the arc but you can do it if you want .

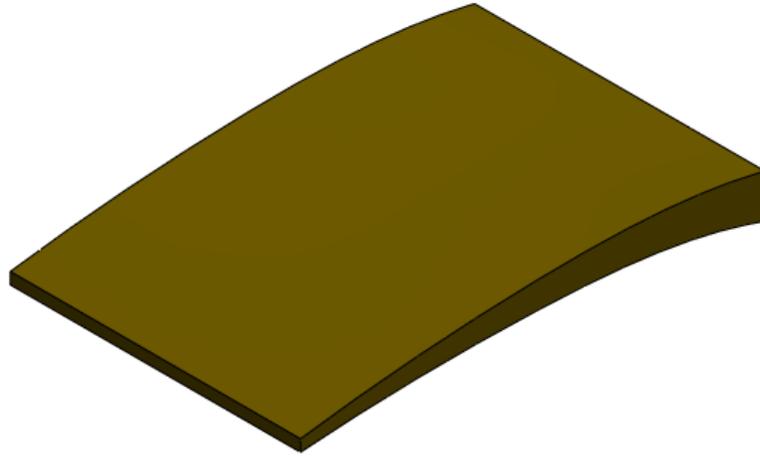Exit the sketch and once again open a sketch on the right plane and draw another curve like this :
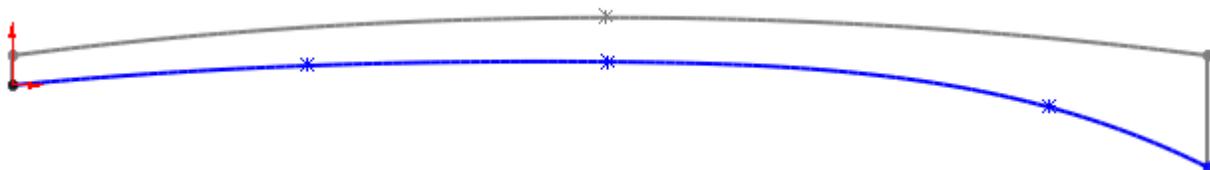
Exit the sketch and go to isometric views to see what you 've done
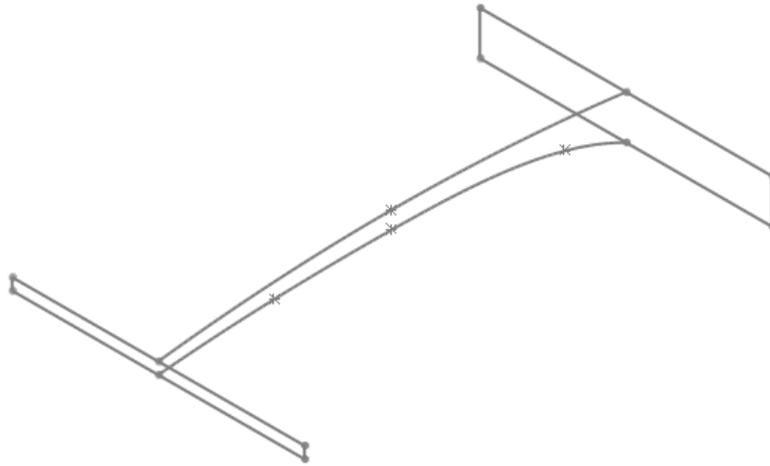
With these guide curves and profiles now you are ready to use the loft feature . Click on the "lofted boss/base " on the features tab . select the sketches again just like the first time but this time click in box of guide curves too and select the two curves:
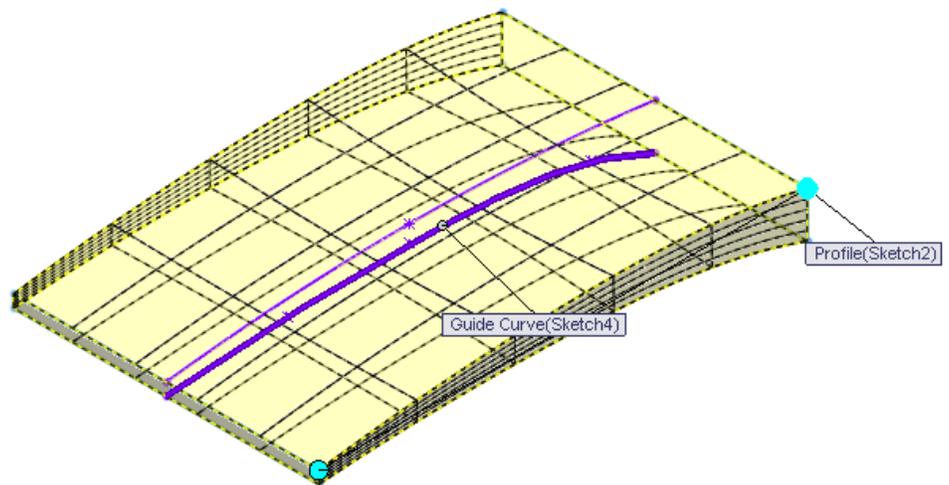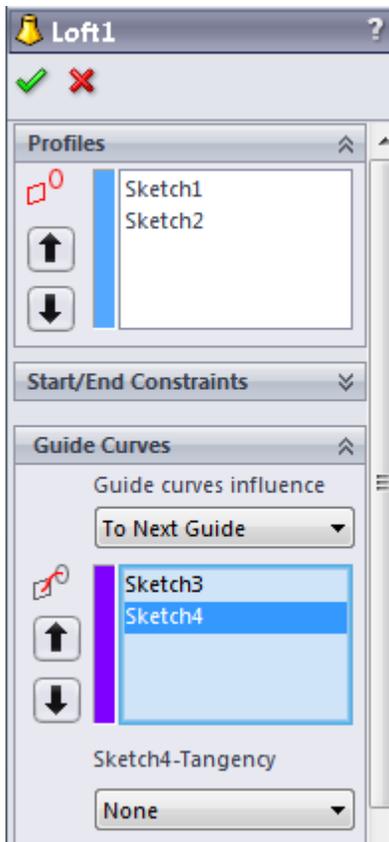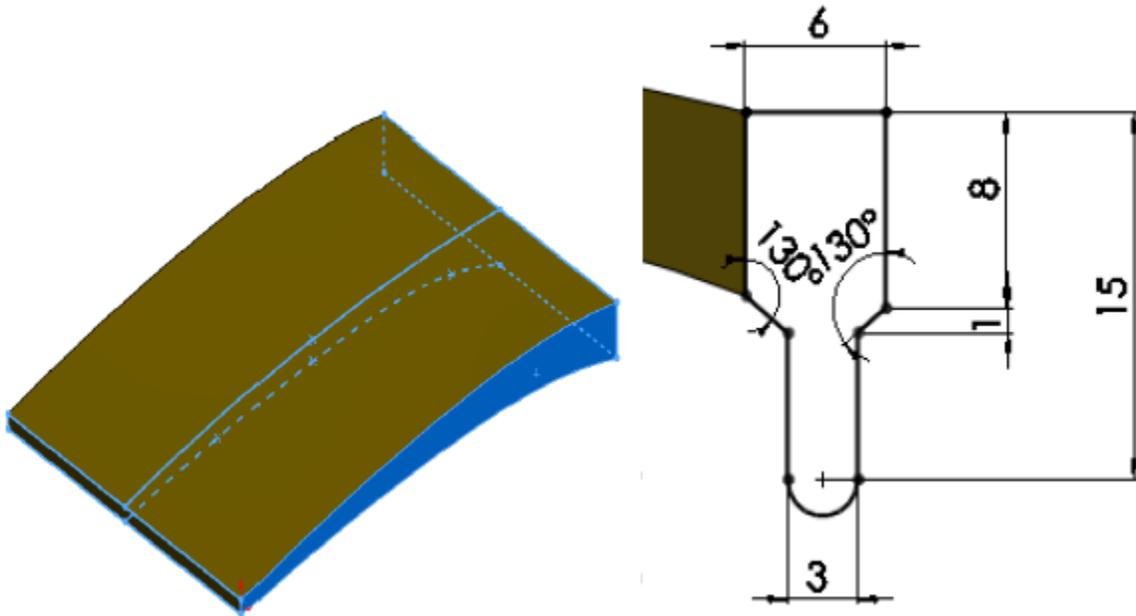
Now you see the differences from the first time . click ok and get the result .
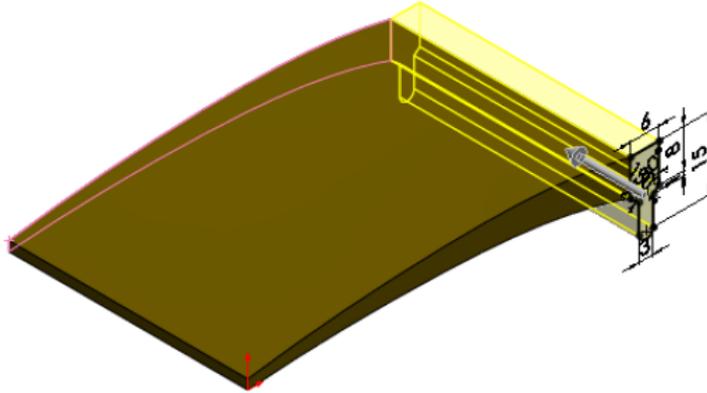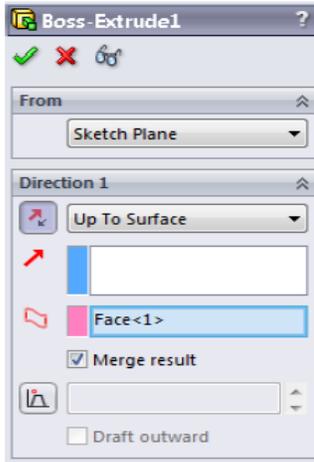
Now you have done the hardest part . what we do in the rest of the tutorial is just to make it a little bit like a real buckle
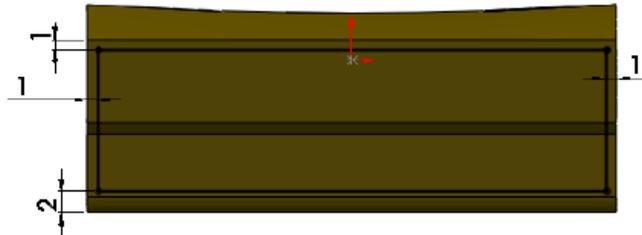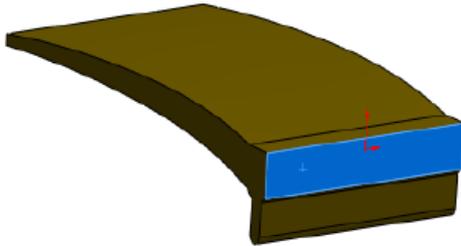.
Select the right side of your model and open a sketch on it . draw a sketch like following picture :
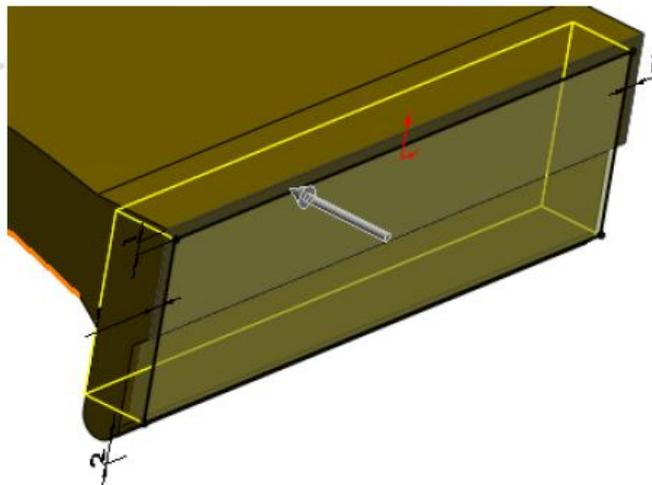
Exit the sketch and extrude it to the opposite side .

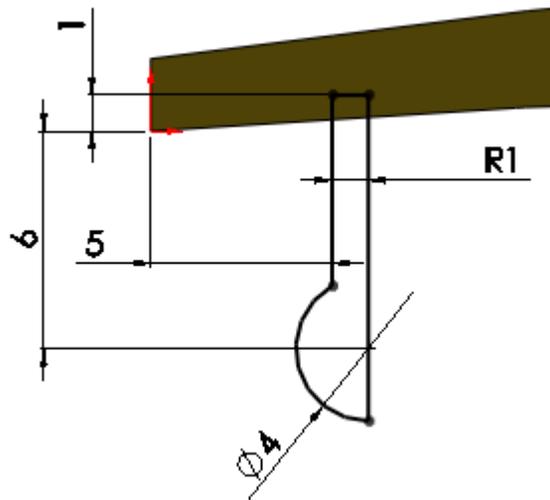Select the face shown here and open a sketch on it . draw the sketch .
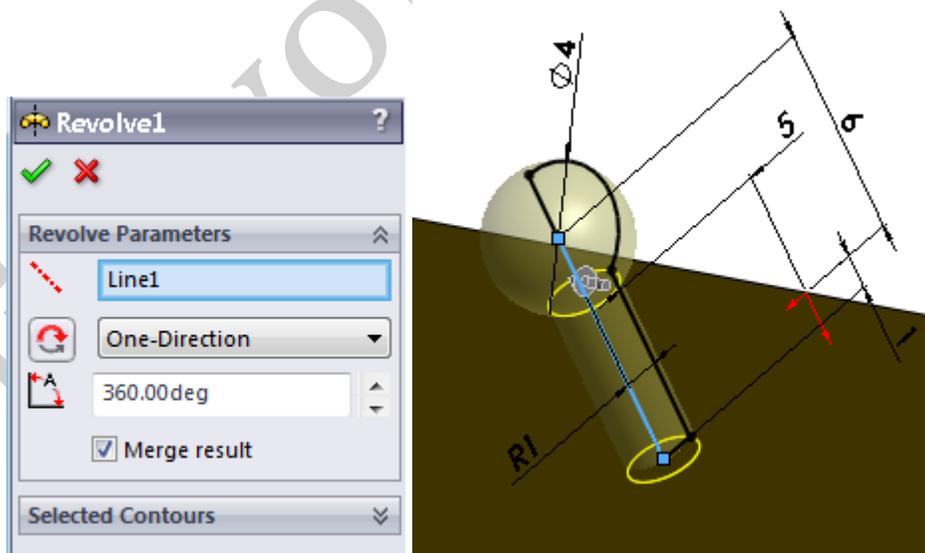
then cut it like this :

just one more step to go . I just tried to keep this tutorial nice and easy and if you were familiar with the basics you can finish it in less than a half of hour .
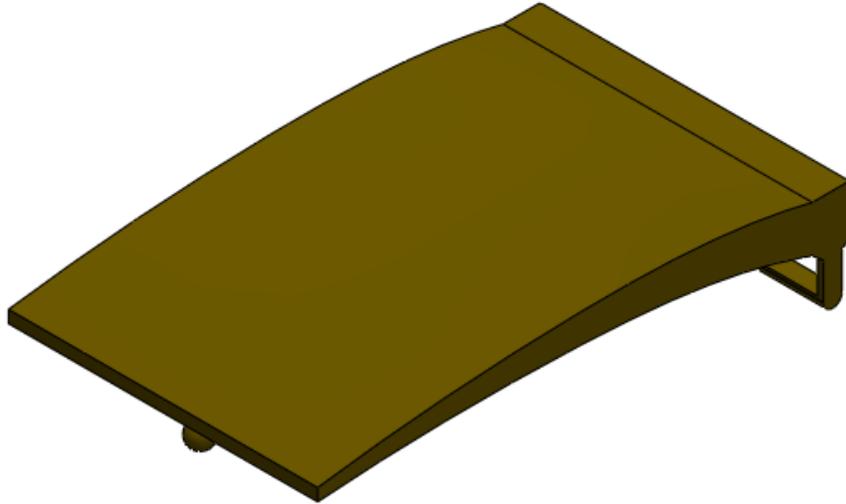
so what is the last step?!

Open a sketch on right plane and create your final sketch just like the following picture .

Now revolve the sketch around its largest line to shape a cylinder like this :

Now it's done:

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