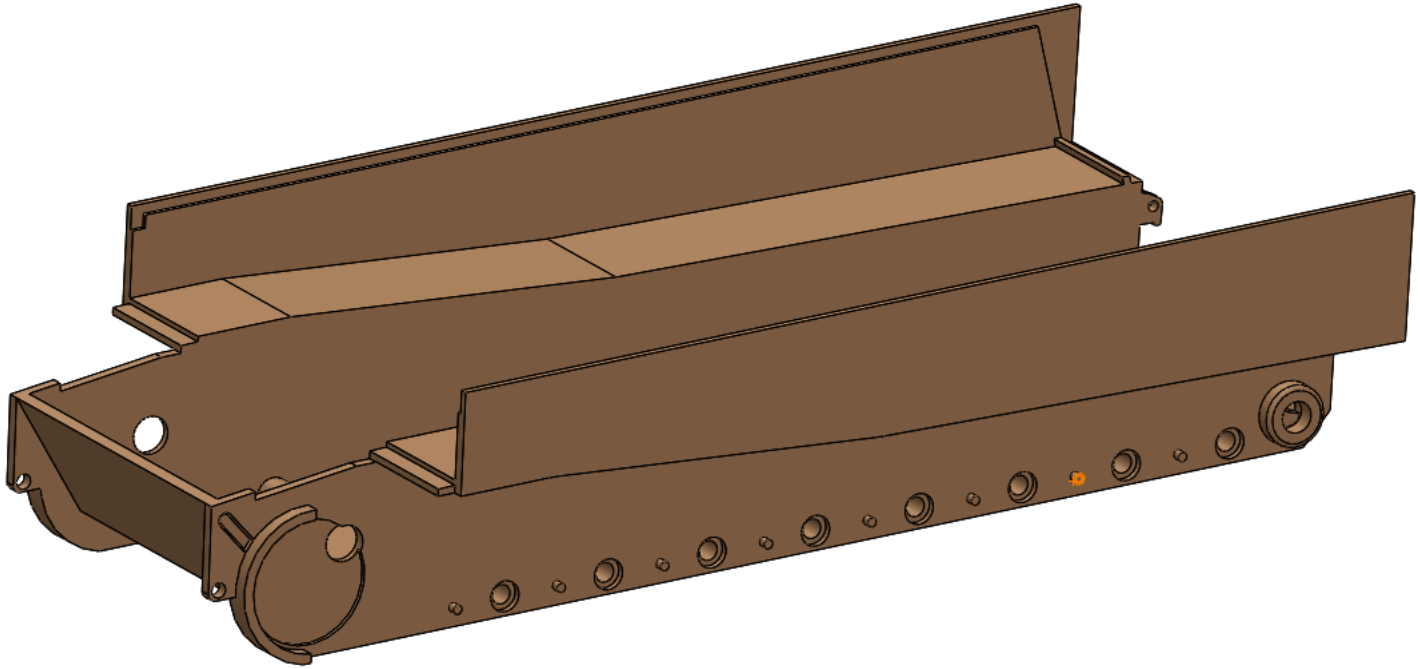
SOLIDWORKS TUTORIALS

TIGER TANK I

PDF TUTORIAL

Part: Chassis
Difficulty: Medium
Pages: 88

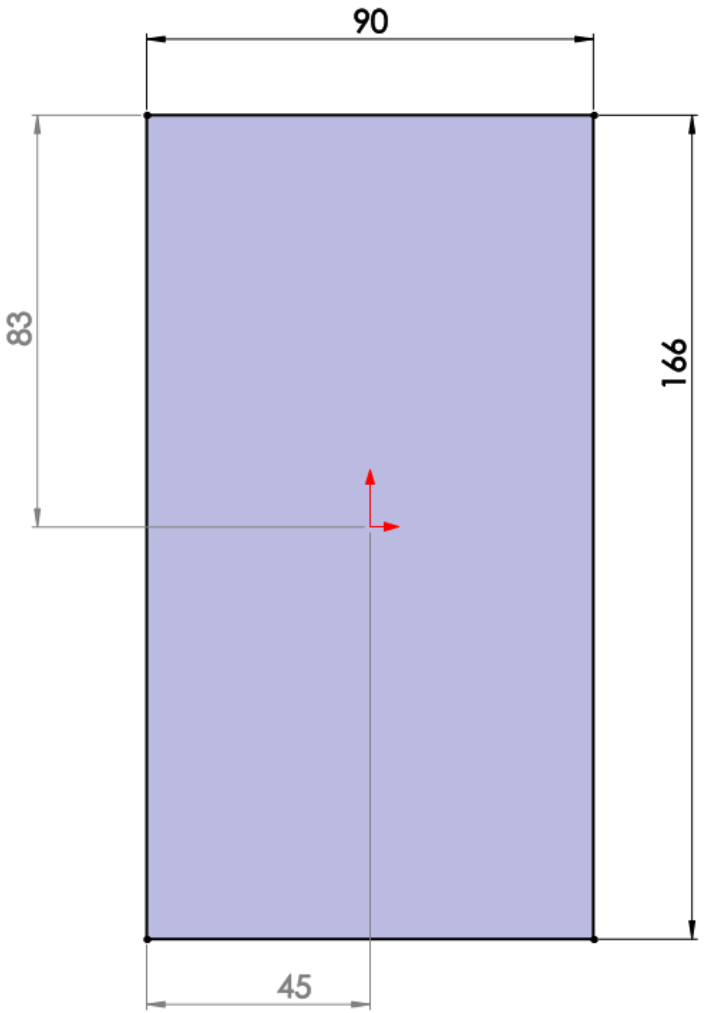
A TAMIYA, INC. Model



PART NAME: Chassis

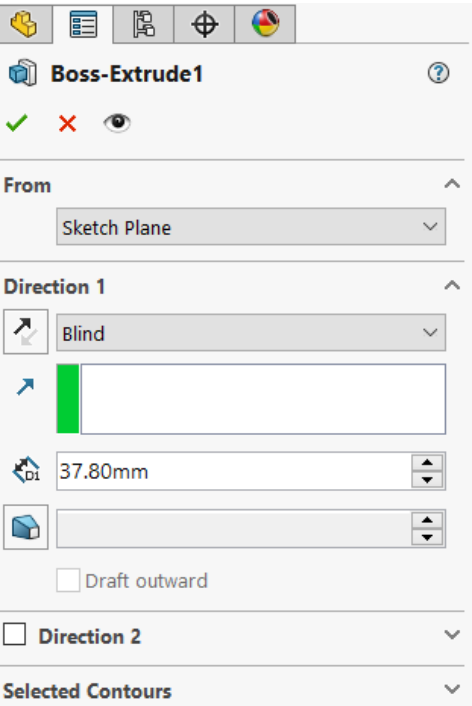
CLARIFICATION: Many of the features on this specific part are only for optical purposes only and do not carry any function. There are always other ways than the ones pointed out here, to achieve the same results. Don't be afraid to explore your options.

All Dimensions are in millimetres

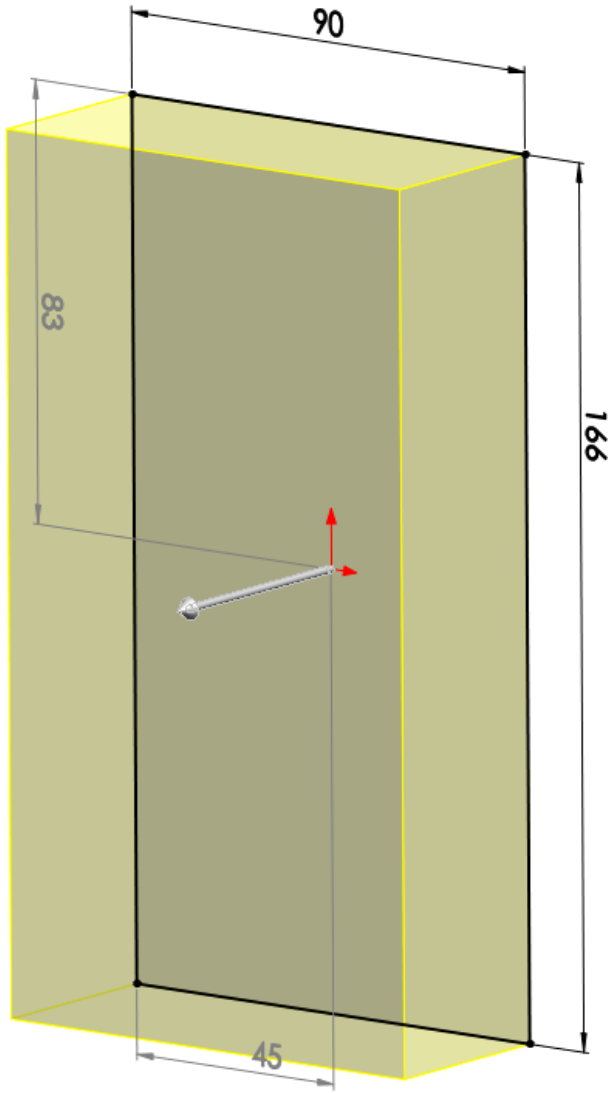


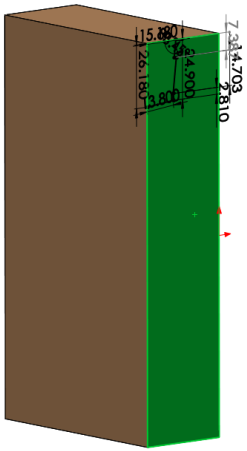
Draw the sketch on the **Front Plane** and set its dimensions.

Set the "Boss-Extrude" at *Mid Plane* and extrude it for **87.80**



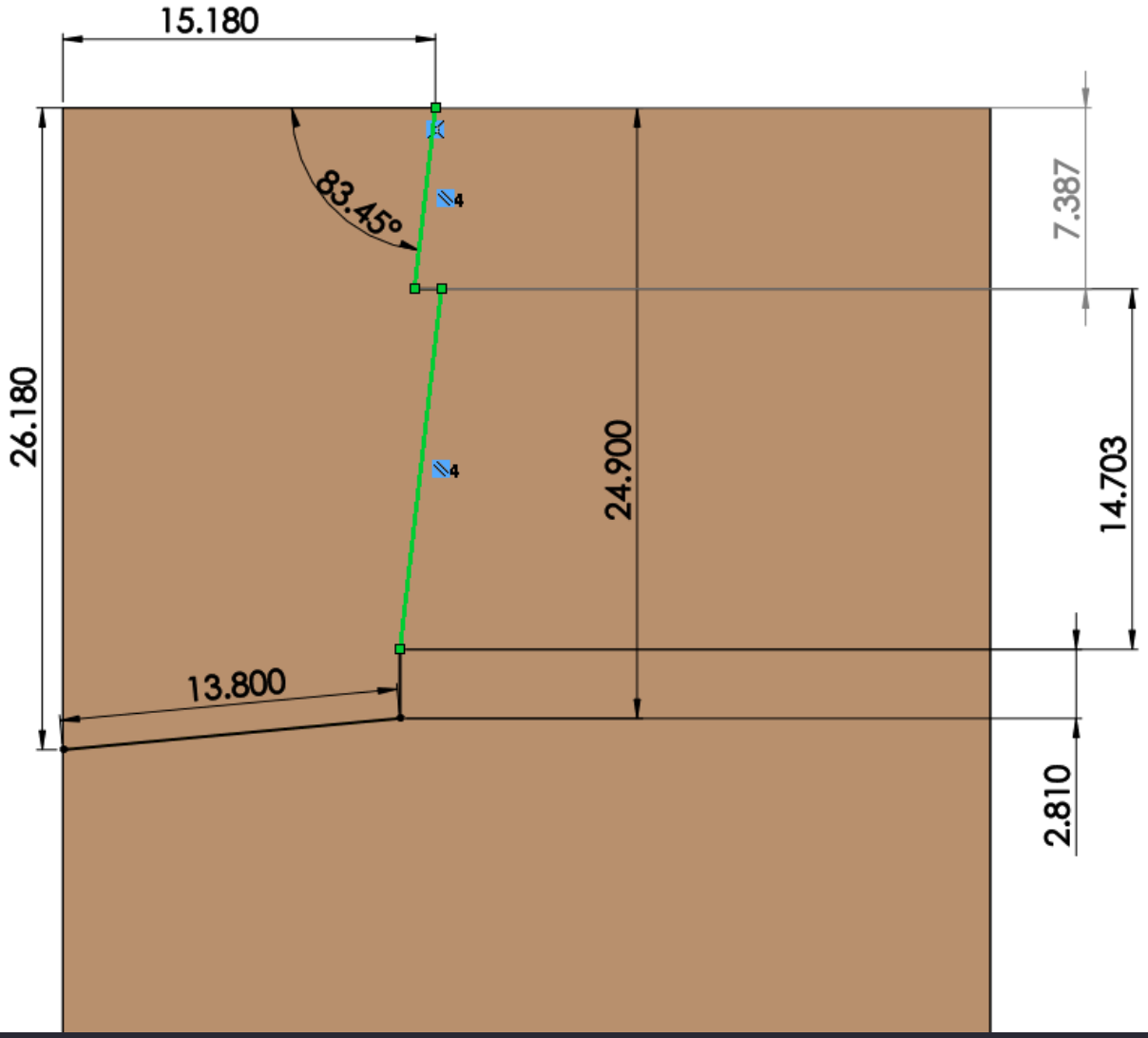
Use “Boss-Extrude” and extrude the sketch for **37.80**

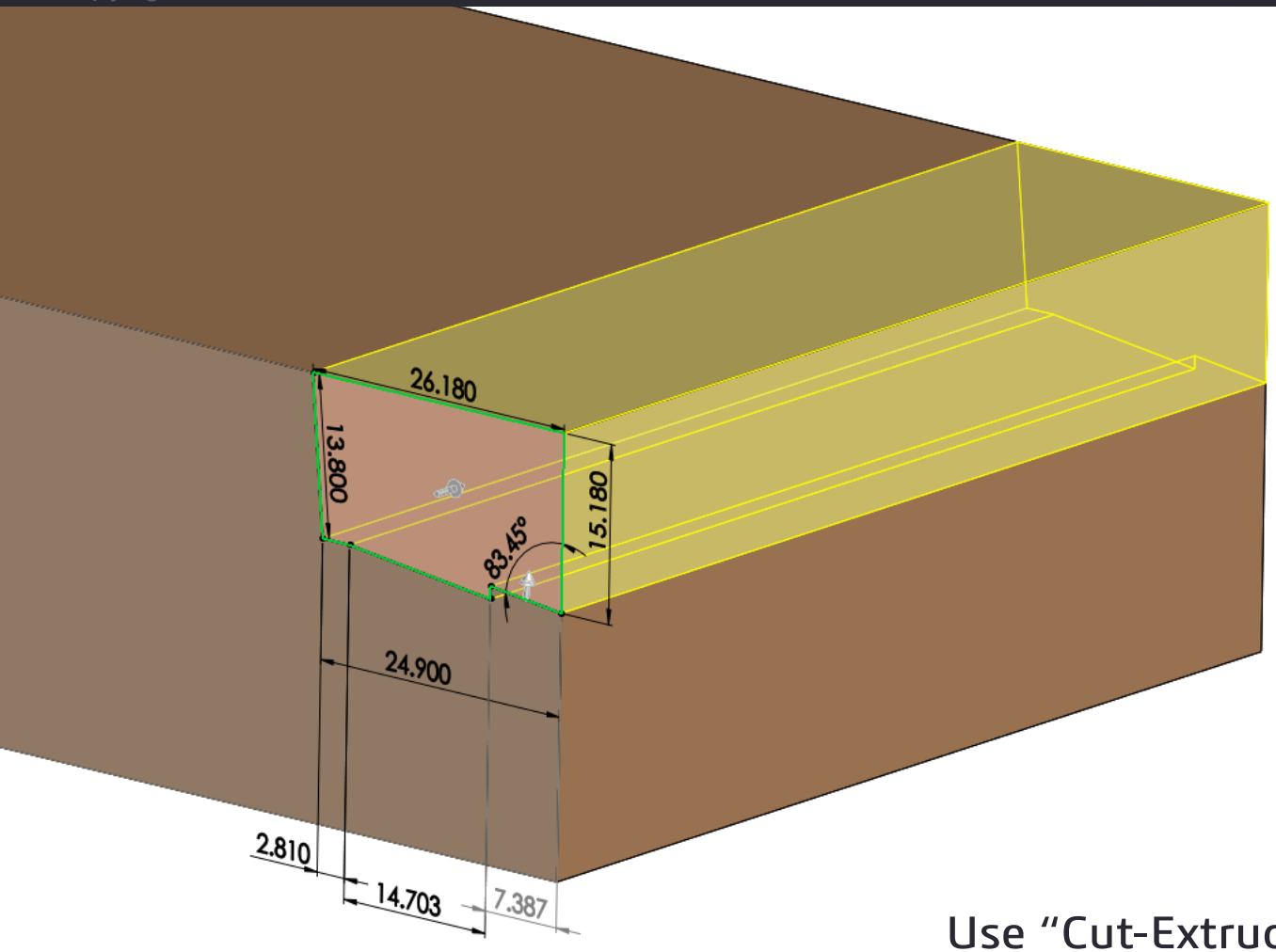


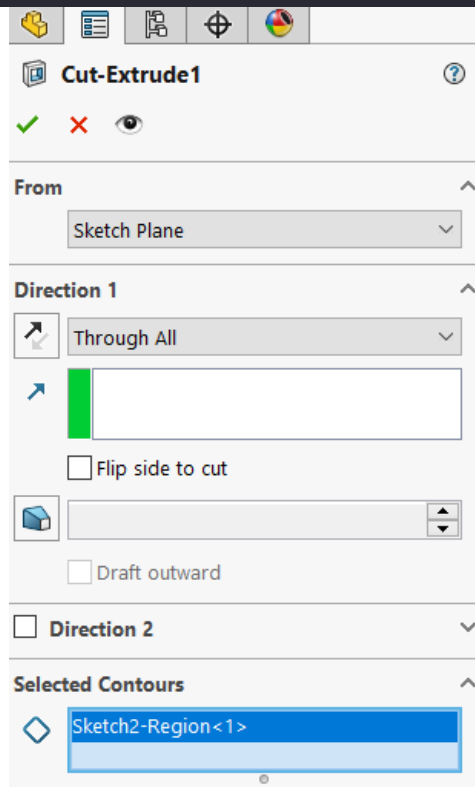


Draw the sketch on the green highlighted surface of the part.

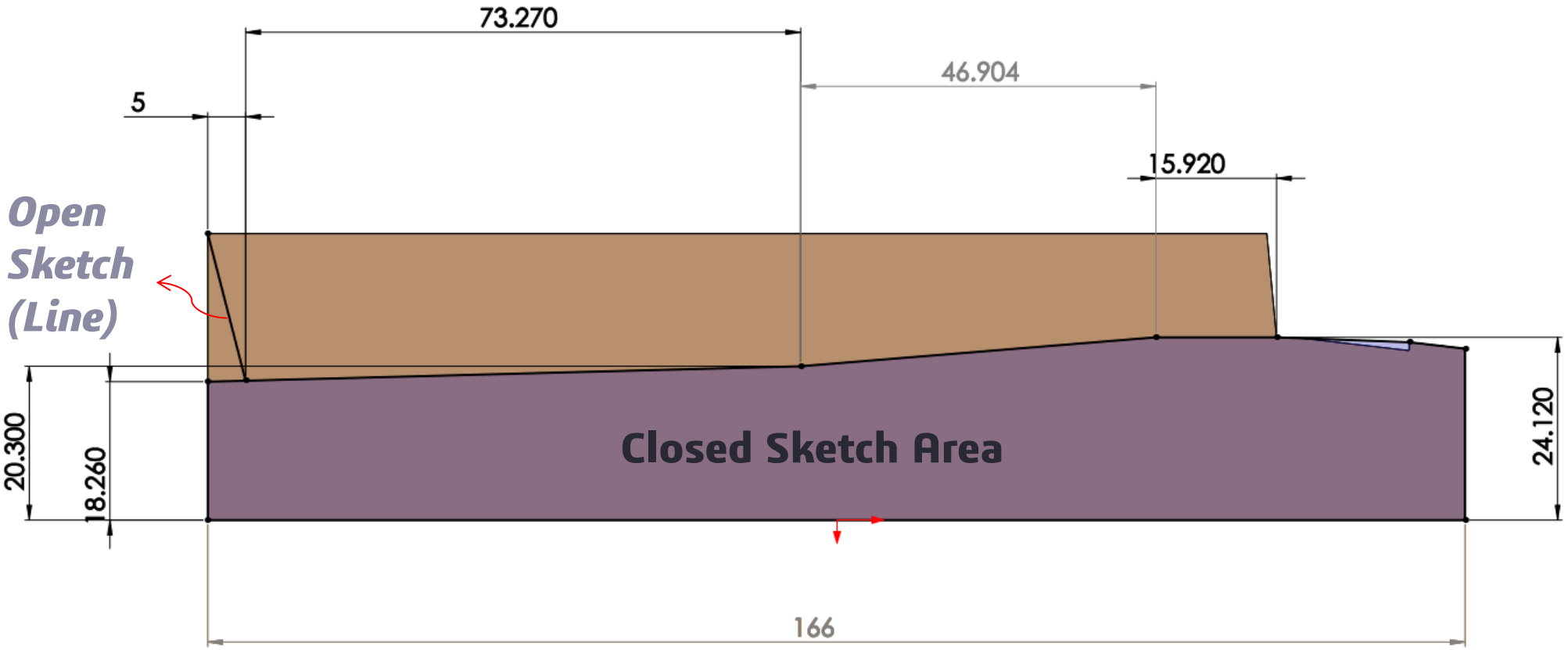
The two green lines are parallel.







Use "Cut-Extrude", set it at *Through-All* and cut it through the part.



Draw this sketch on the side surface of the part and make its corners ***Coincident*** with the edges of the part. Set its dimensions once done.

Cut-Extrude2

From

Sketch Plane

Direction 1

Blind

18.20mm

Flip side to cut

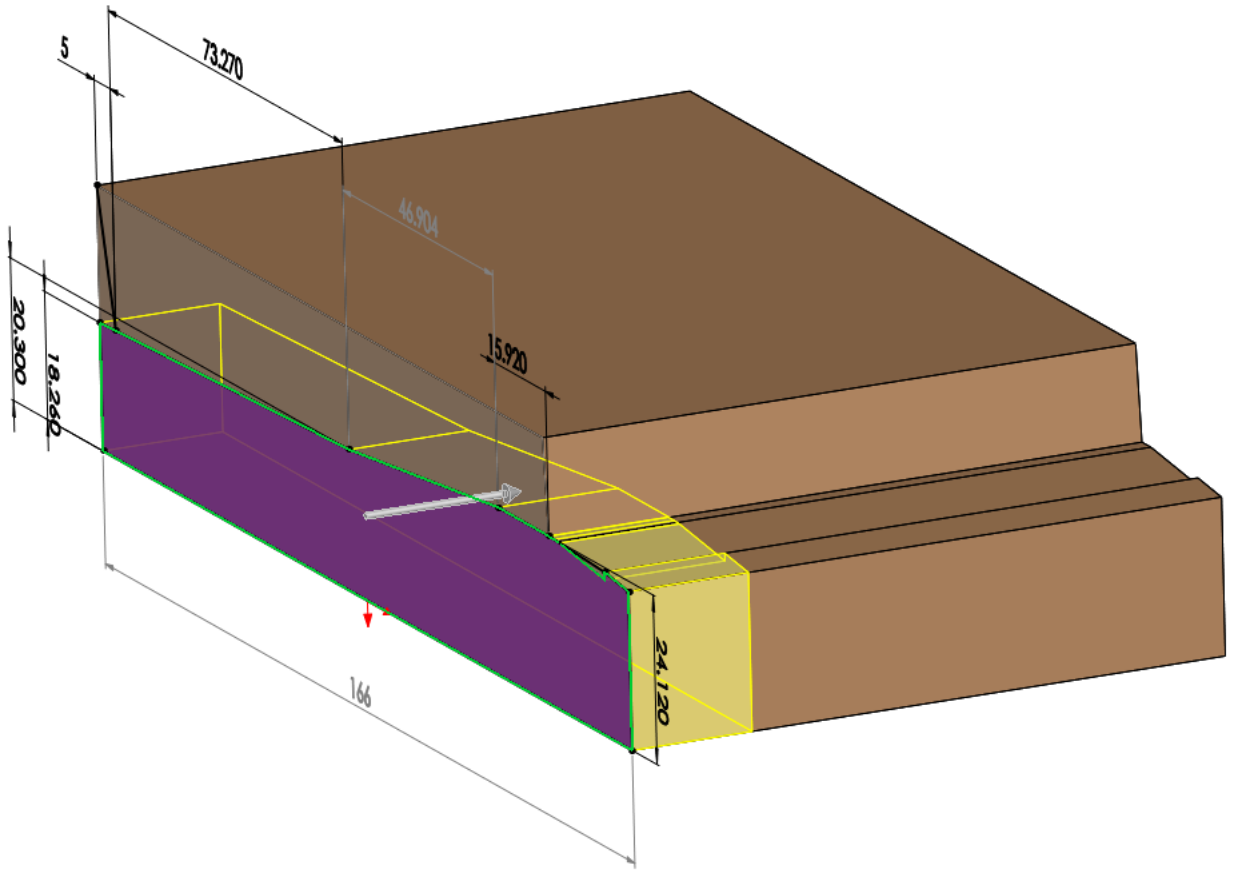
Draft outward

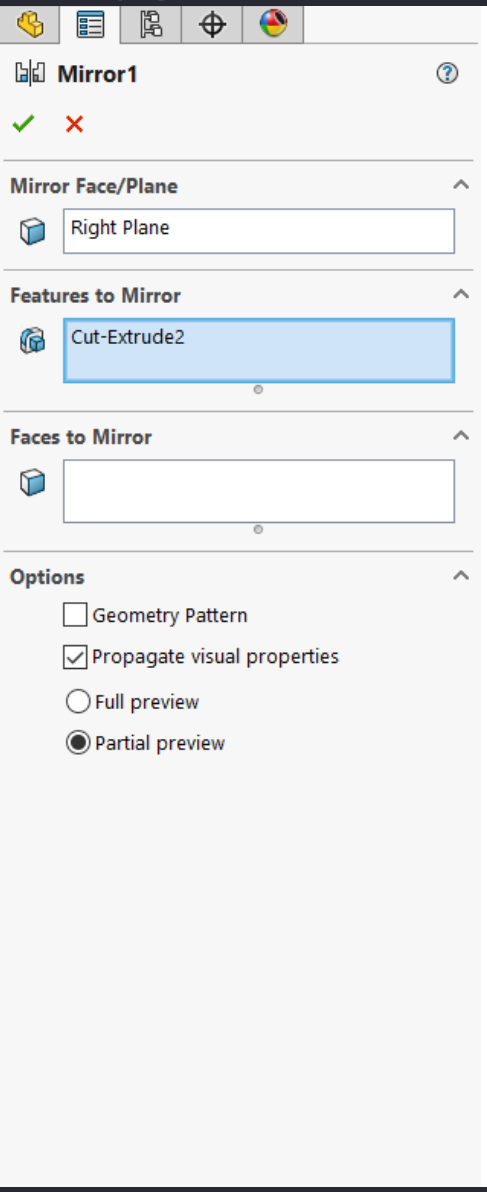
Direction 2

Selected Contours

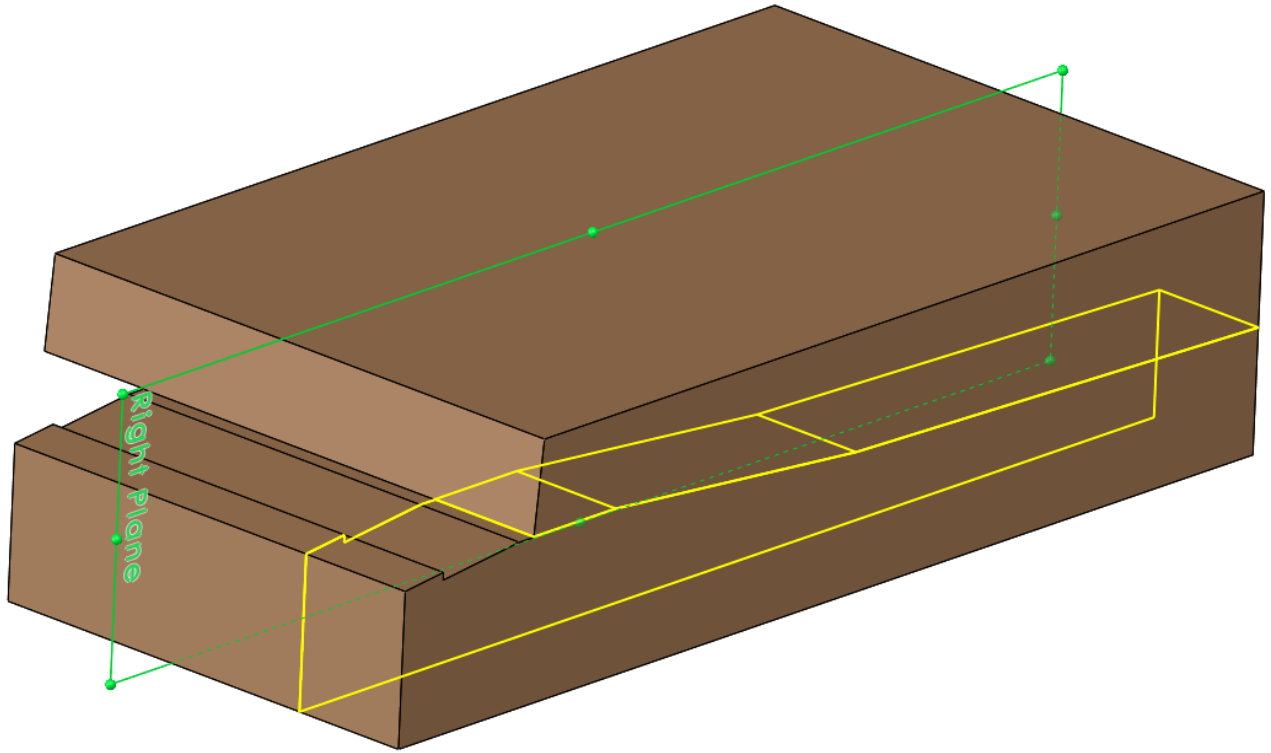
Sketch3-Region<1>

Use “Cut-Extrude”, set it to *Blind* at **18.20** mm and cut it into the part.

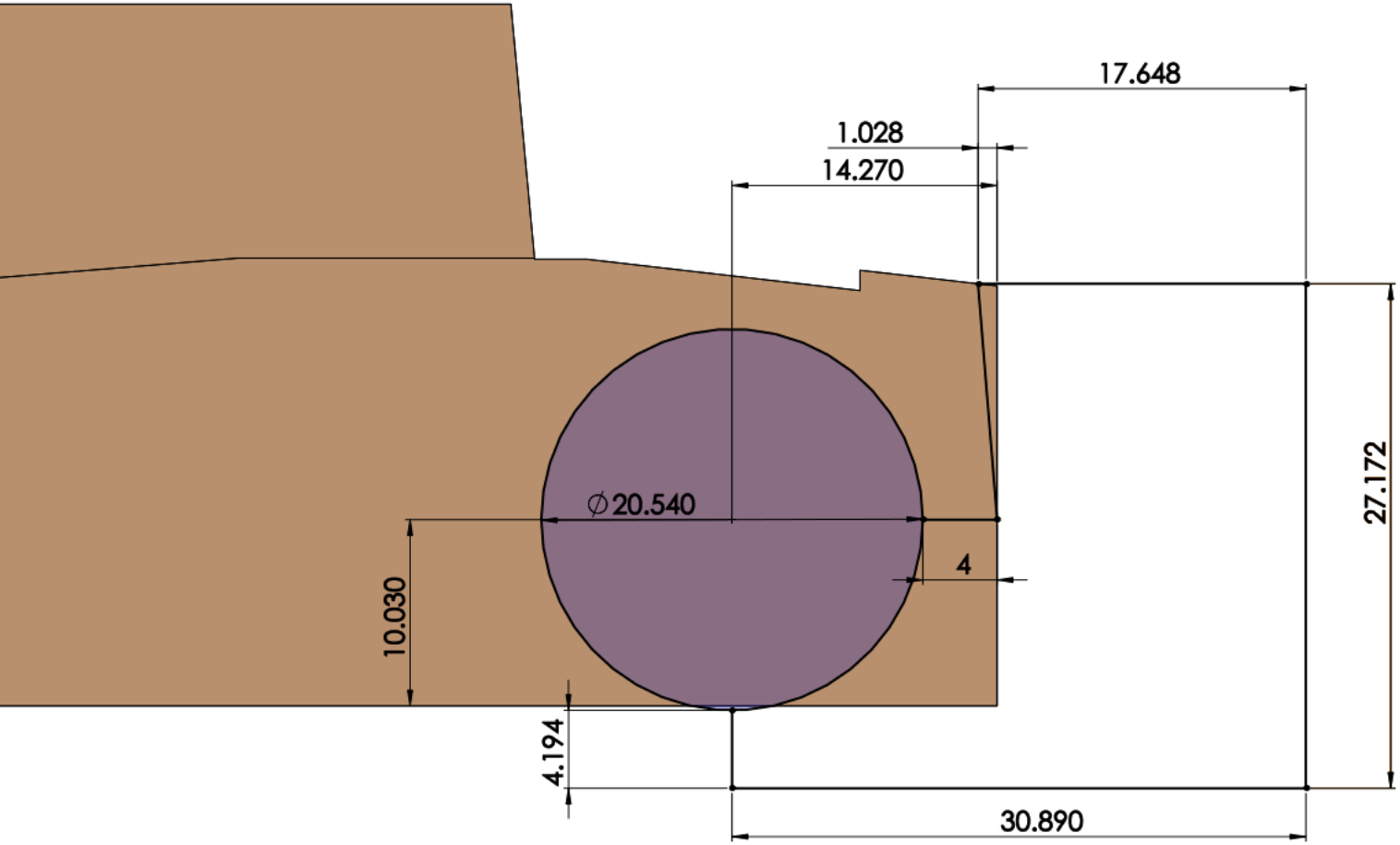




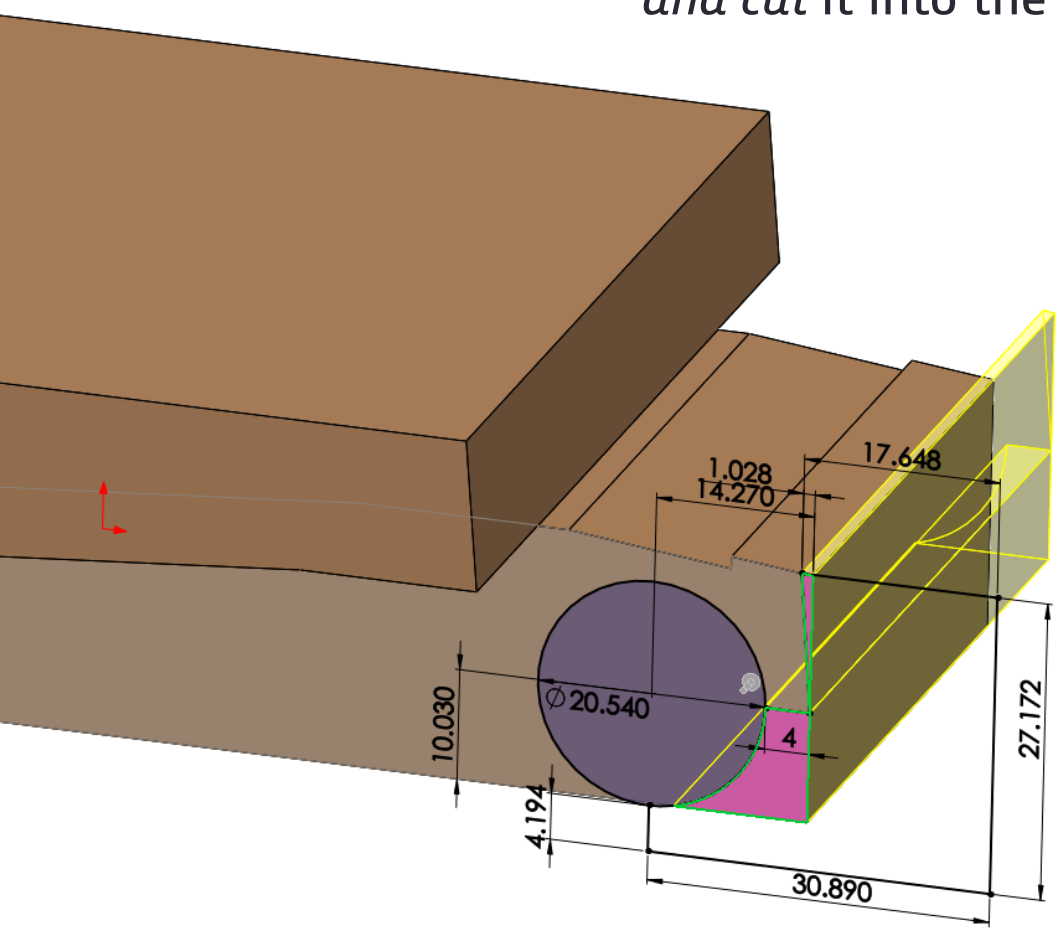
Use “Mirror”, select ***Right Plane*** for the Mirror Face/Plane and duplicate the cut from the last step.

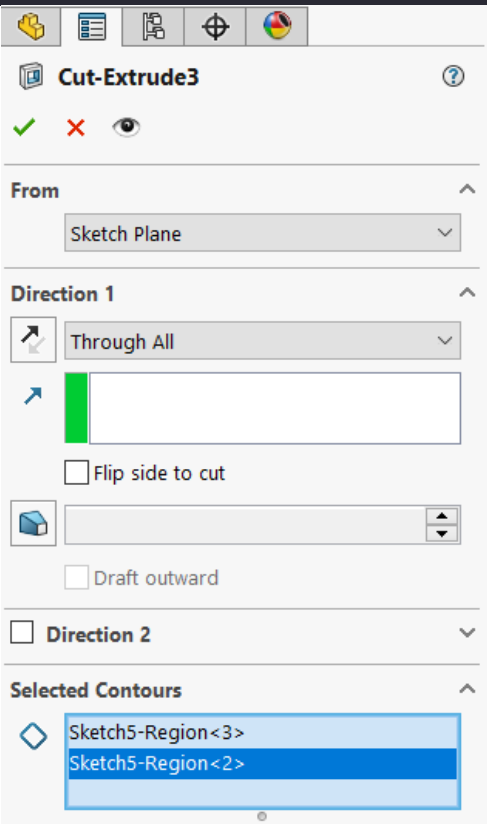


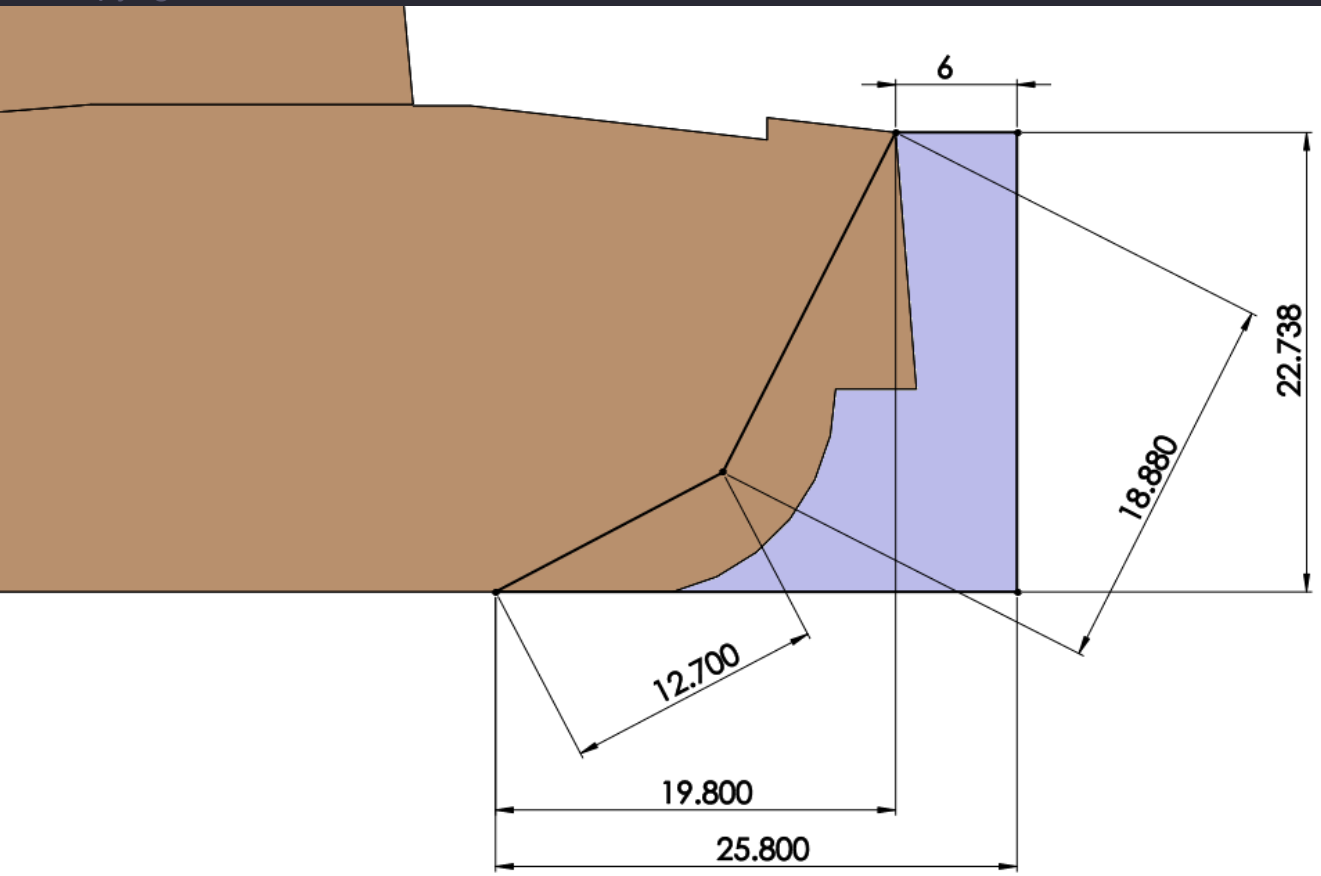
Draw the sketch on the surface of the part and set its dimensions.



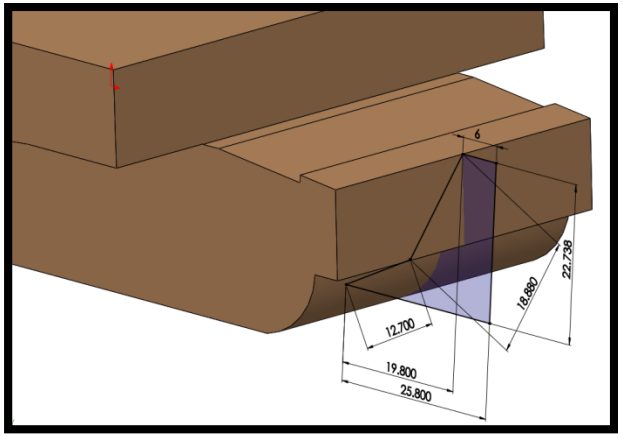
Use "Cut-Extrude", set it to *Through All* *select the two Pink highlighted areas* *and cut* it into the part.

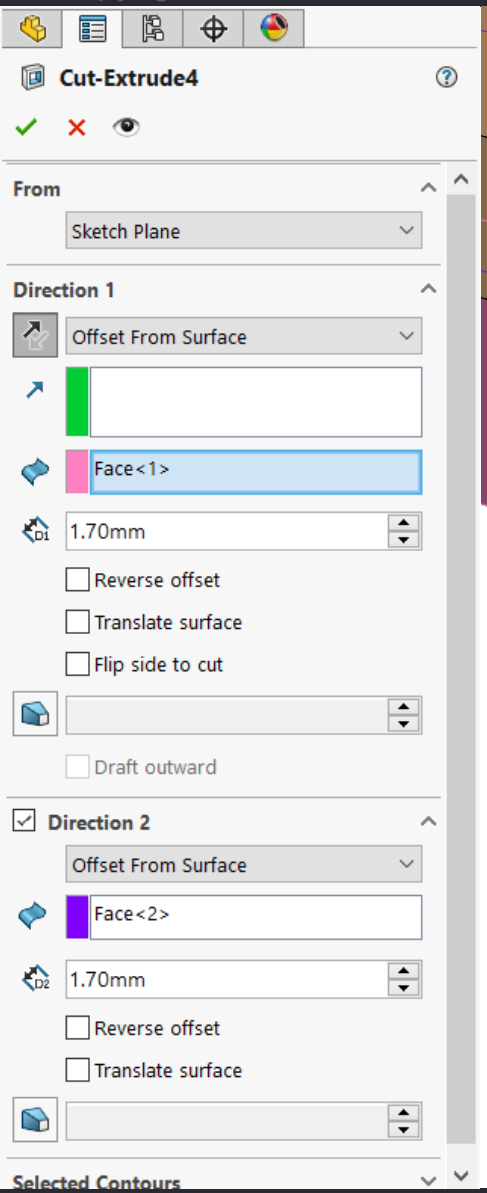


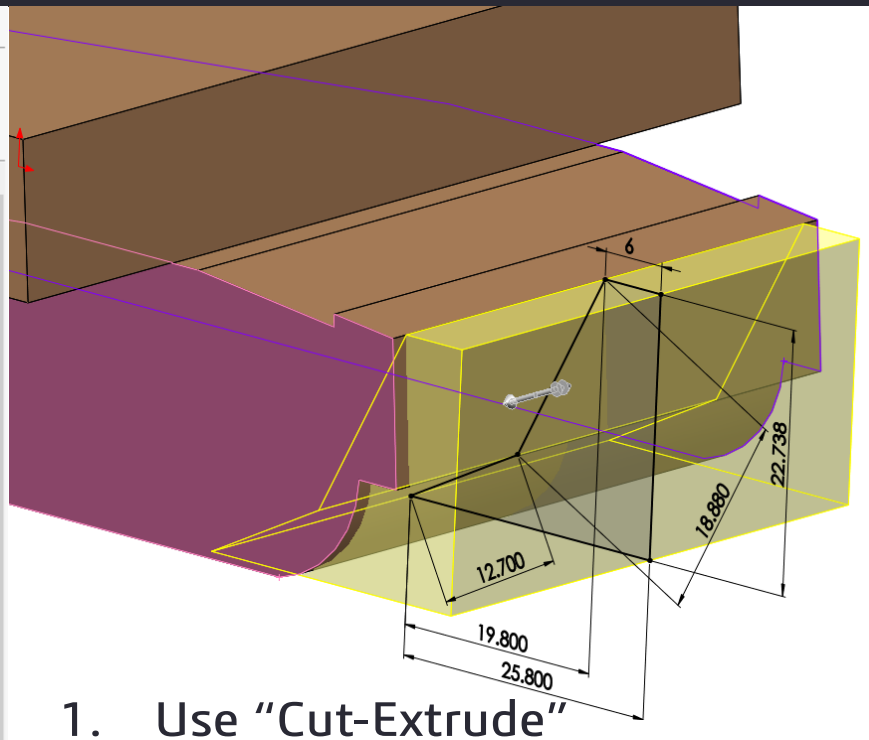


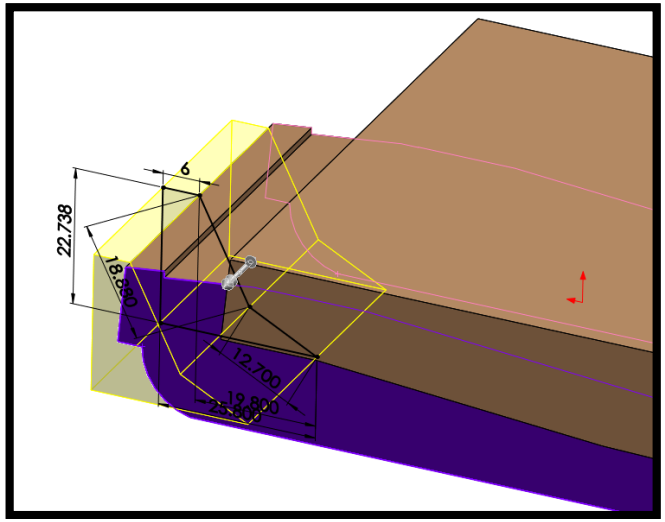


Draw the sketch on the ***Right Plane*** and set its dimensions.

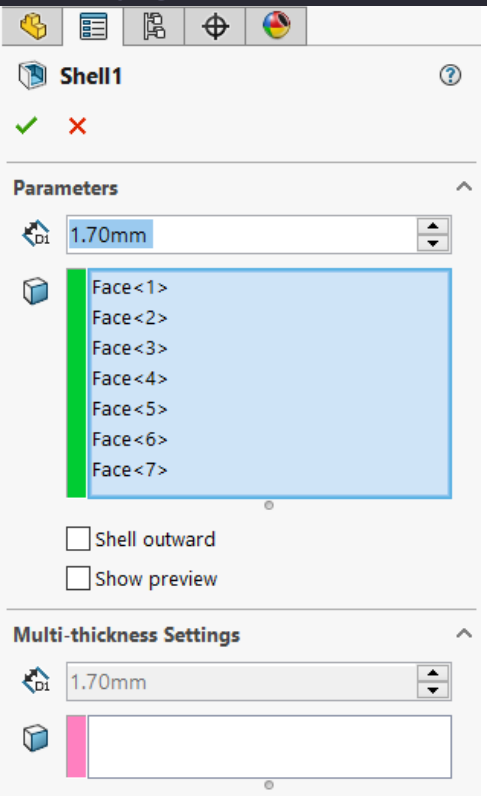




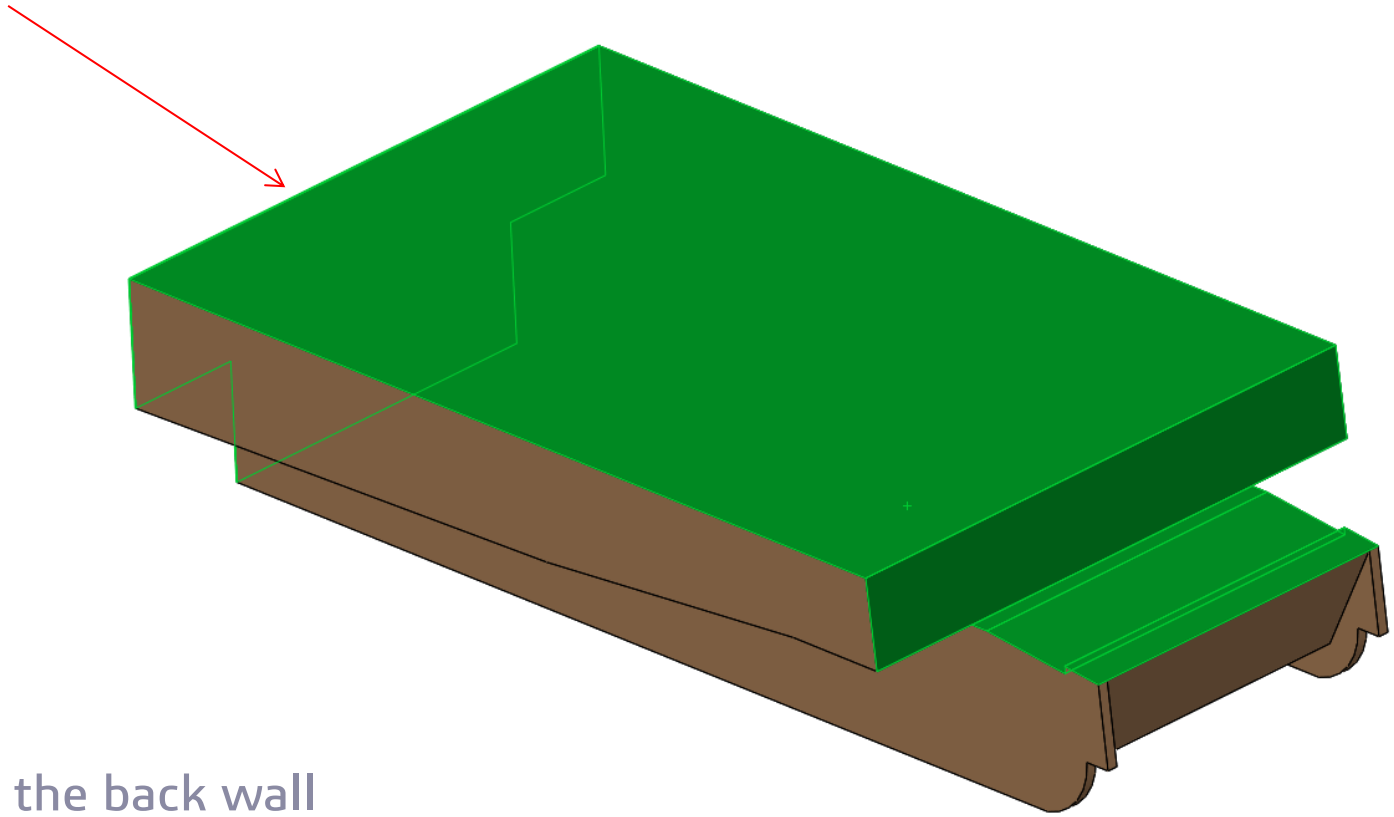




1. Use "Cut-Extrude"
2. Set *Direction 1* to *Offset From Surface*
3. *Select the Pink highlighted surface as the first reference.*
4. *Set the distance to **1.7** mm*
5. *Set Direction 2* to *Offset From Surface*
6. *Select the purple surface as the second reference*
7. *Set the distance to **1.7** mm*
8. *Click OK*

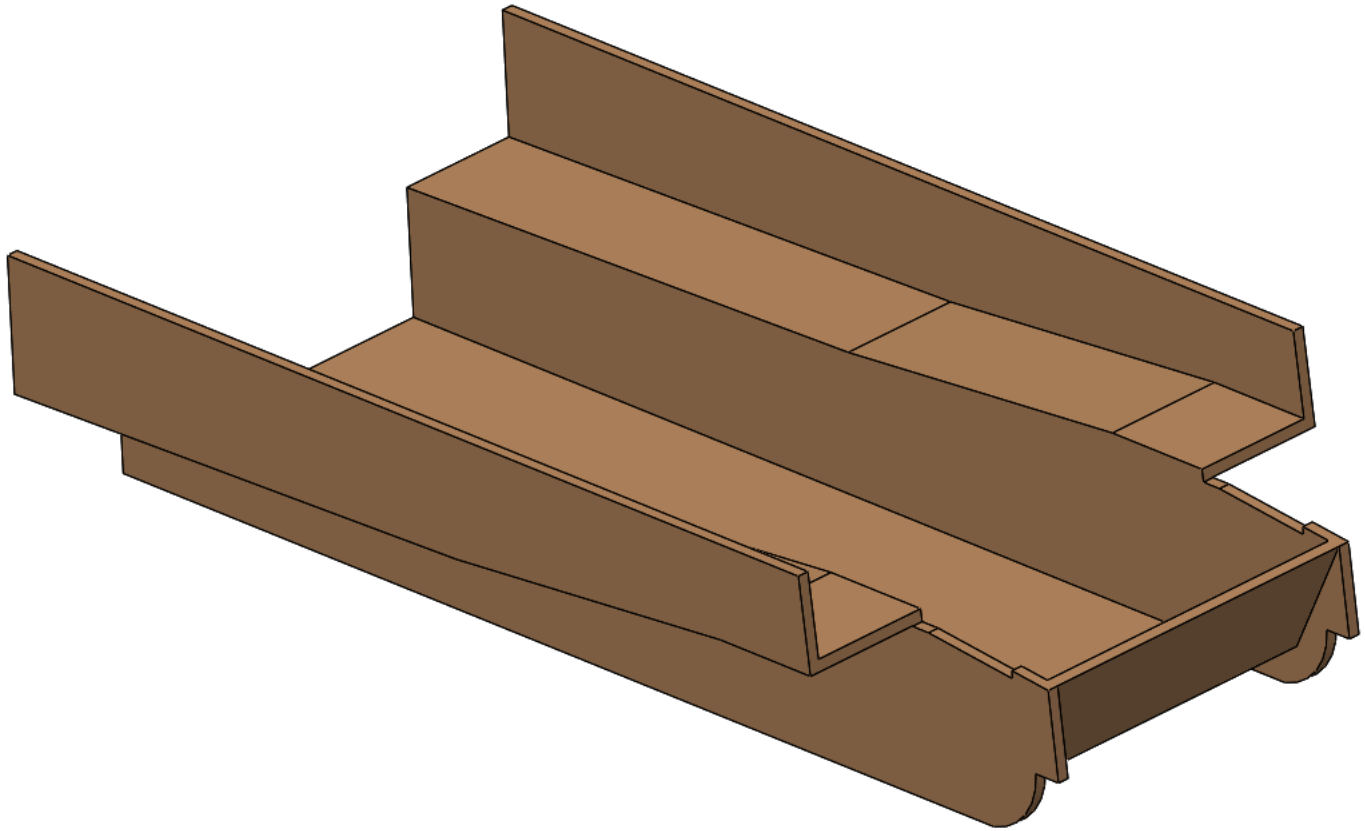


Use “Shell” set it at **1.7** mm, select the 7* green highlighted surfaces and make the part shallow.

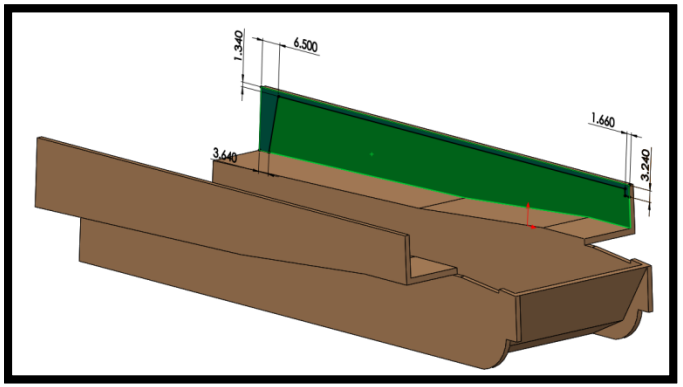


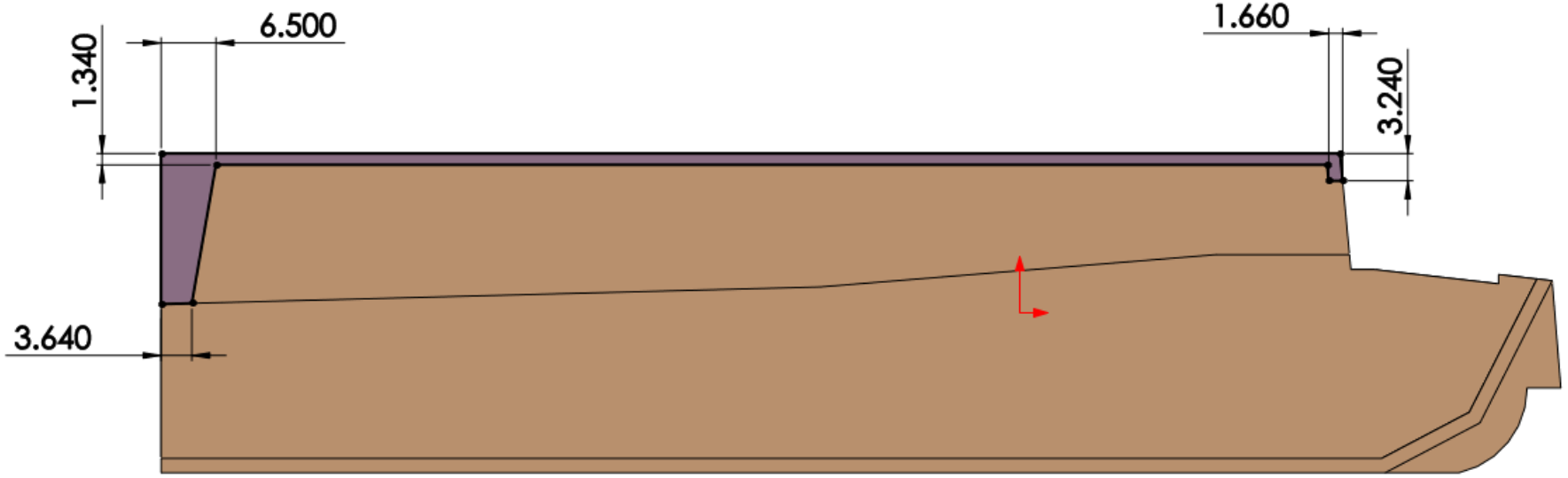
*One is the back wall

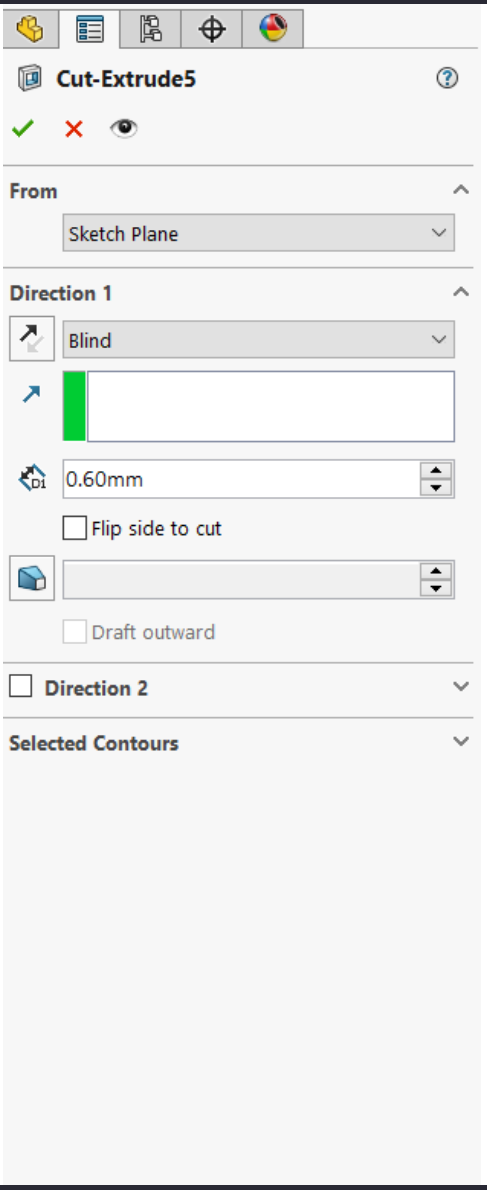
Your Part so far...



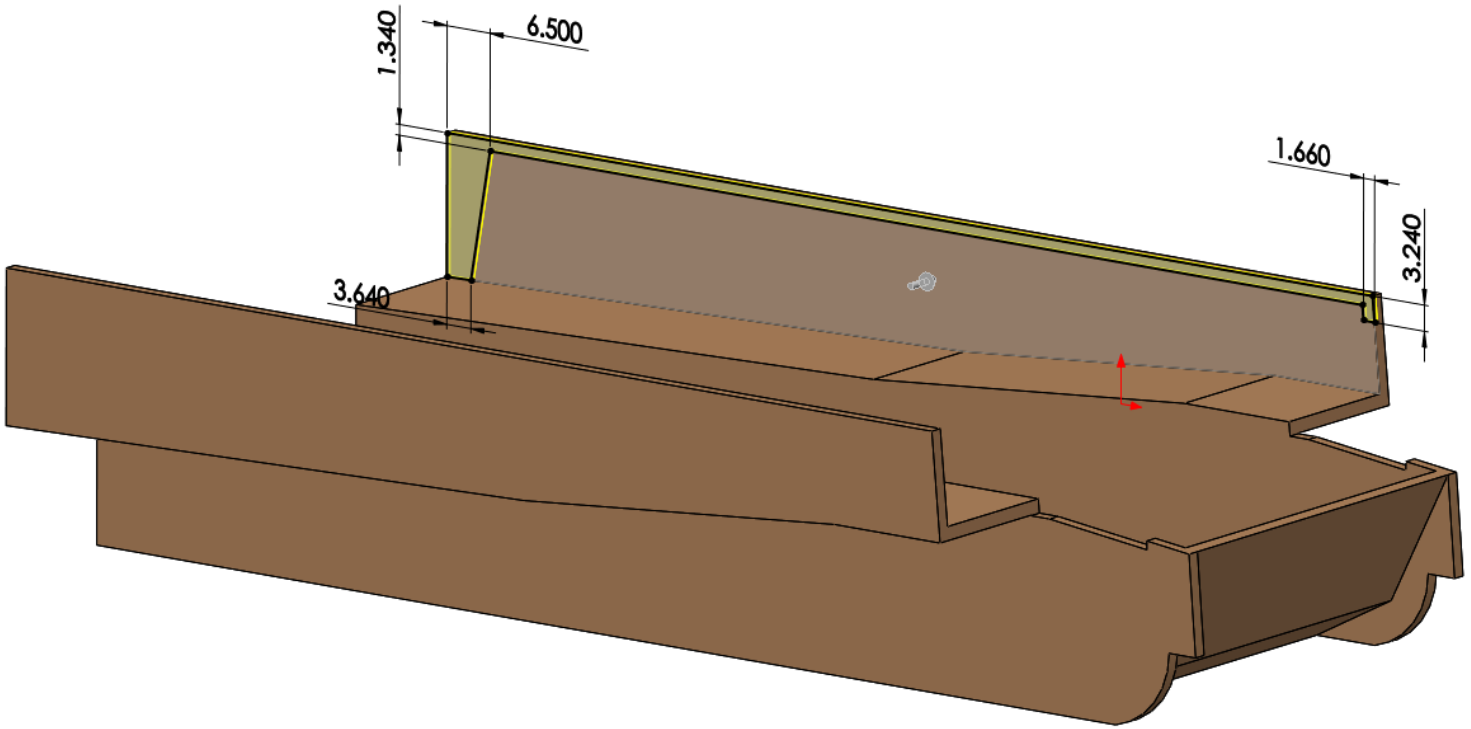
Draw the sketch on the green highlighted surface and set its dimensions.

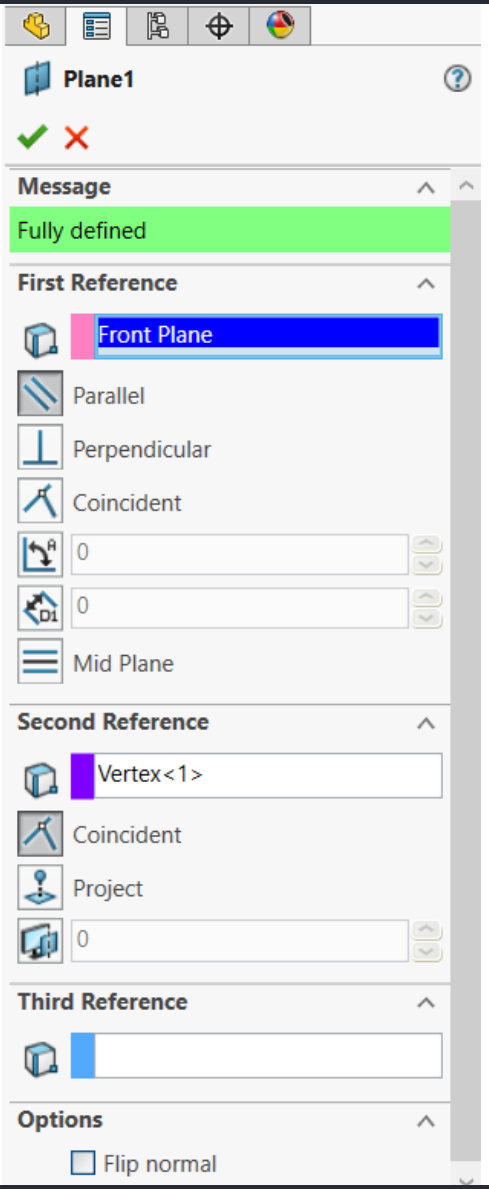




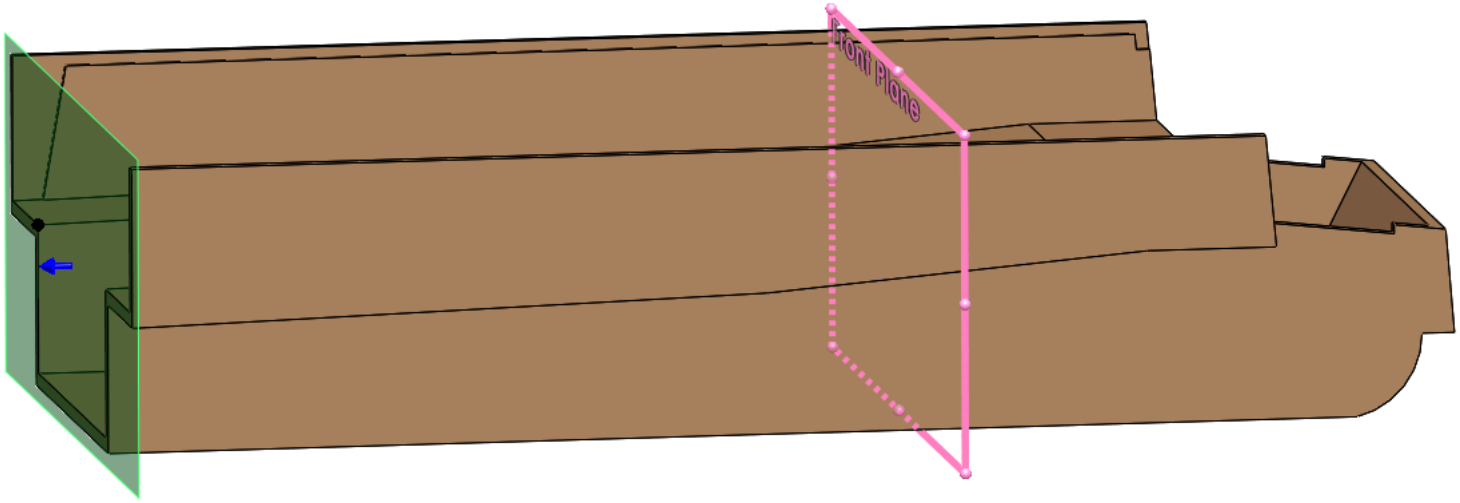


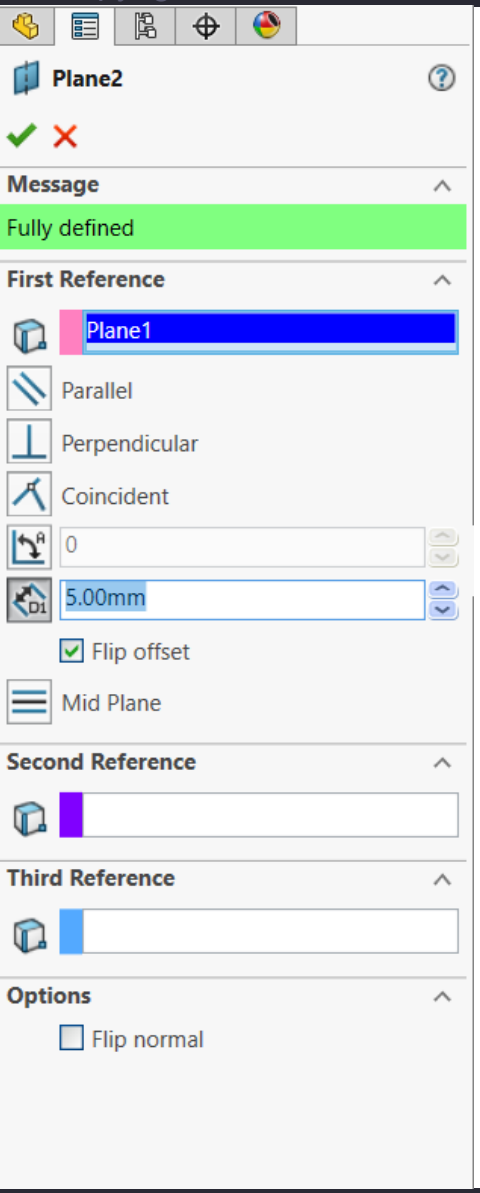
Use "Cut-Extrude", set it to *Blind* and cut it **0.6** mm into the part.



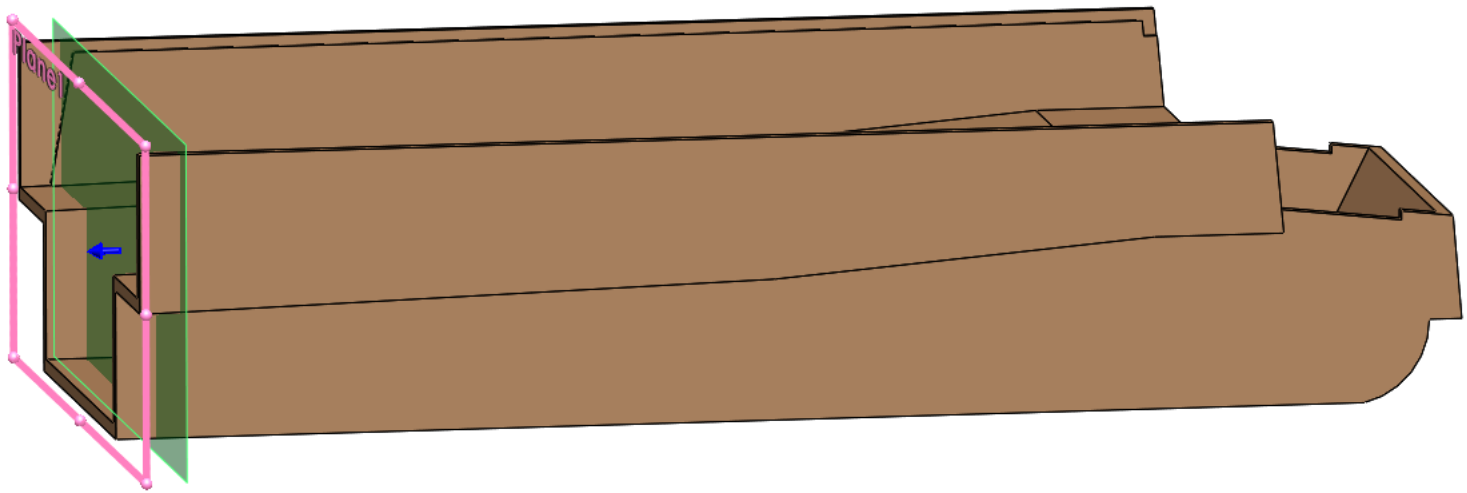


1. Go to Insert>Reference Geometry> Plane.
2. Select the Front Plane
3. Choose Parallel as the 1st reference
4. Choose a vertex or point at the end of the part as the 2nd
5. Click Ok

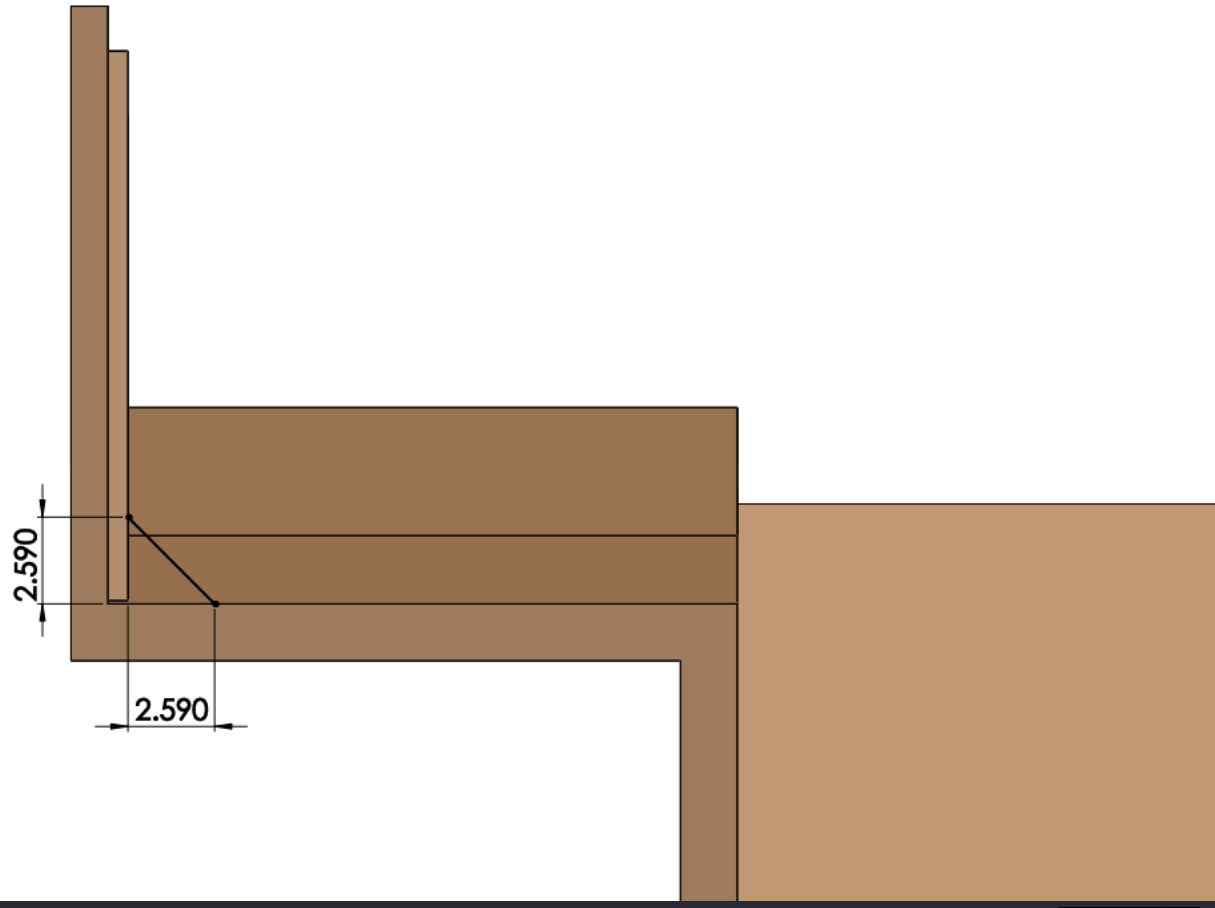




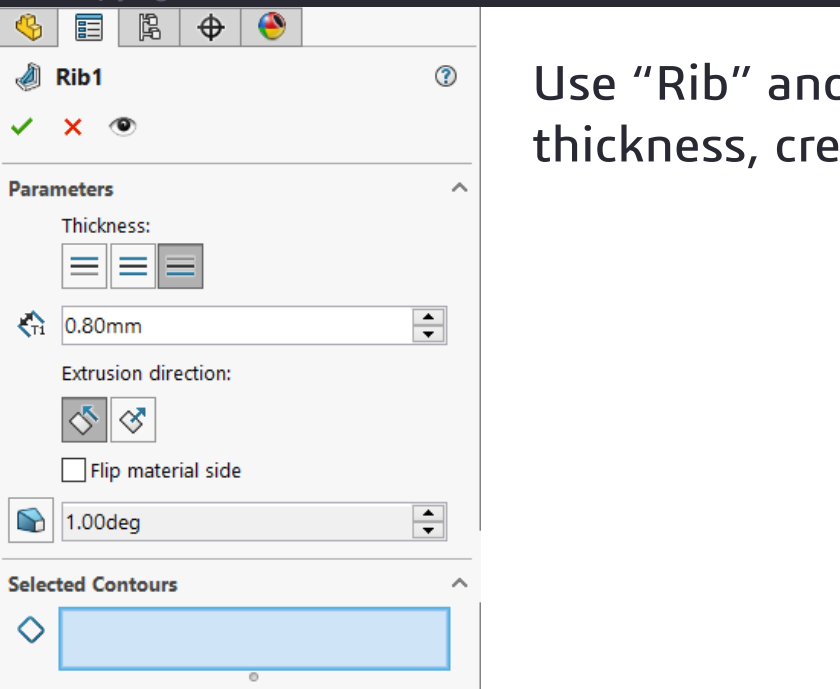
1. Go to Insert>Reference Geometry> Plane.
2. Select the recently created plane from the last step
3. Set the distance to 5 mm
4. Activate “Flip Offset” if neccessary
5. Click Ok

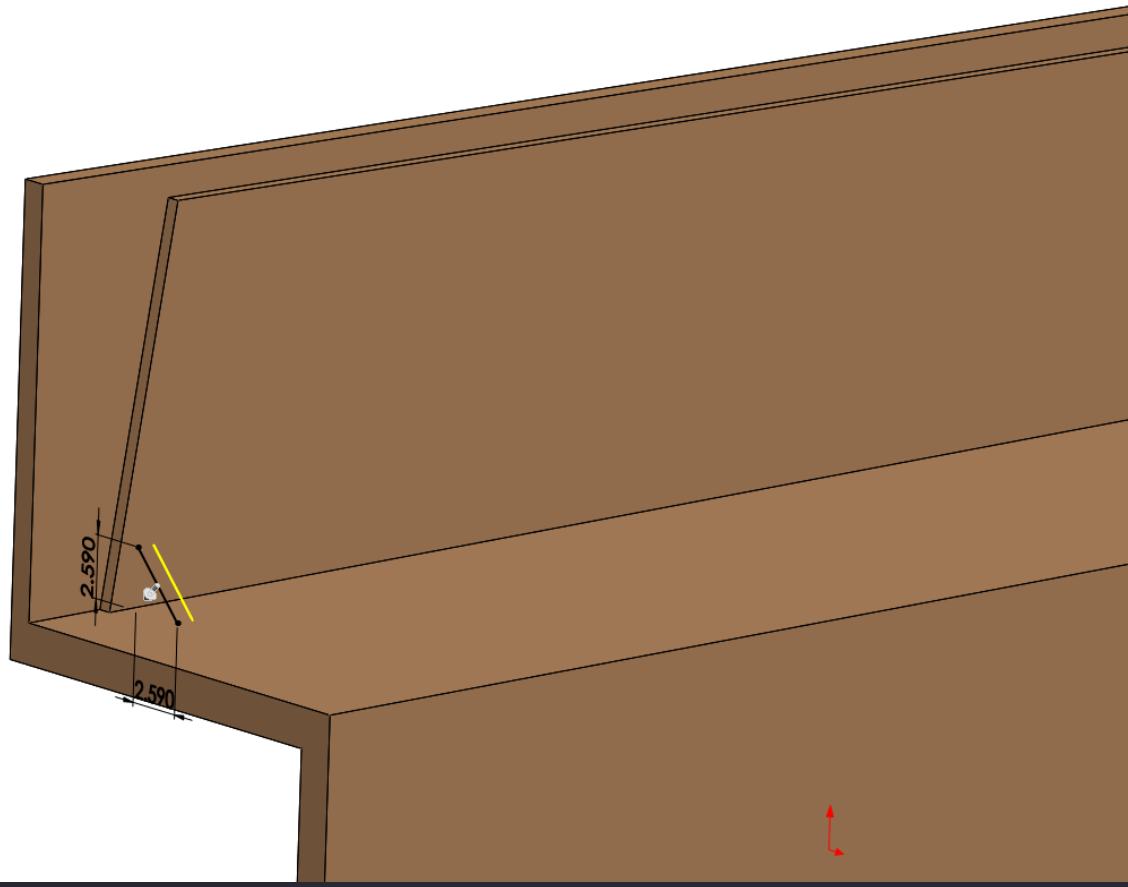


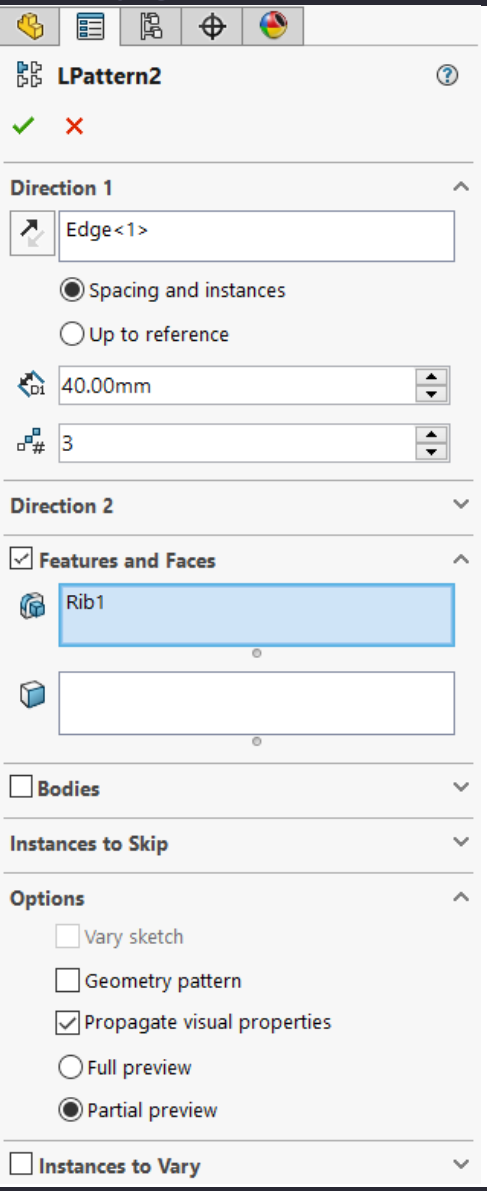
Draw a line on the last plane you created and set its dimensions like the figure below. The two ends are connected to the surfaces of the part.



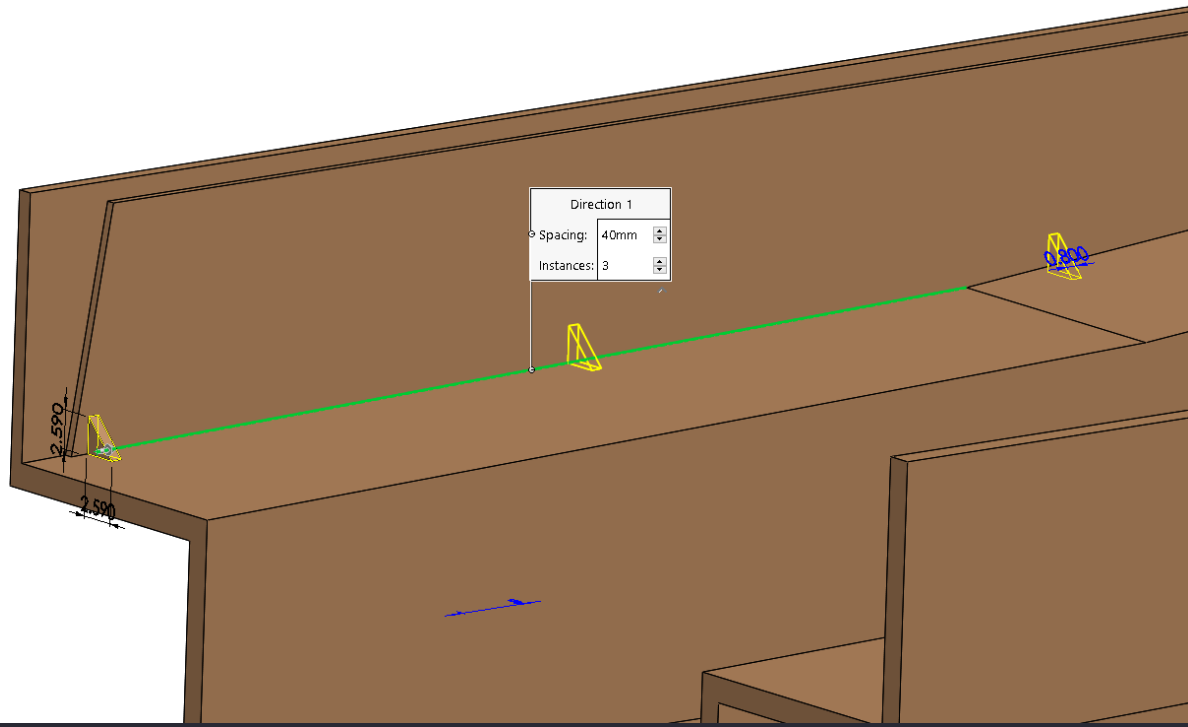
Use “Rib” and using a **0.8** mm one-sided thickness, create rib.





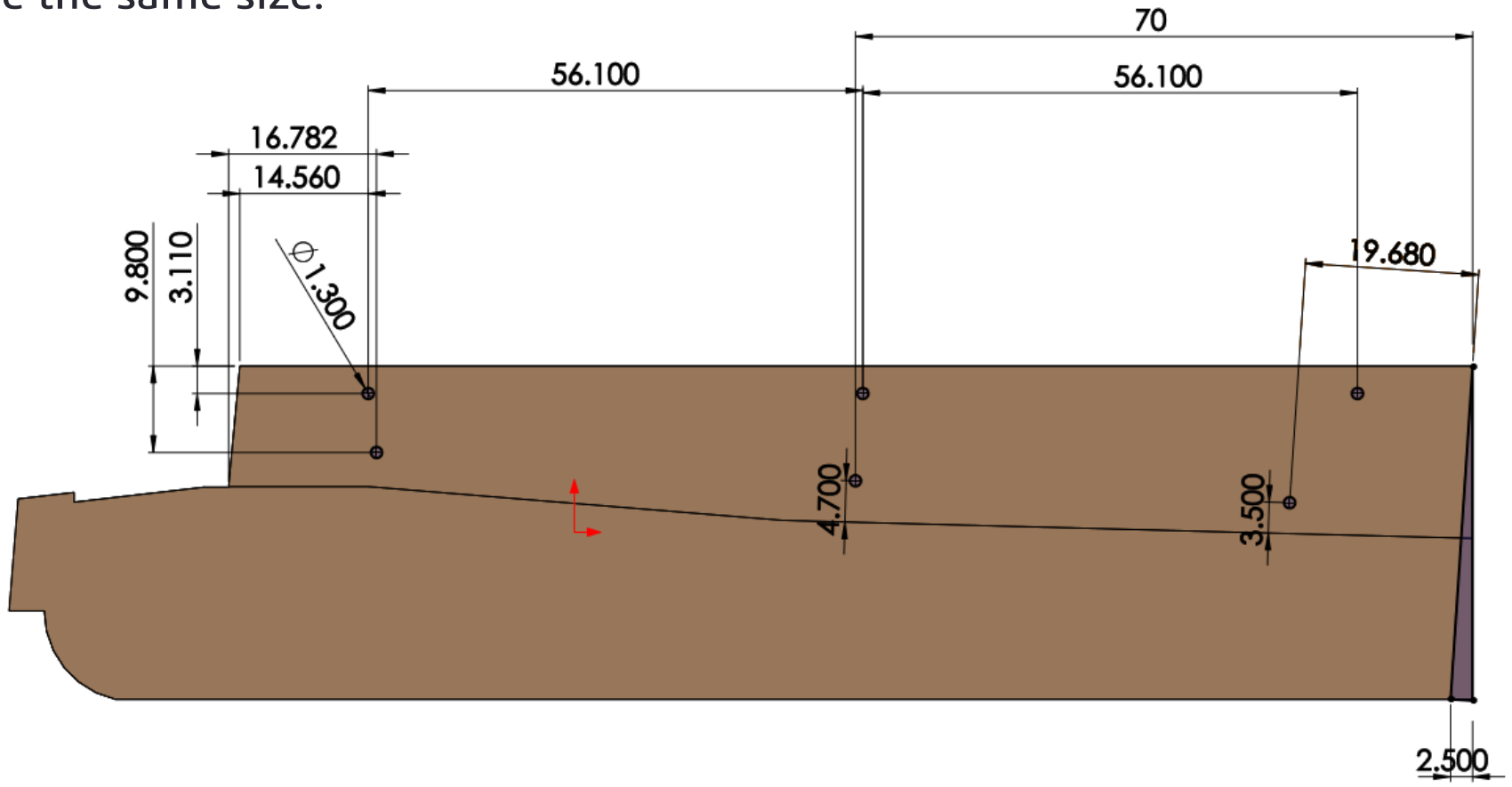


1. Use “Linear Pattern”
2. Select the green highlighted edge as *Direction 1*
3. Set the distance to **40** mm
4. Set the number of instances to 3
5. Select the Rib
6. Click Ok

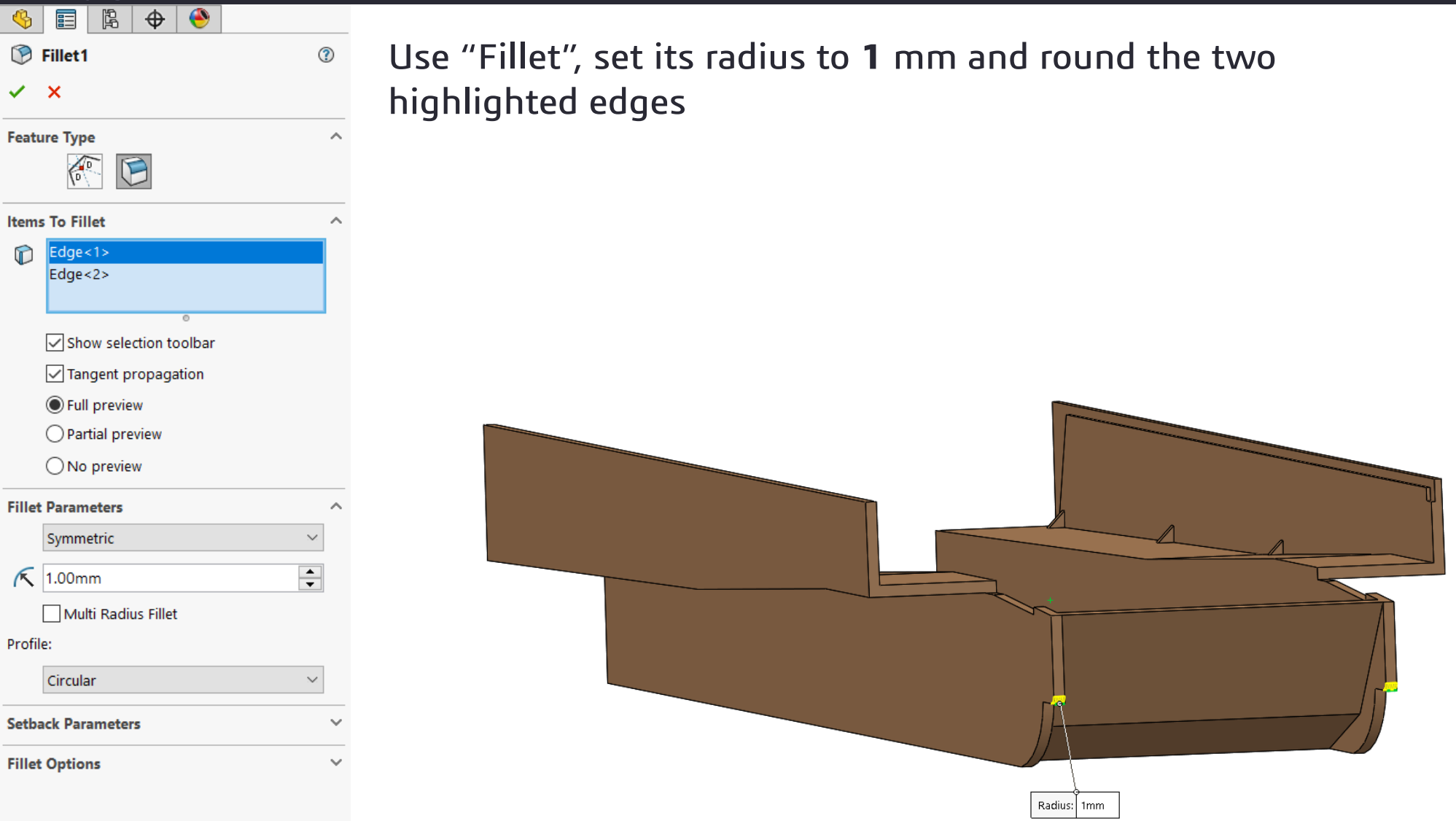


Draw six circles and a rectangle on the surface of the part and set their dimensions.

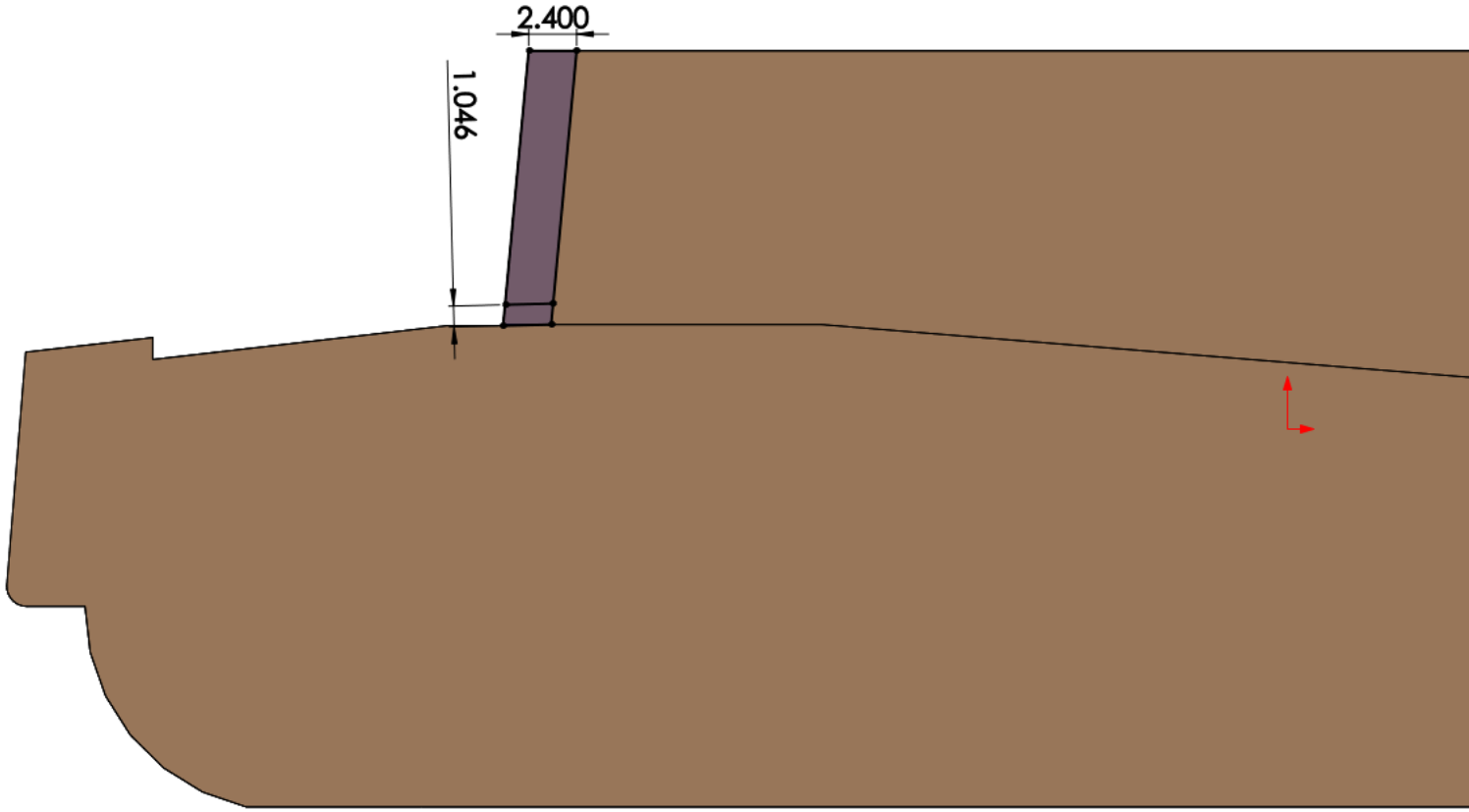
All circles are the same size.

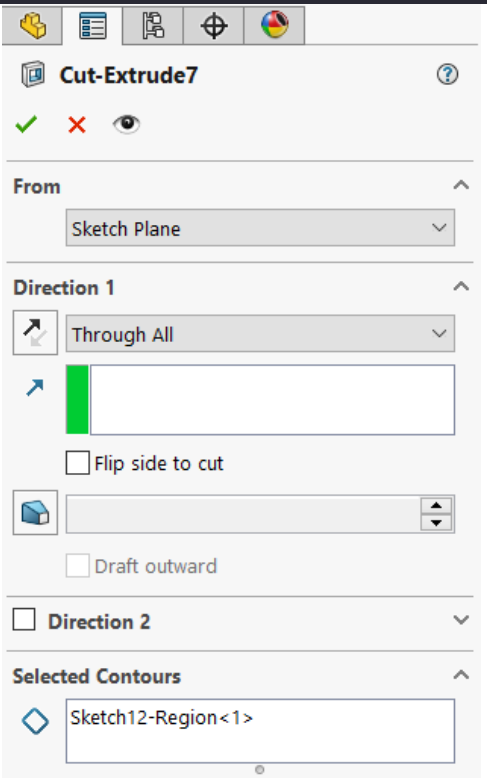


Use “Fillet”, set its radius to **1** mm and round the two highlighted edges

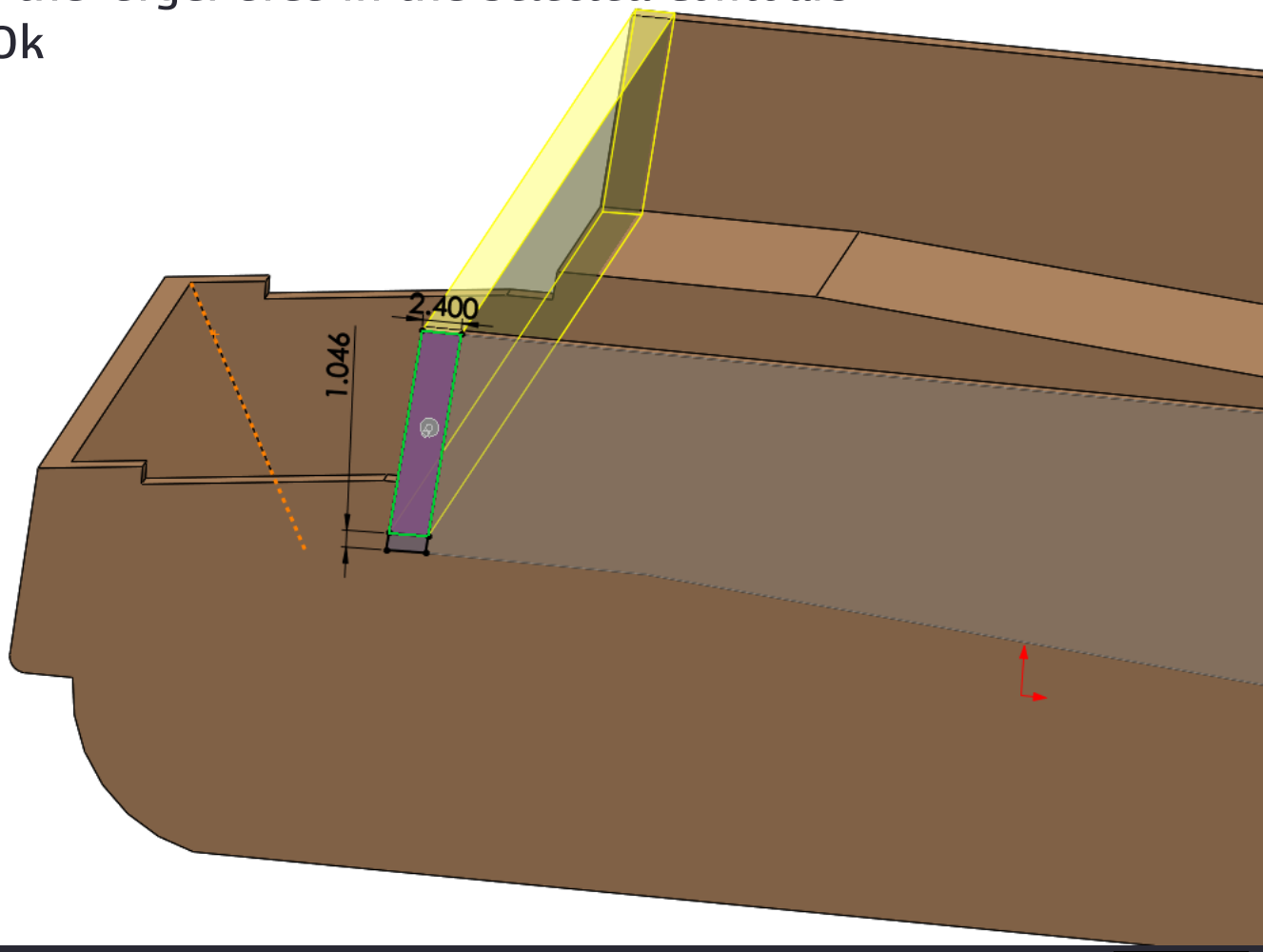


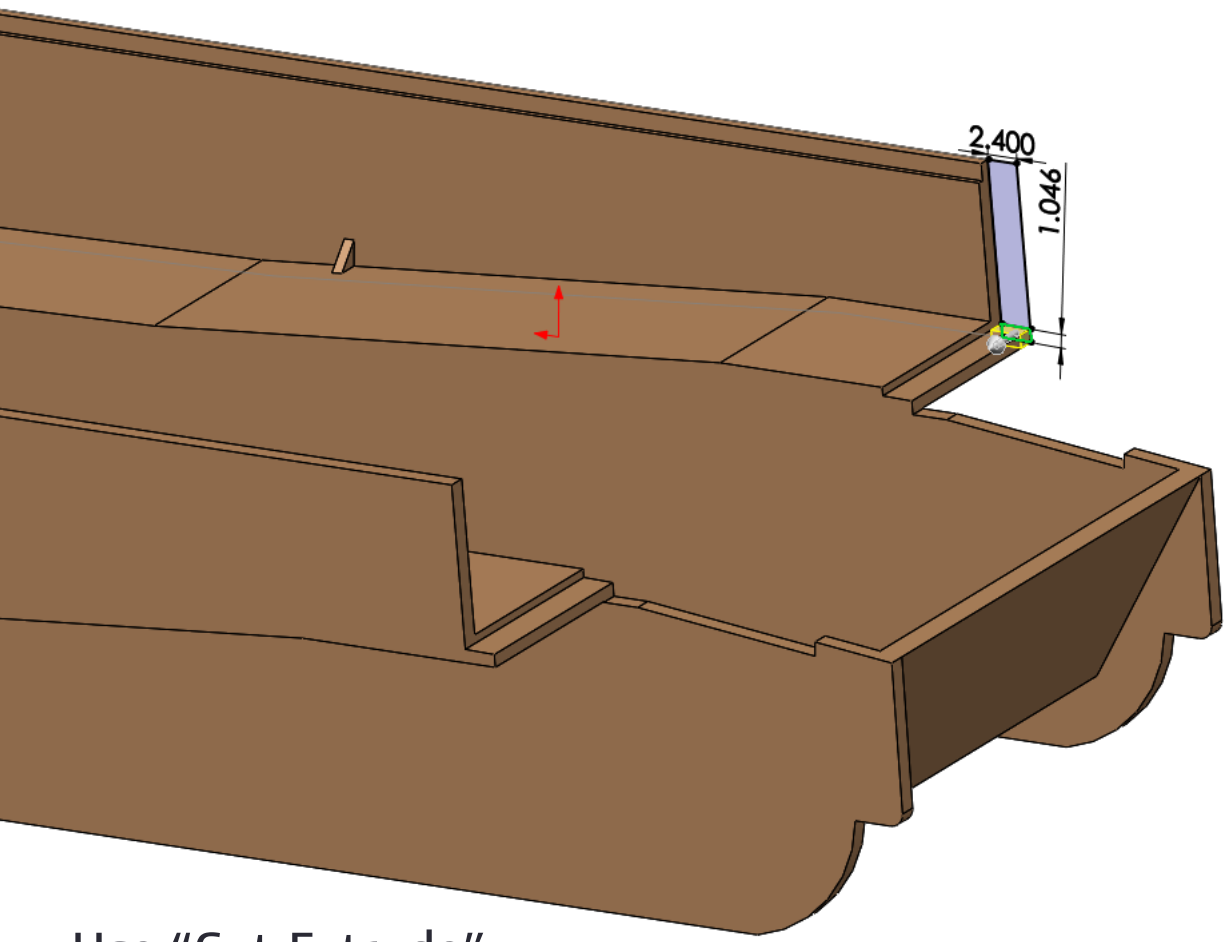
Draw the sketch on the surface of the part and set its dimensions.

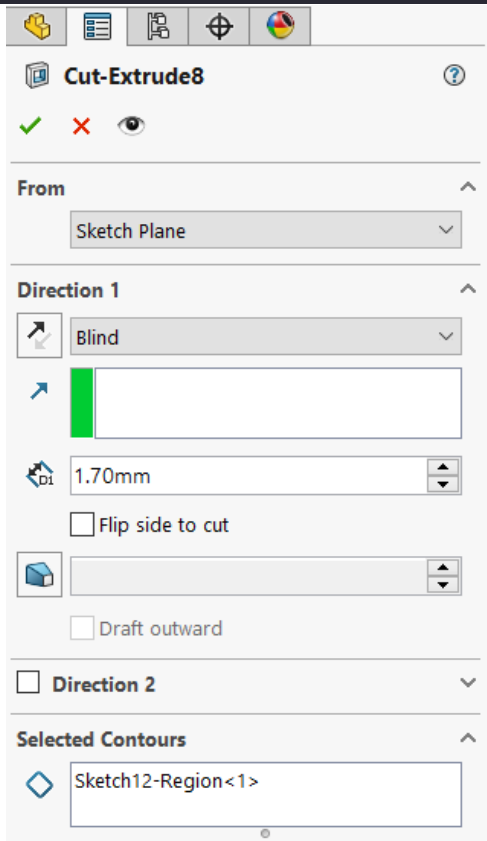




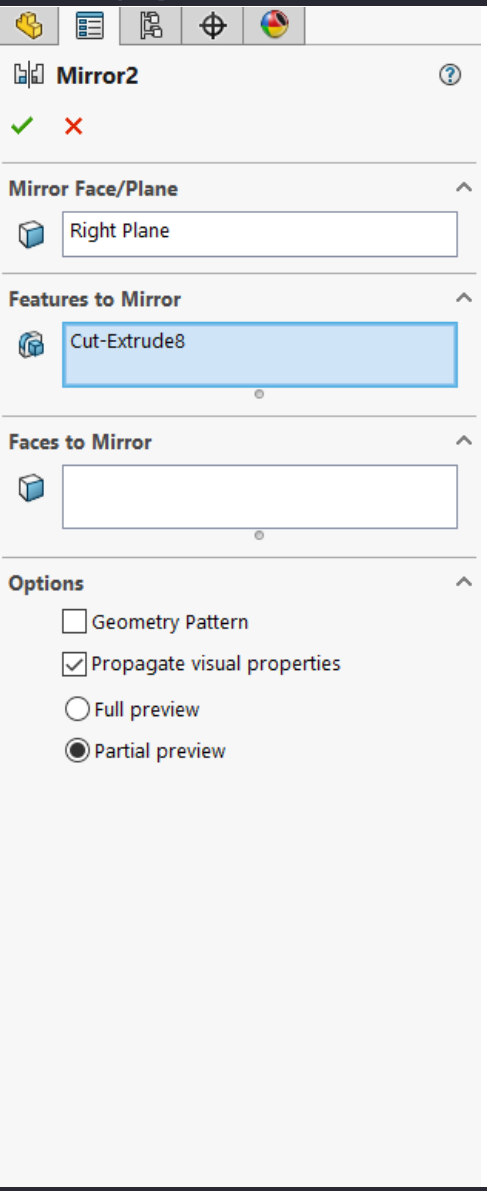
1. Use “Cut-Extrude”,
2. Set it at *Through All*
3. Select the larger area in the *Selected Contours*
4. Click Ok



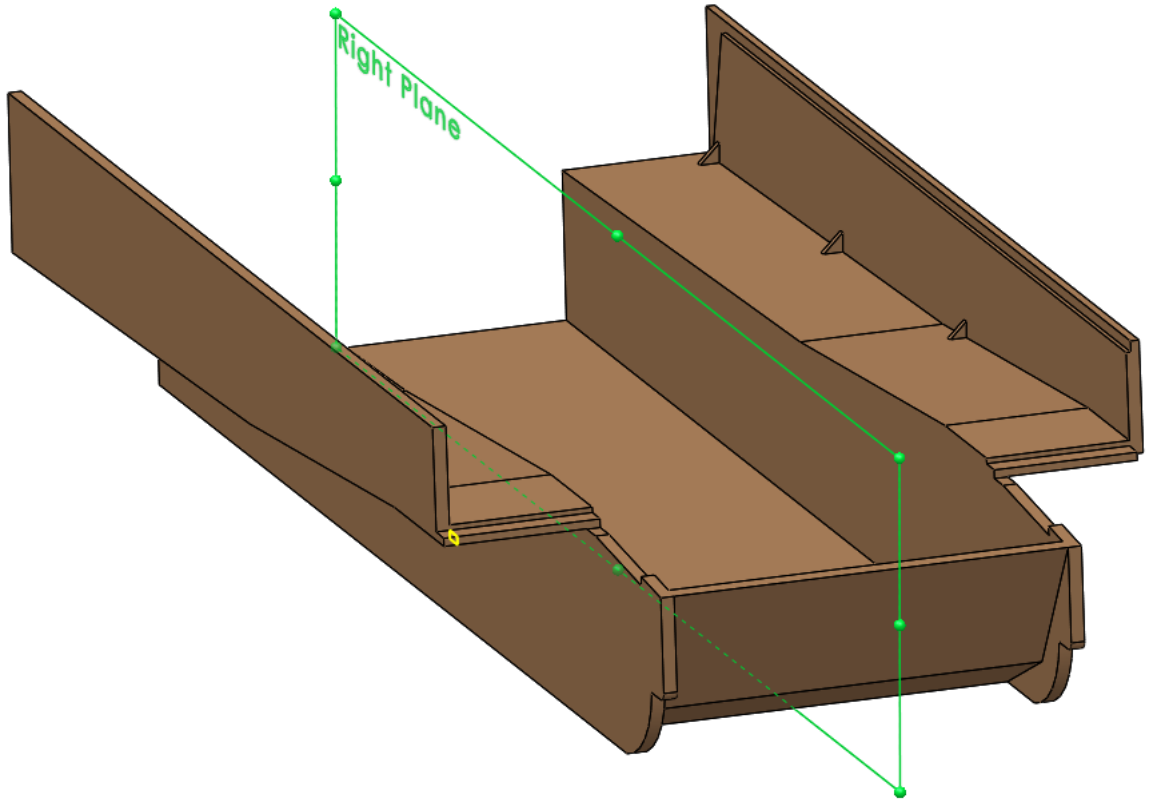




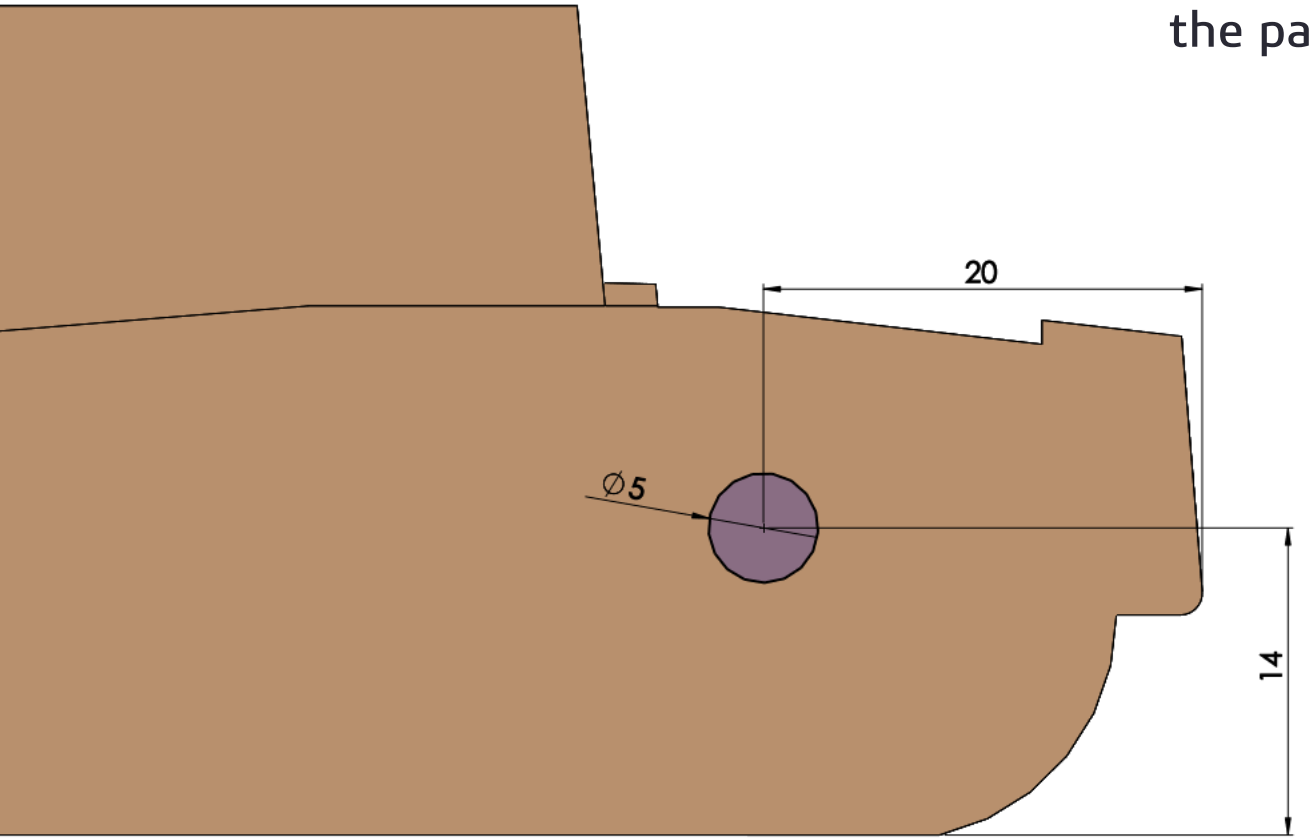
Use "Cut-Extrude",
Set it at *Blind*
Select the smaller area in the Selected Contours
Click Ok

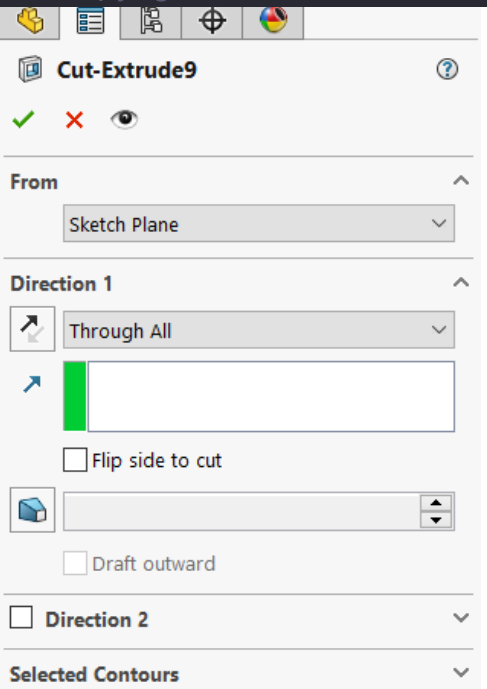


1. Use “Mirror”
2. Select the ***Right Plane*** as the *Mirror Face/Plane*
3. Select the Cut-Extrude from the last step
4. And mirror it to the other side.

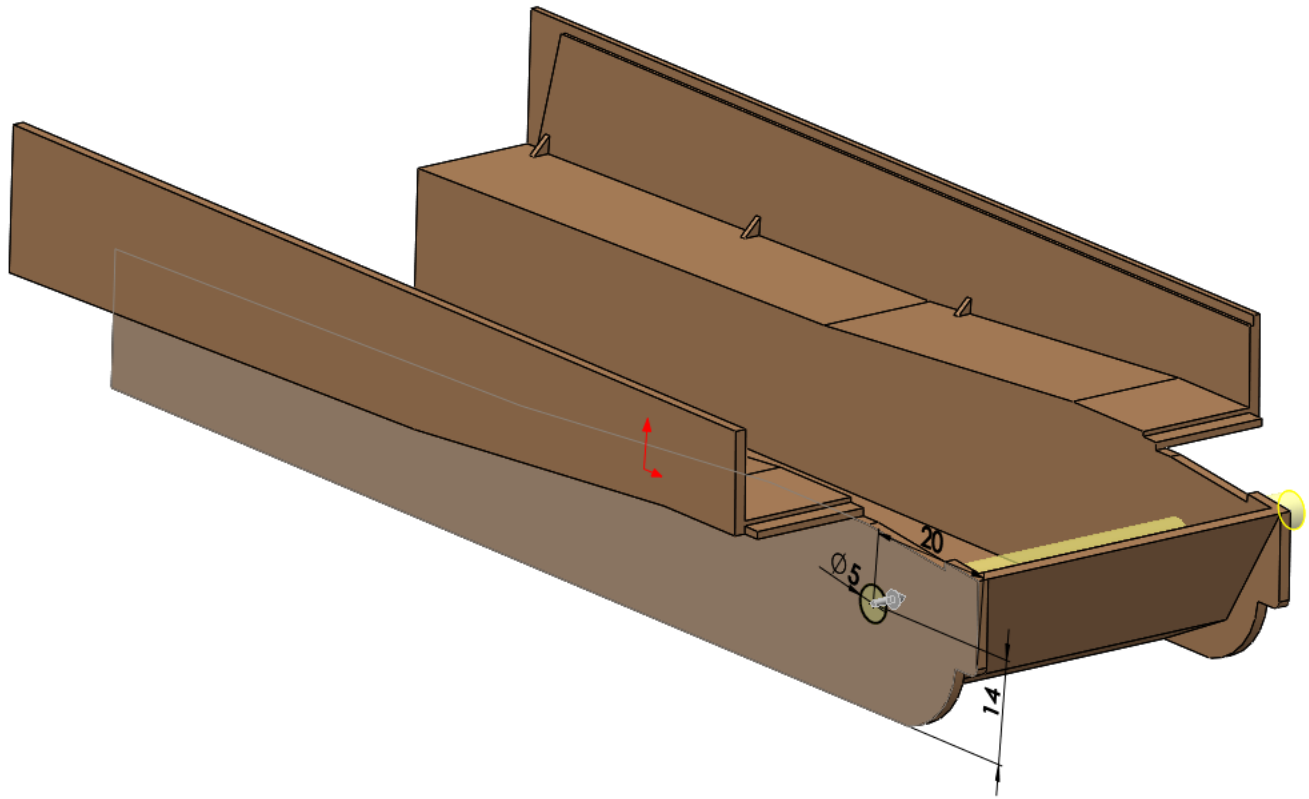


Draw the sketch on the surface of the part and set its dimensions.

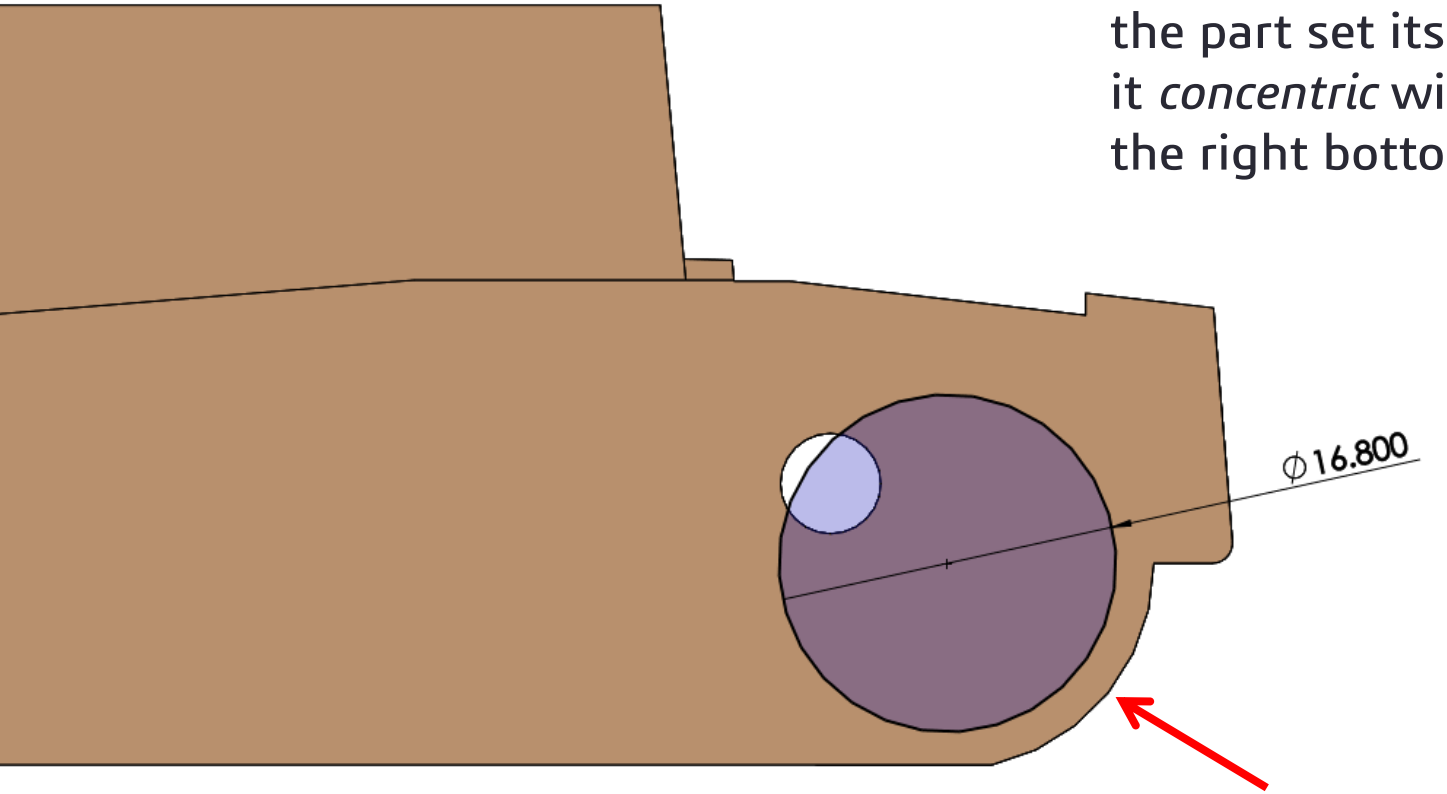


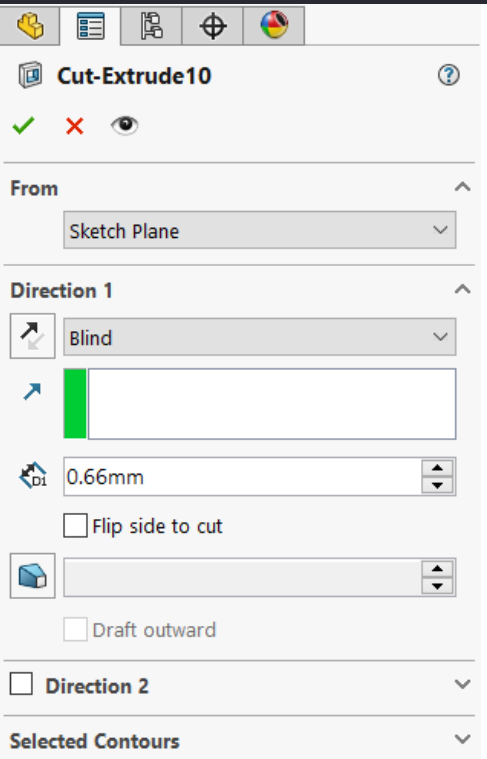


Use "Cut-Extrude" set it to *Through-All* and cut it through the part.

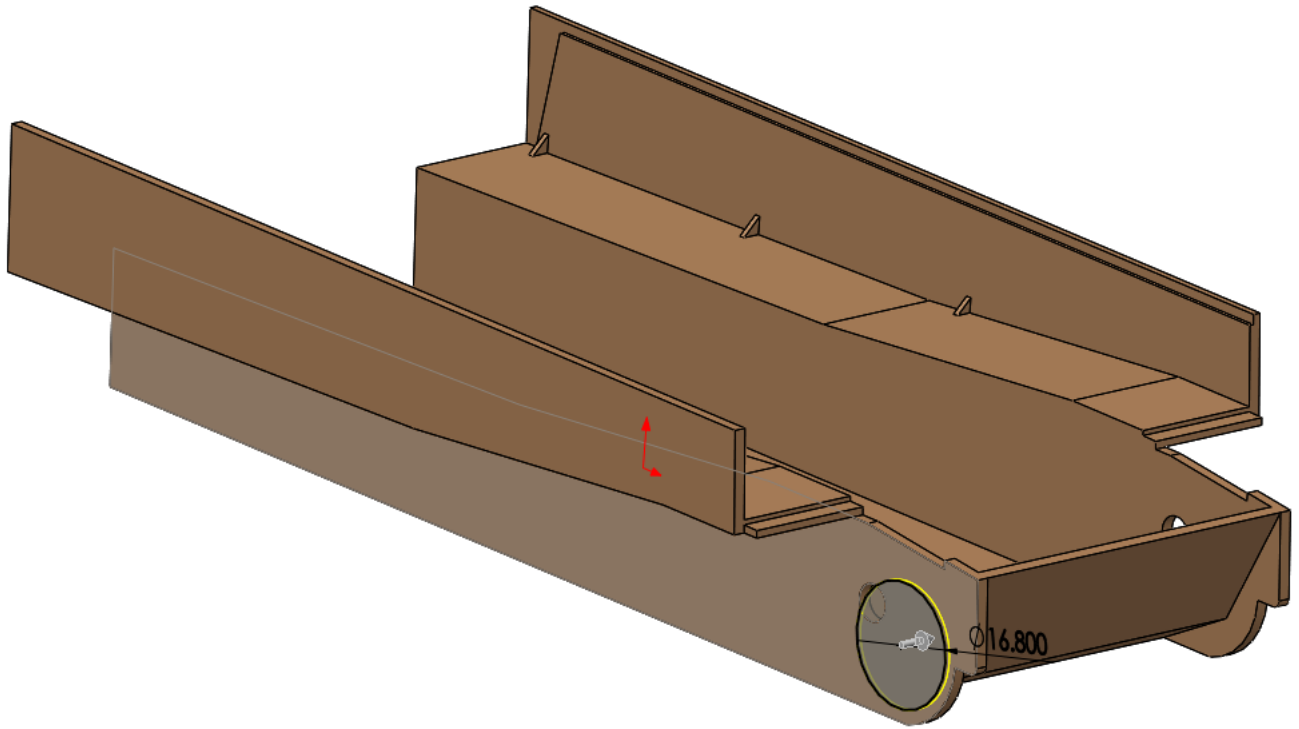


Draw the sketch on the surface of the part set its dimension and make it *concentric* with the round edge on the right bottom corner.

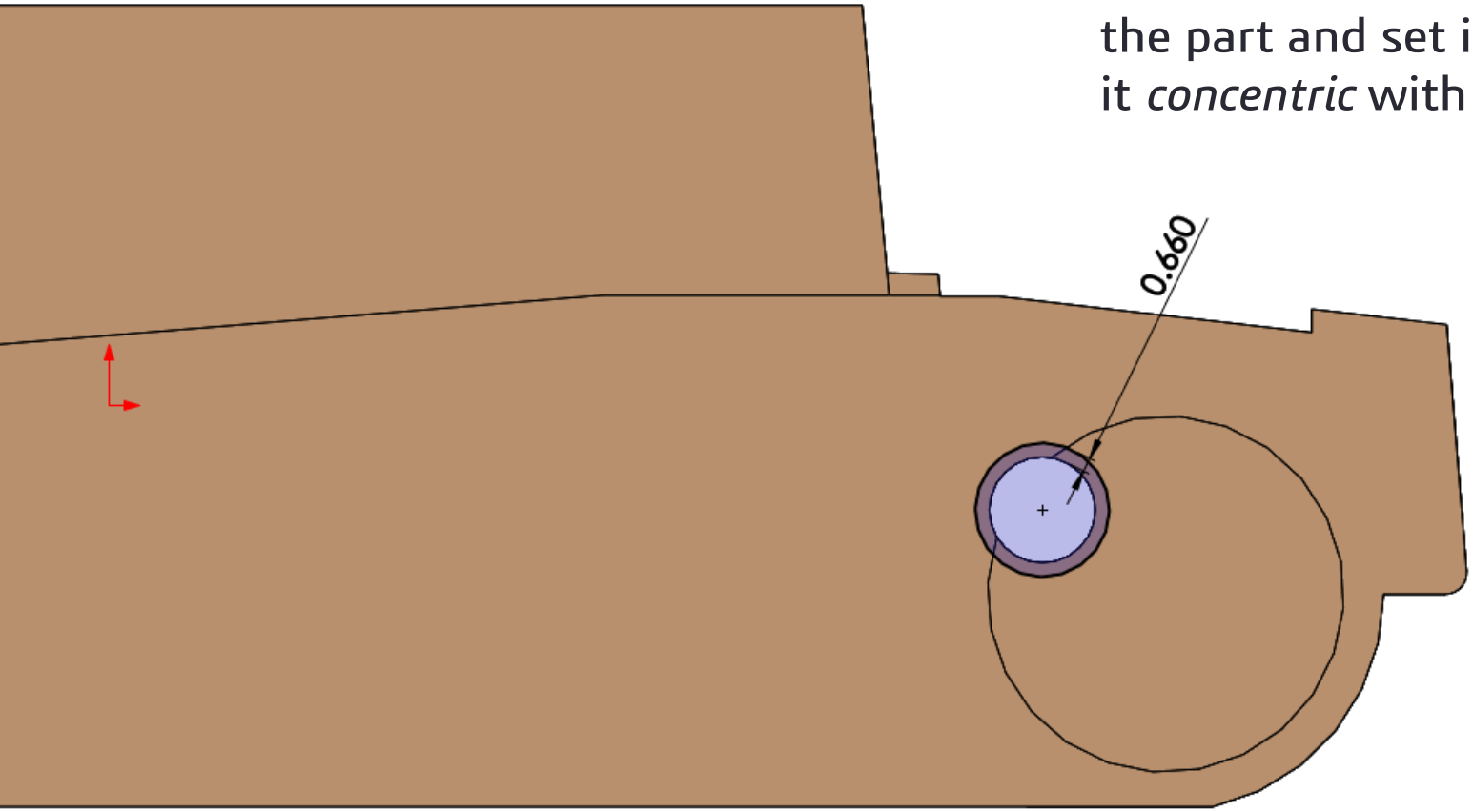


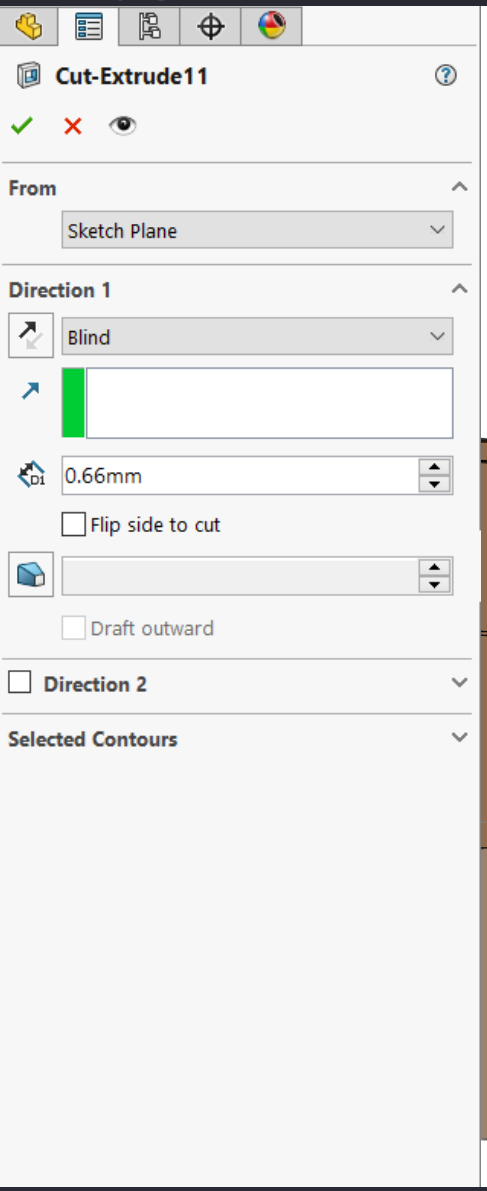


Use “Cut-Extrude” , set Direction 1 to *Blind* at **0.66** mm and cut it into the part.

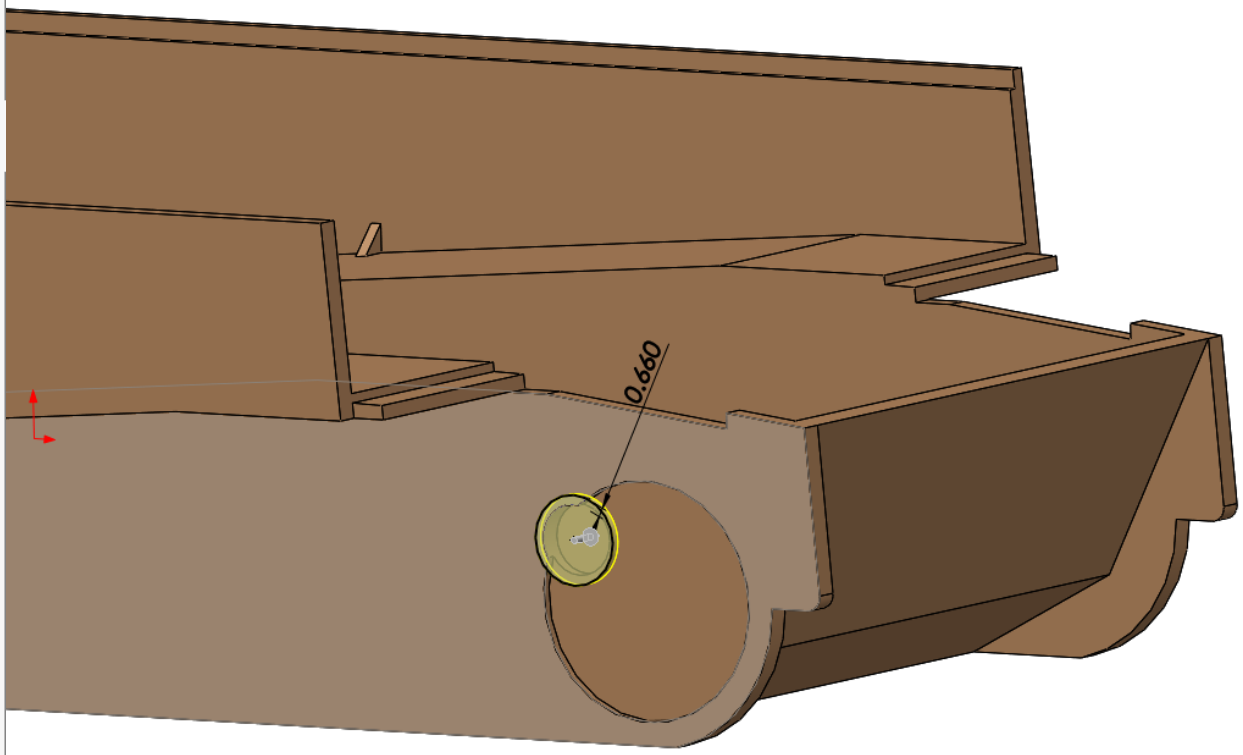


Draw the sketch on the surface of the part and set its dimension. Make it *concentric* with the circular hole.



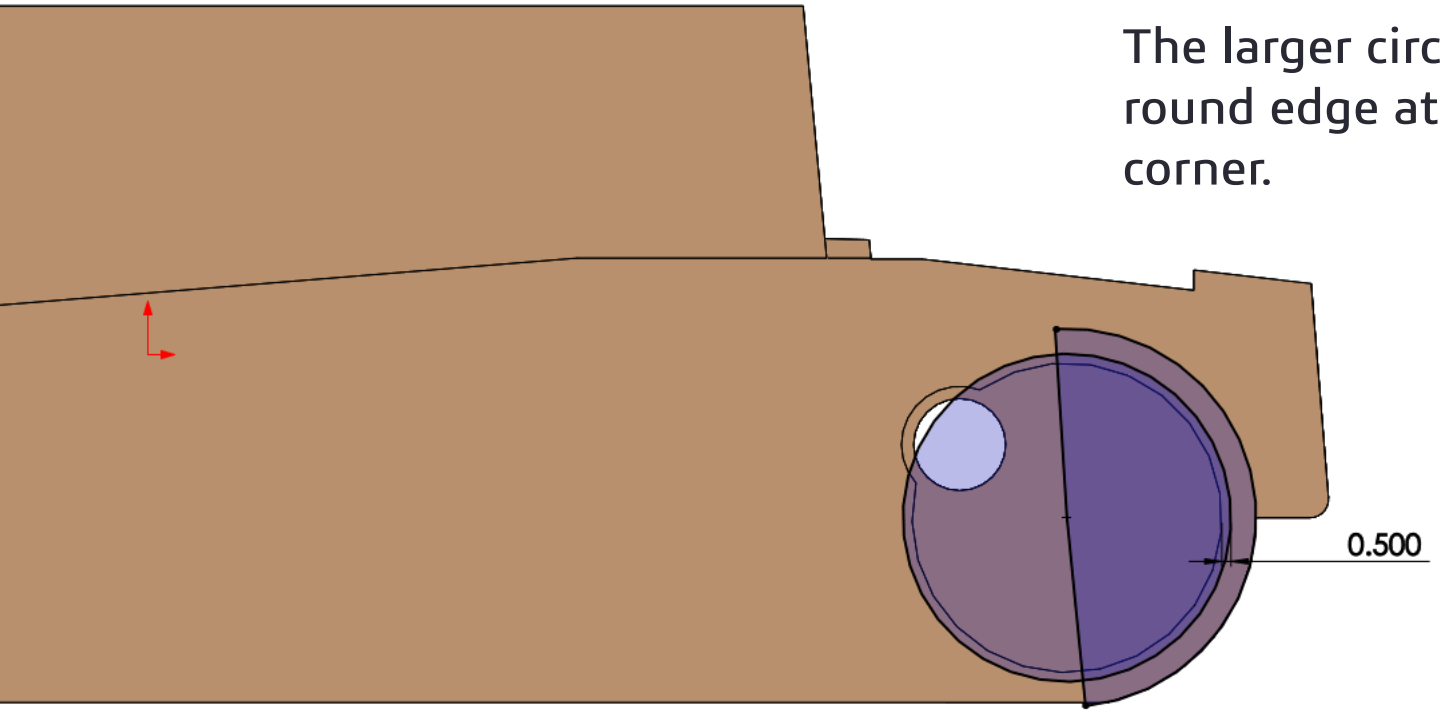


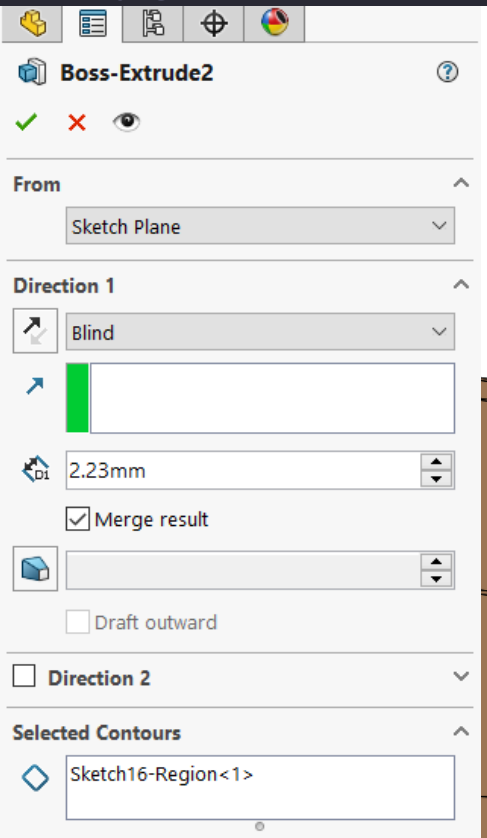
Use “Cut-Extrude” , set Direction 1 to *Blind* at **0.66** mm and cut it into the part.



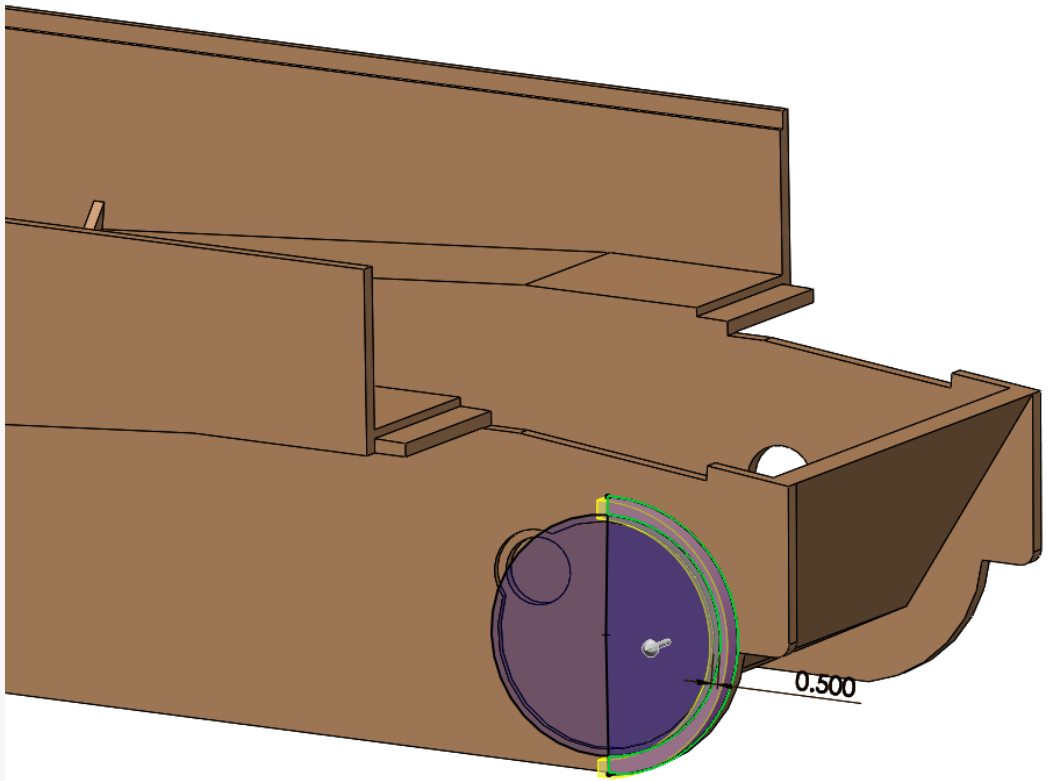
Draw the sketch on the surface of the part set its dimension and make it *concentric* with the round edge on the right bottom corner.

The larger circle is *coradial* with the round edge at the right bottom corner.

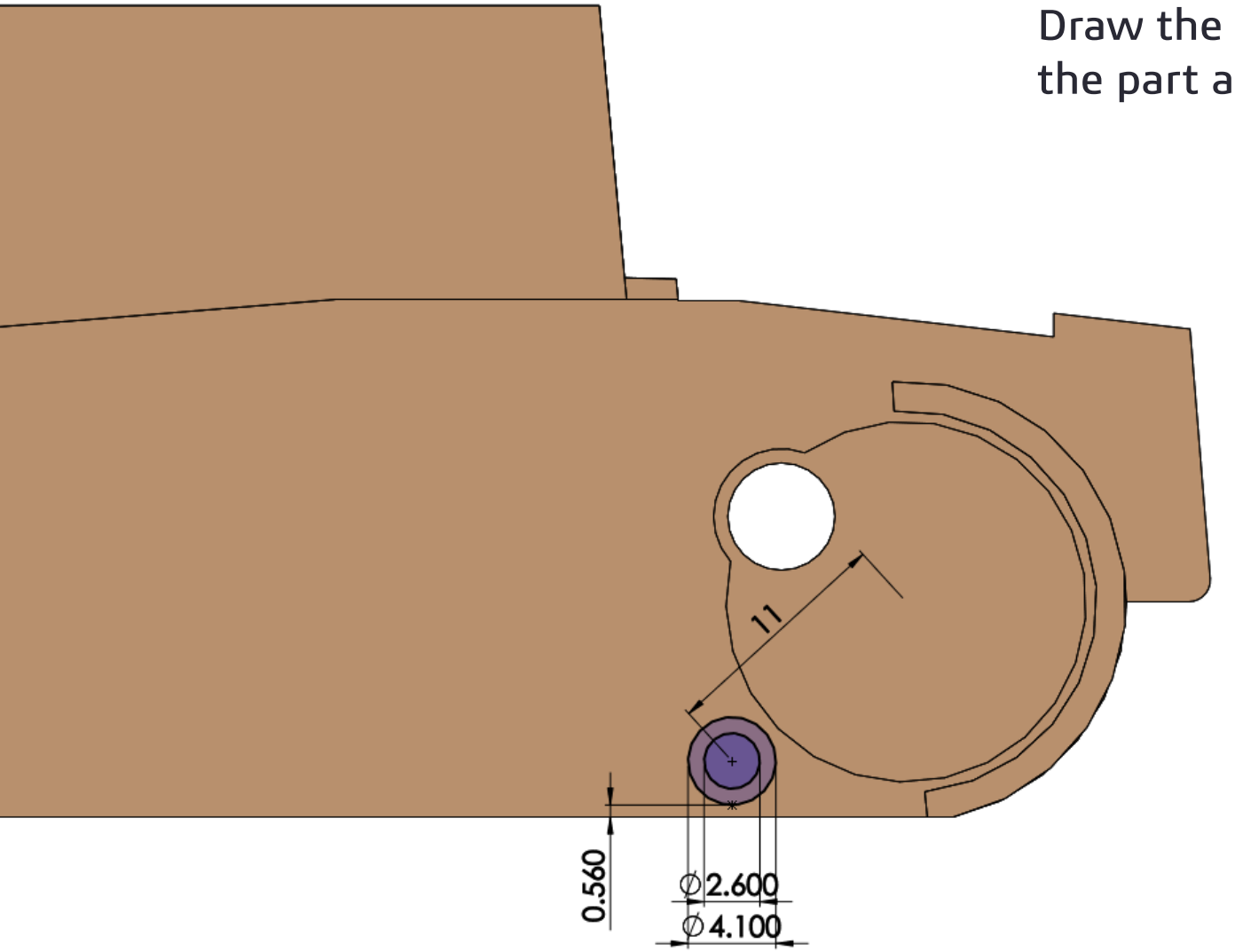


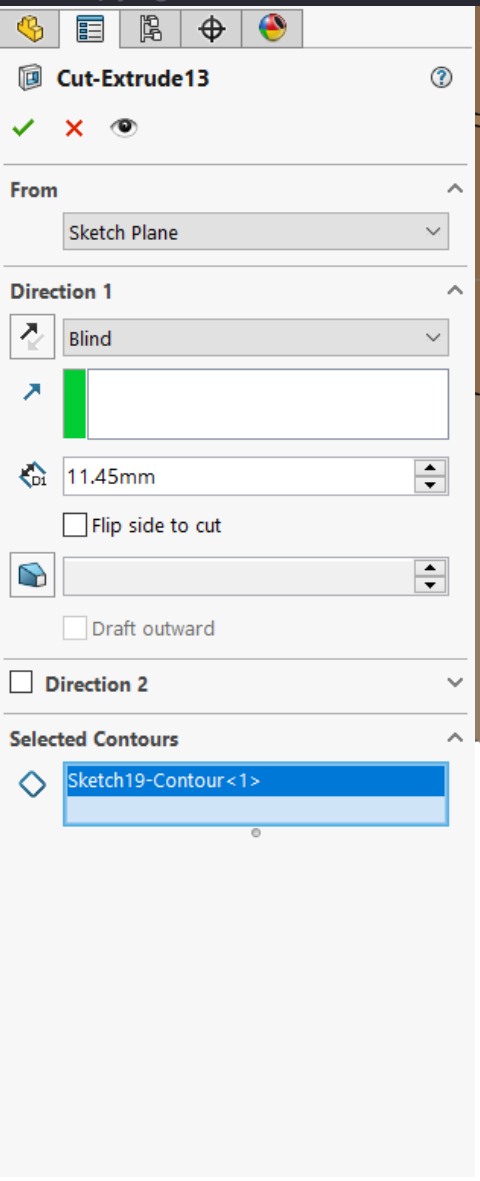


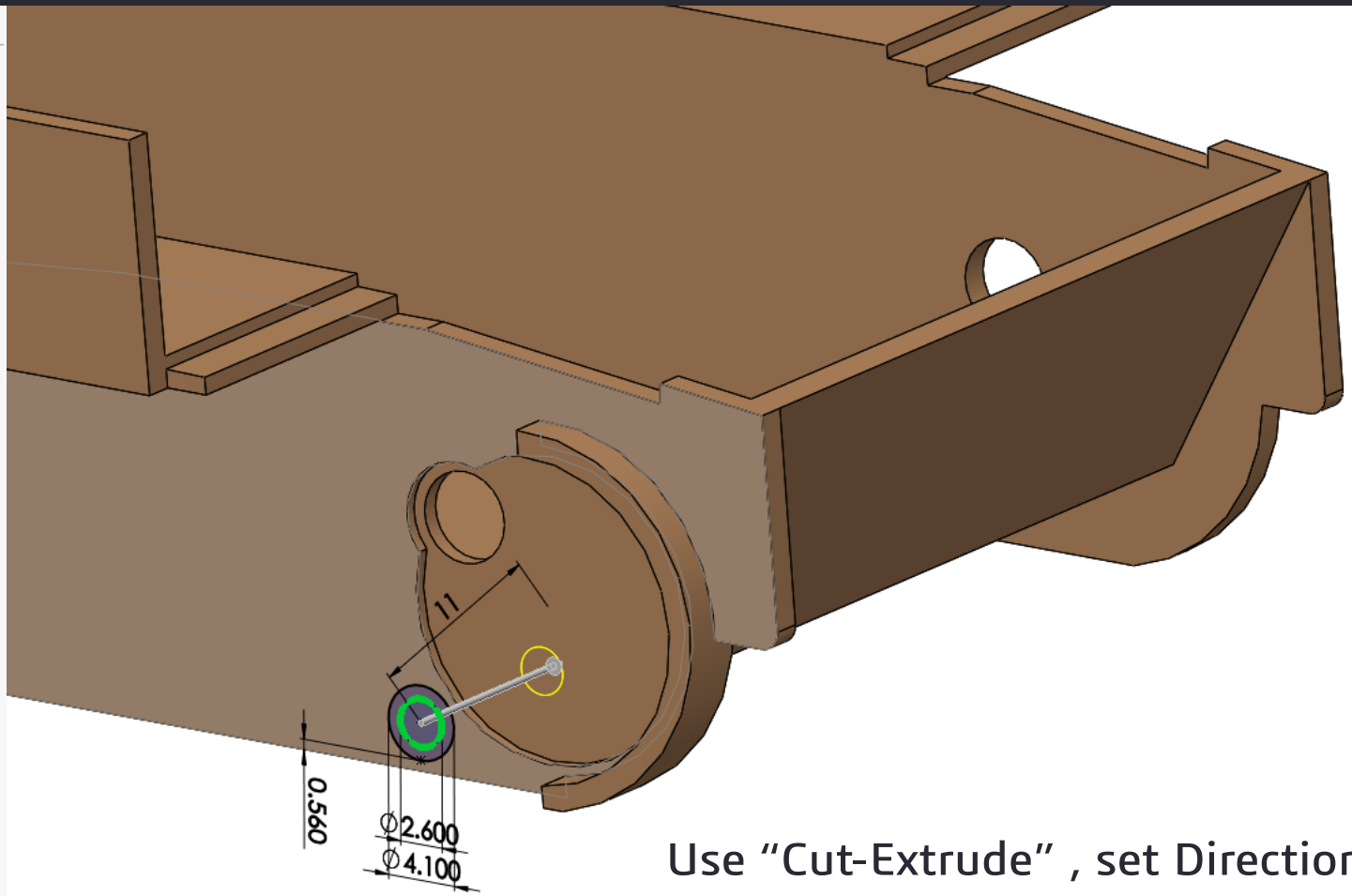
Use “Boss-Extrude” , set Direction 1 to *Blind* at **2.23** mm, select the outer half circle area in the *Selected Contour* and extrude it.



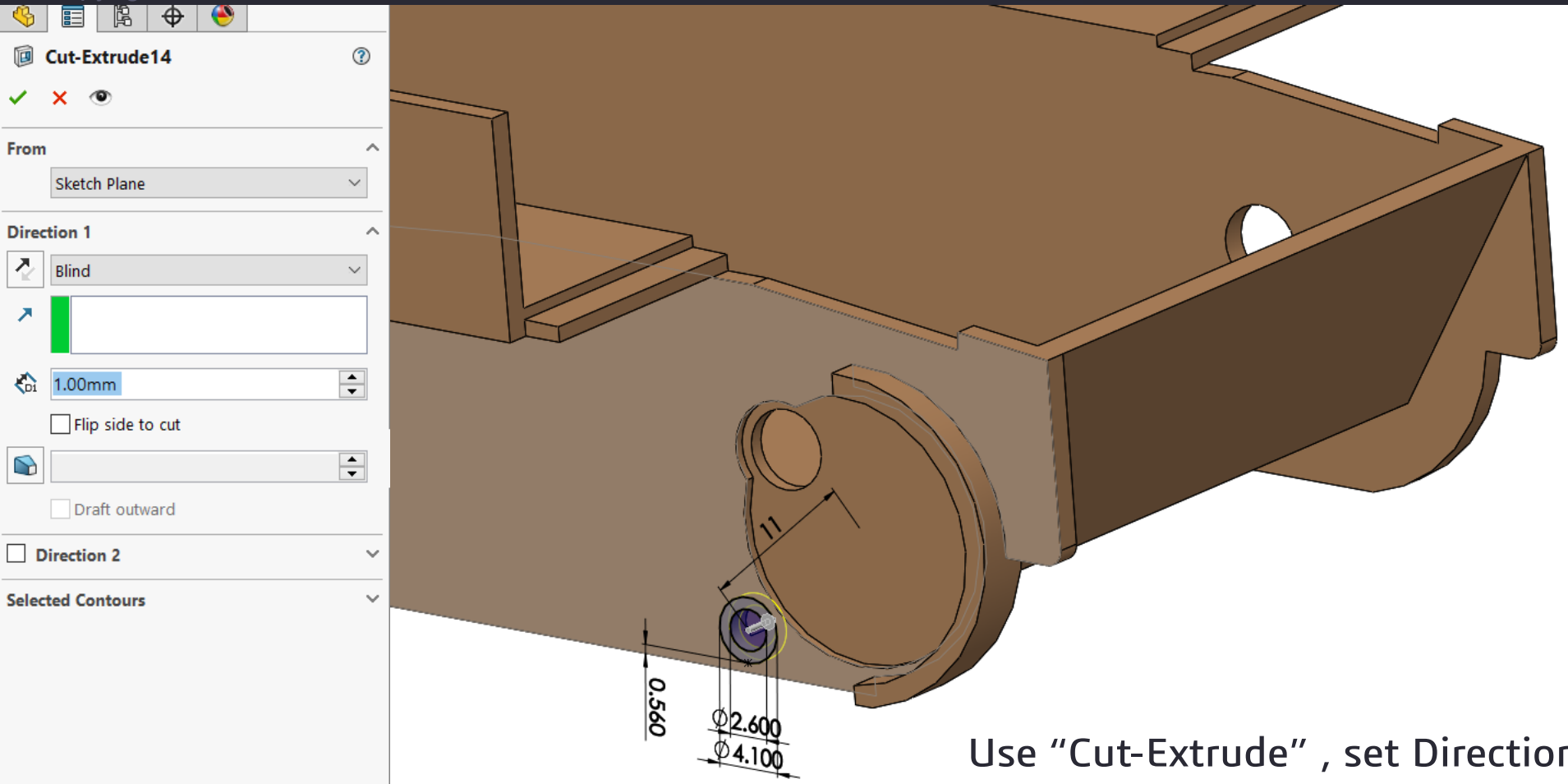
Draw the sketch on the surface of the part and set its dimensions.



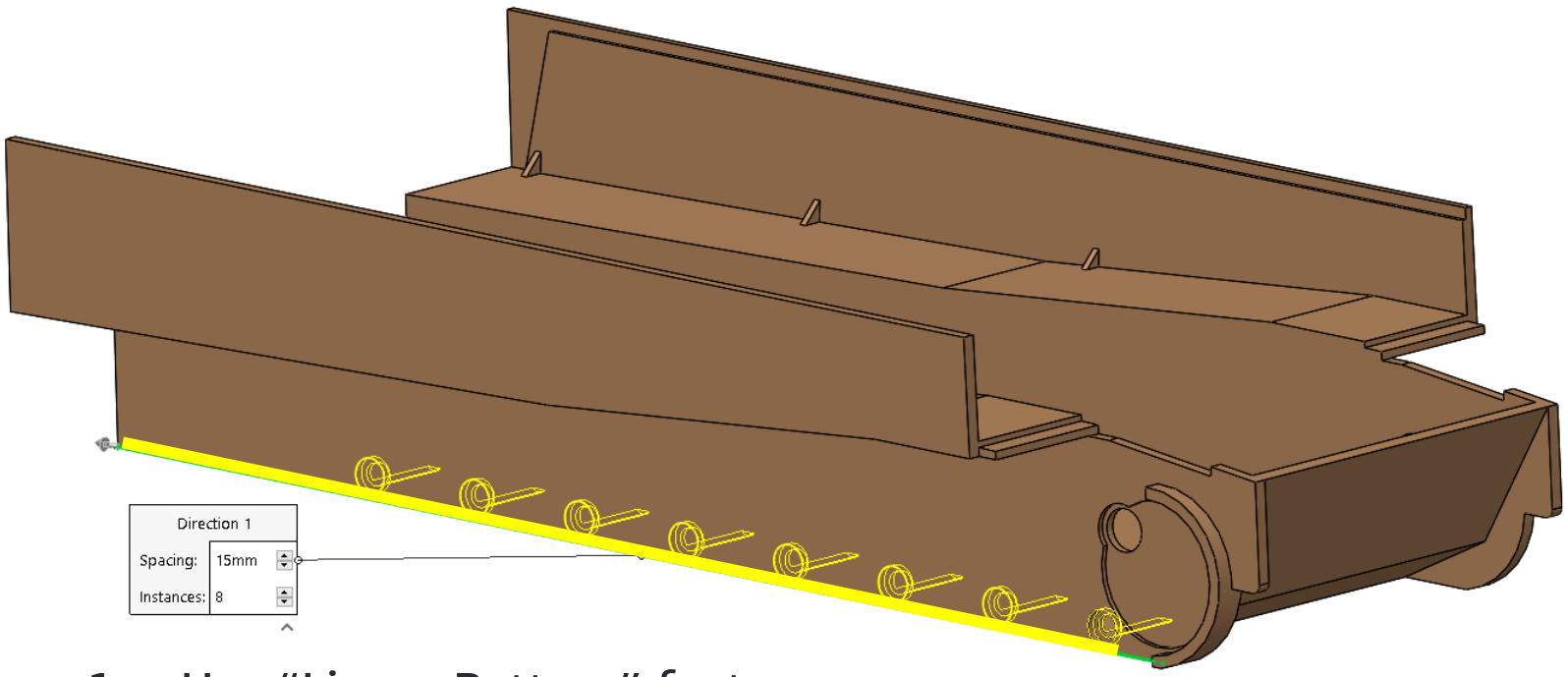


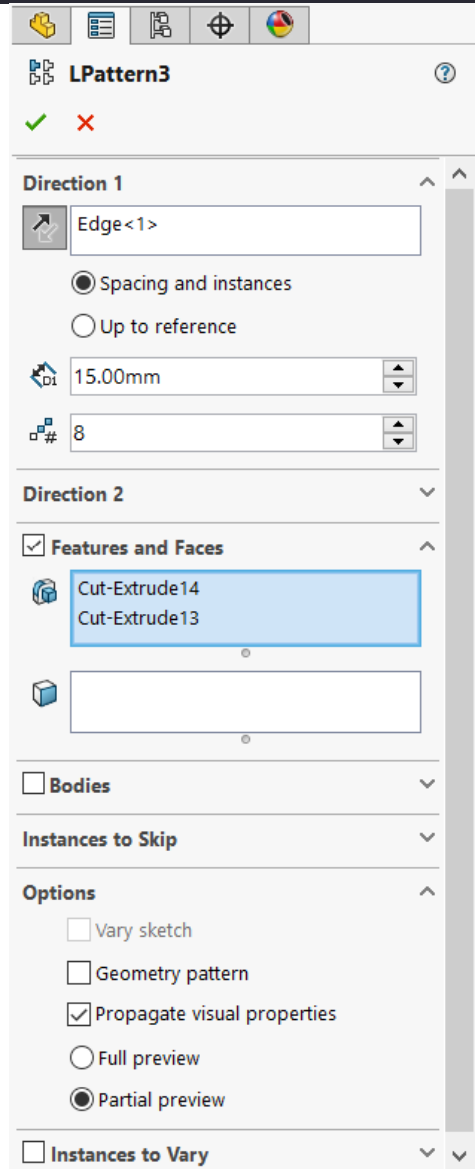


Use “Cut-Extrude” , set Direction 1 to *Blind* at **11.45** mm, select the inner circle in the *Selected Contour* and cut it into the part.

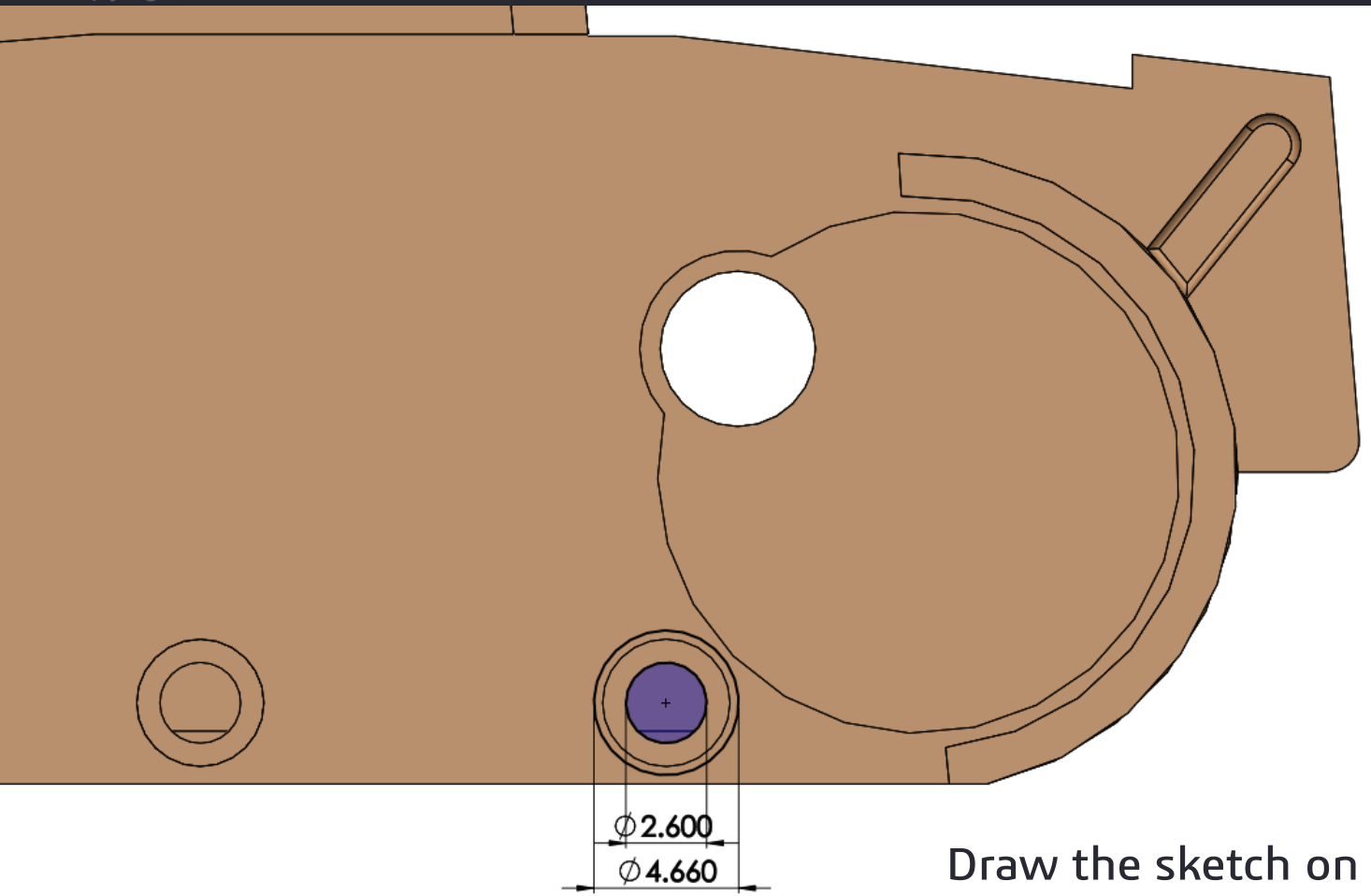


Use “Cut-Extrude” , set Direction 1 to *Blind* at **1** mm and cut it into the part.



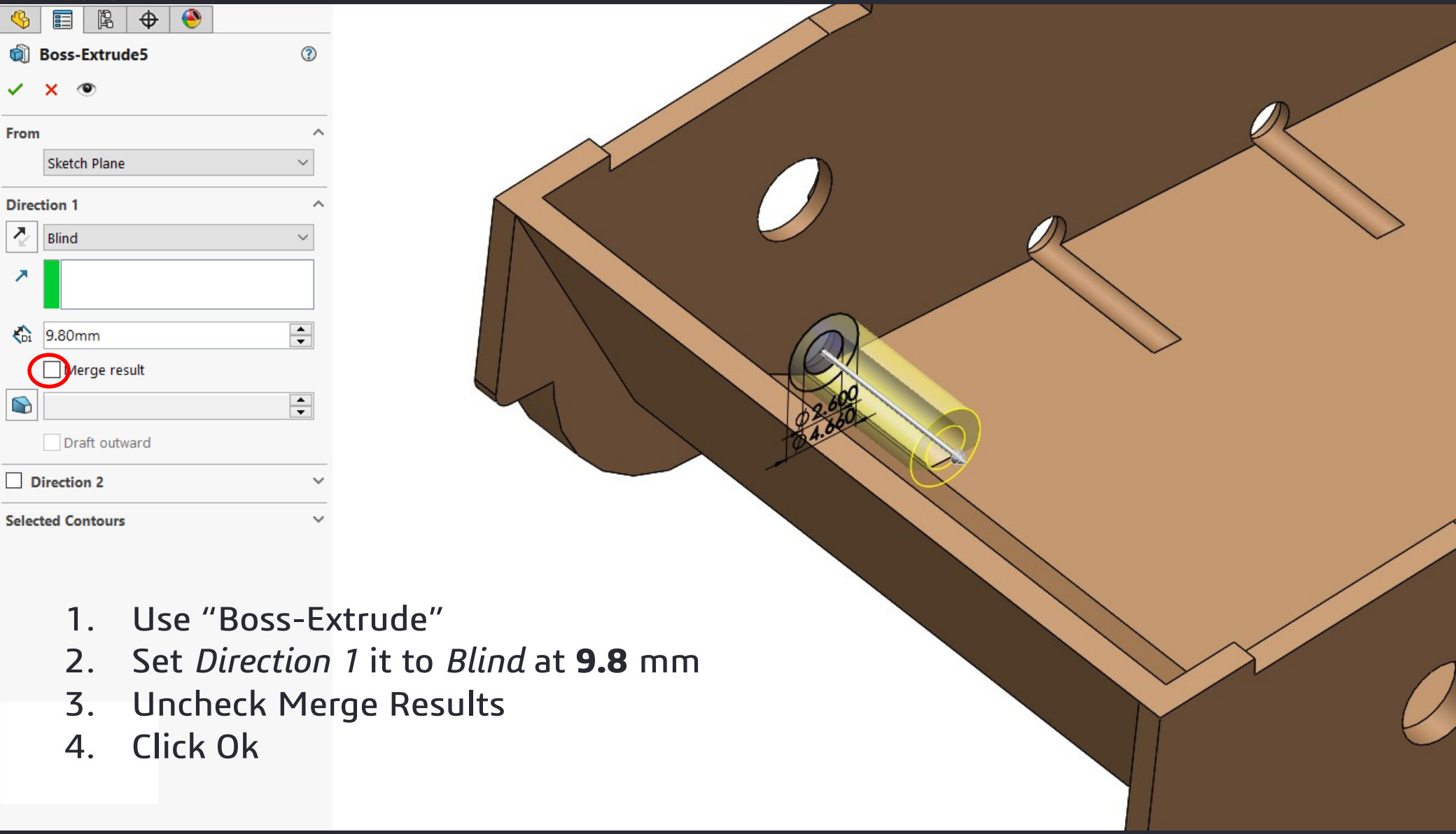


1. Use “Linear Pattern” feature
2. Select the yellow edge as *Direction 1*
3. Set the distance to **15** mm
4. Set the number of instances to 8
5. Select the two recently Cut-Extrudes in the *Features and Faces*
6. Click Ok

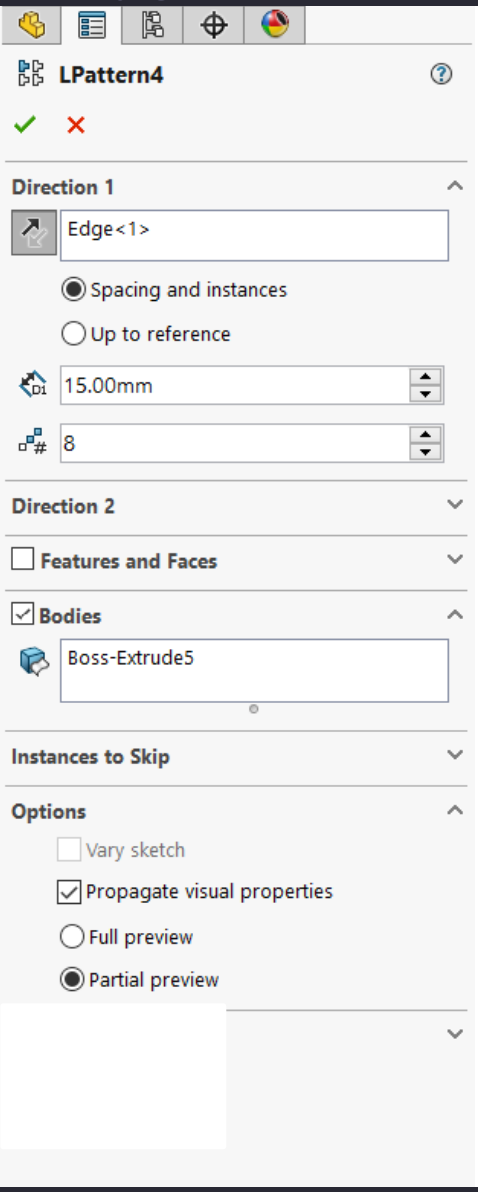


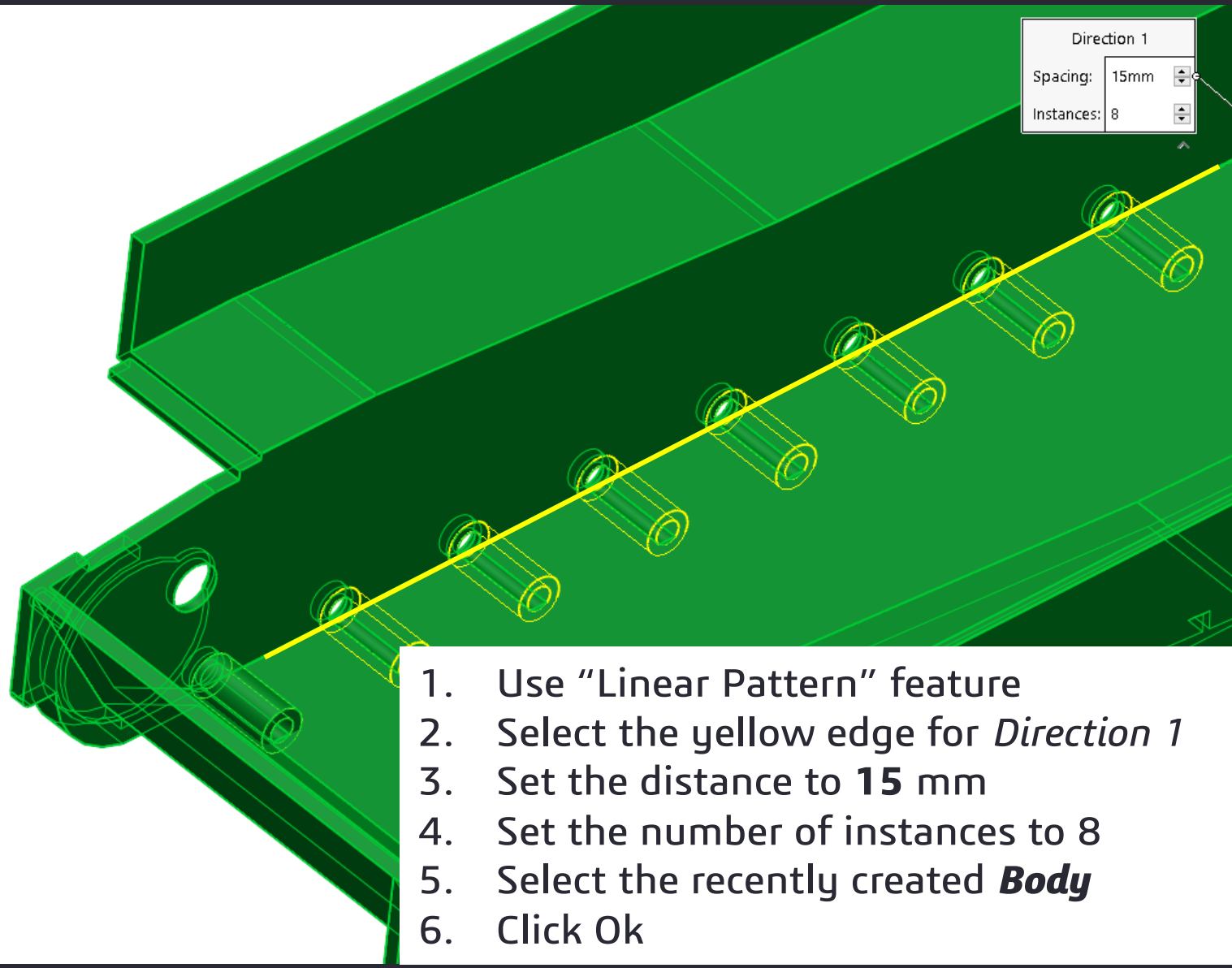
Draw the sketch on the surface
Of the part and set its dimensions.

Make them *concentric* with the circular hole

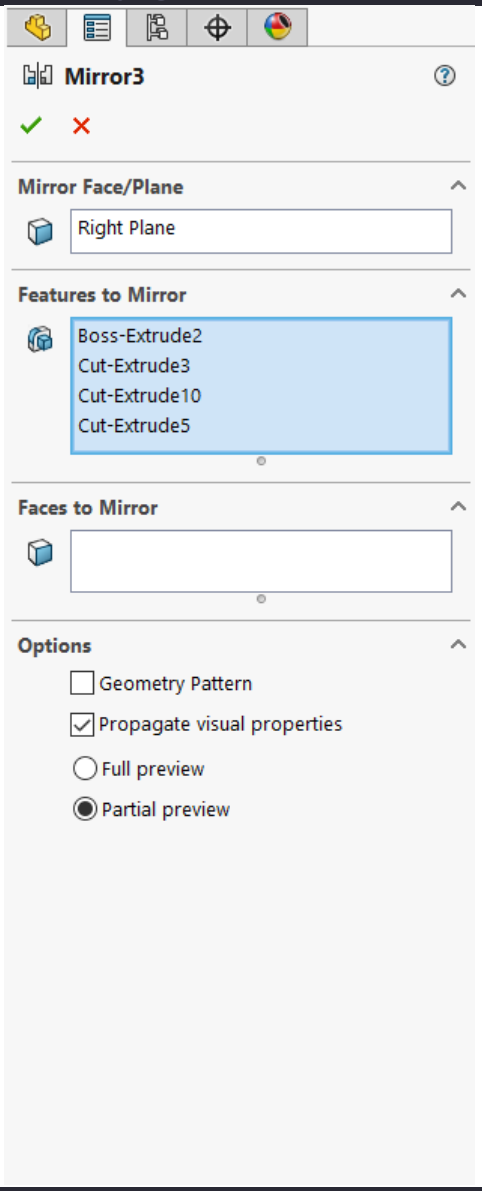


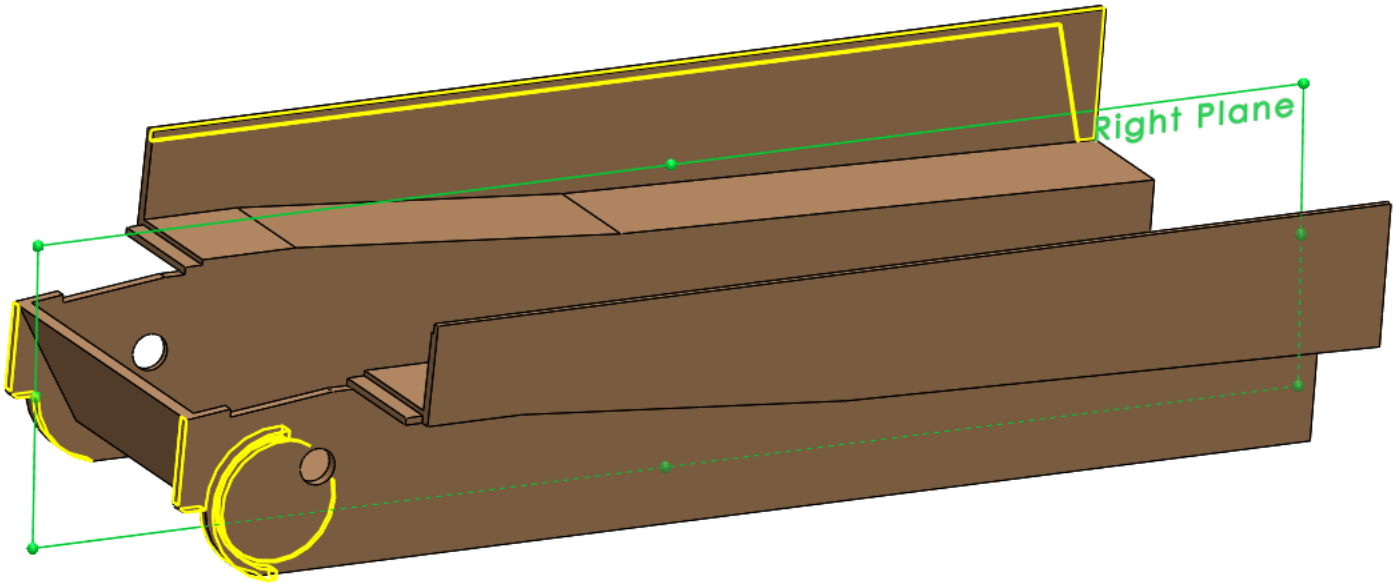
1. Use “Boss-Extrude”
2. Set *Direction 1* it to *Blind* at **9.8** mm
3. Uncheck Merge Results
4. Click Ok



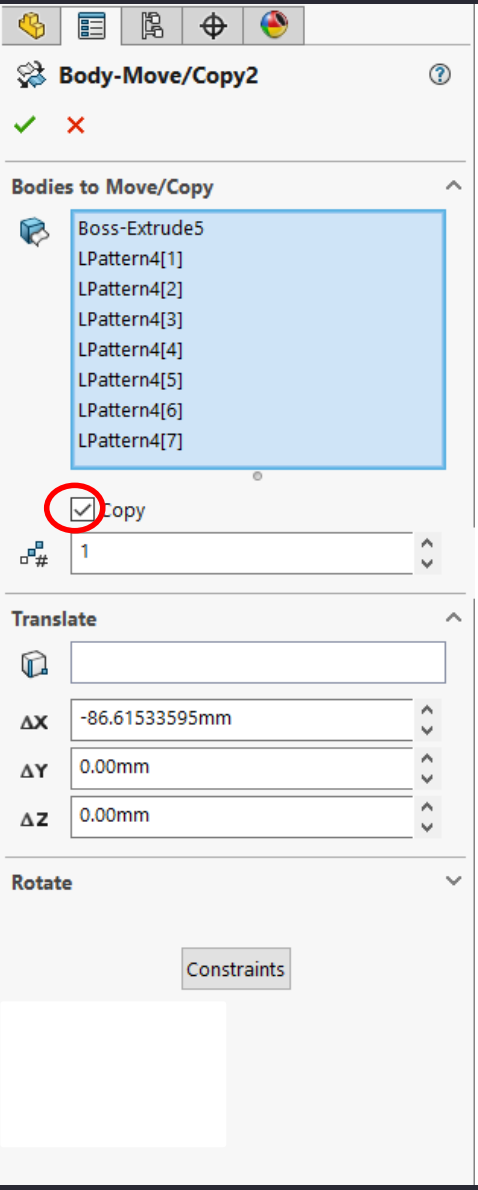


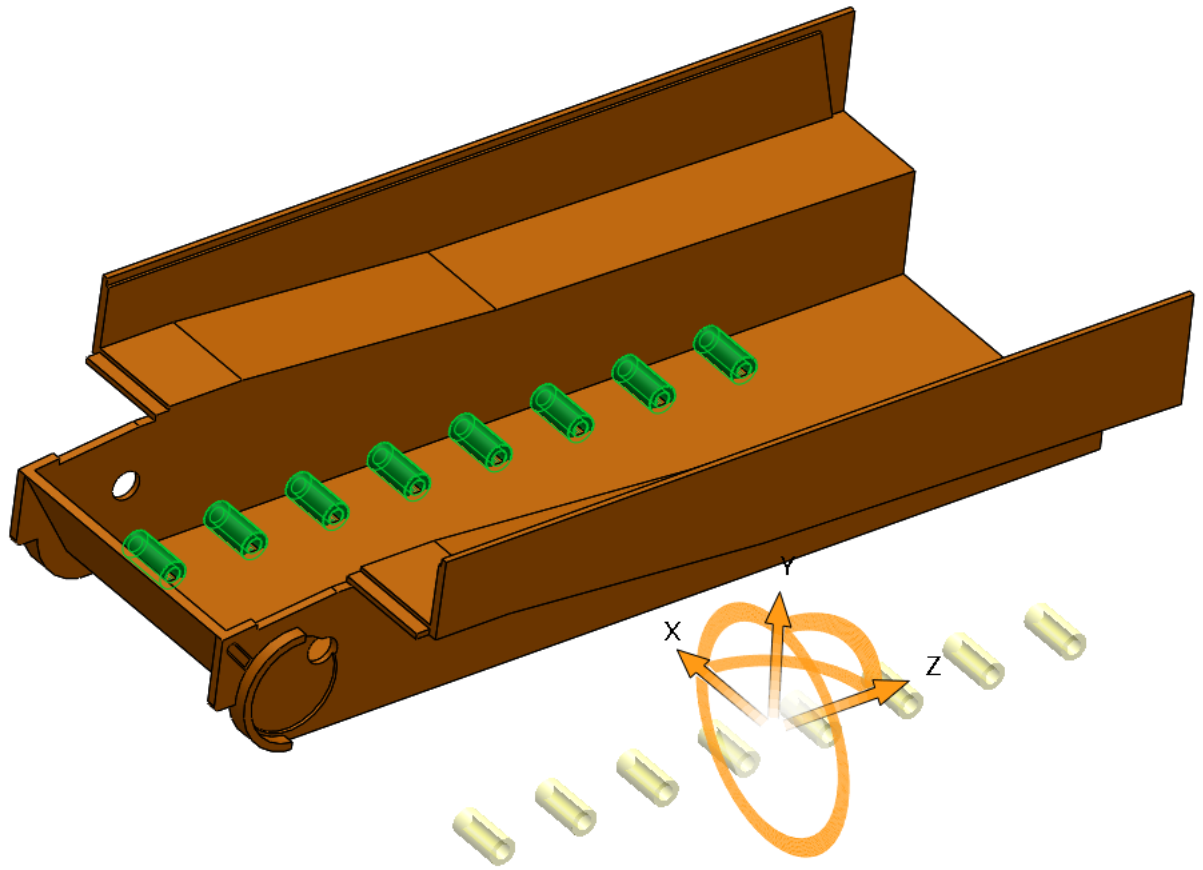
1. Use “Linear Pattern” feature
2. Select the yellow edge for *Direction 1*
3. Set the distance to **15** mm
4. Set the number of instances to 8
5. Select the recently created ***Body***
6. Click Ok



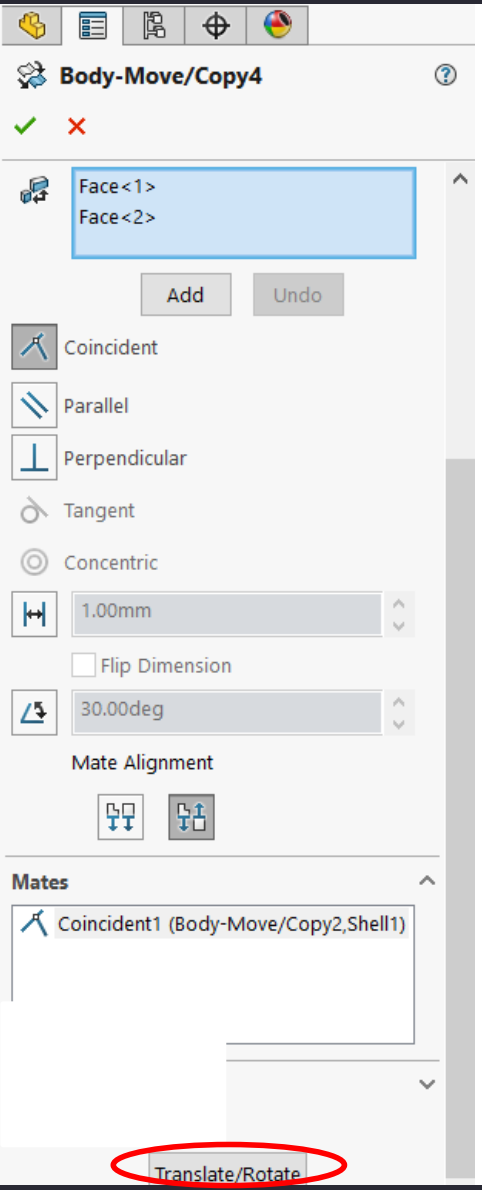


1. Use “Mirror”
2. Select the **Right Plane** for Mirror Face/Plane
3. Select the four yellow highlighted features to mirror

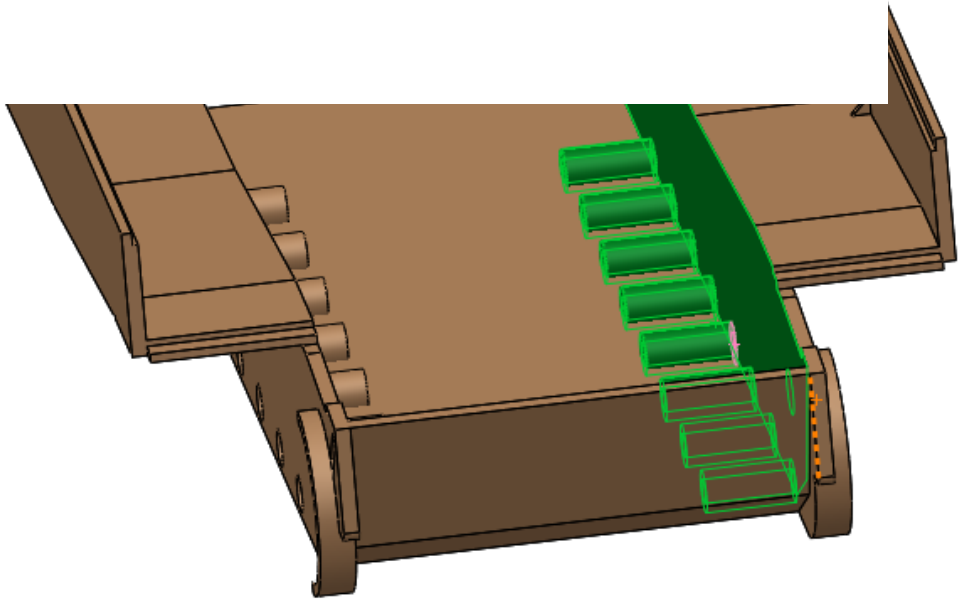


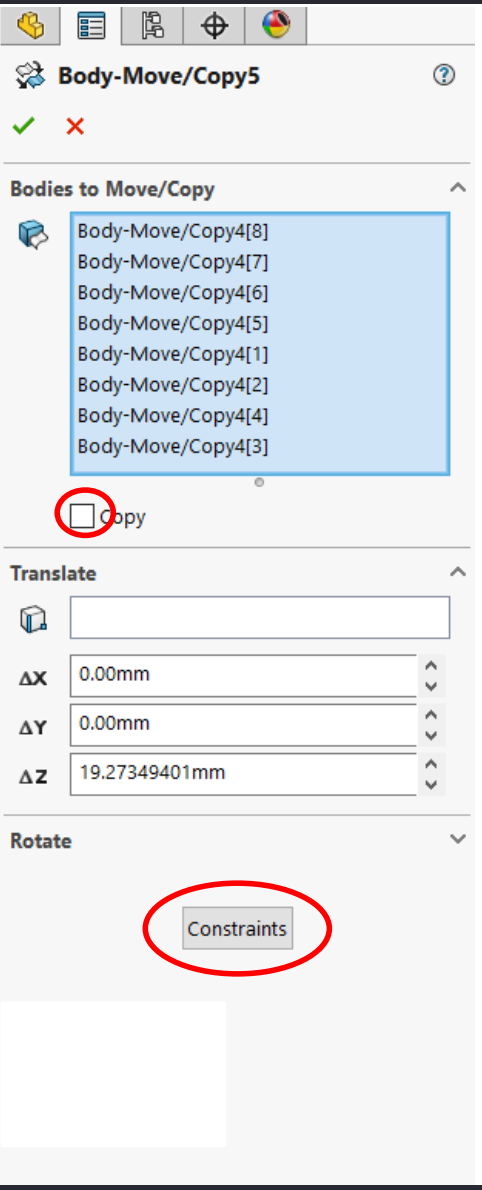


1. Use “Body-Move/Copy”
2. Check **Copy**
3. Select the eight green highlighted bodies (Cylinders)
4. Drag them about **86** mm in the **X** direction

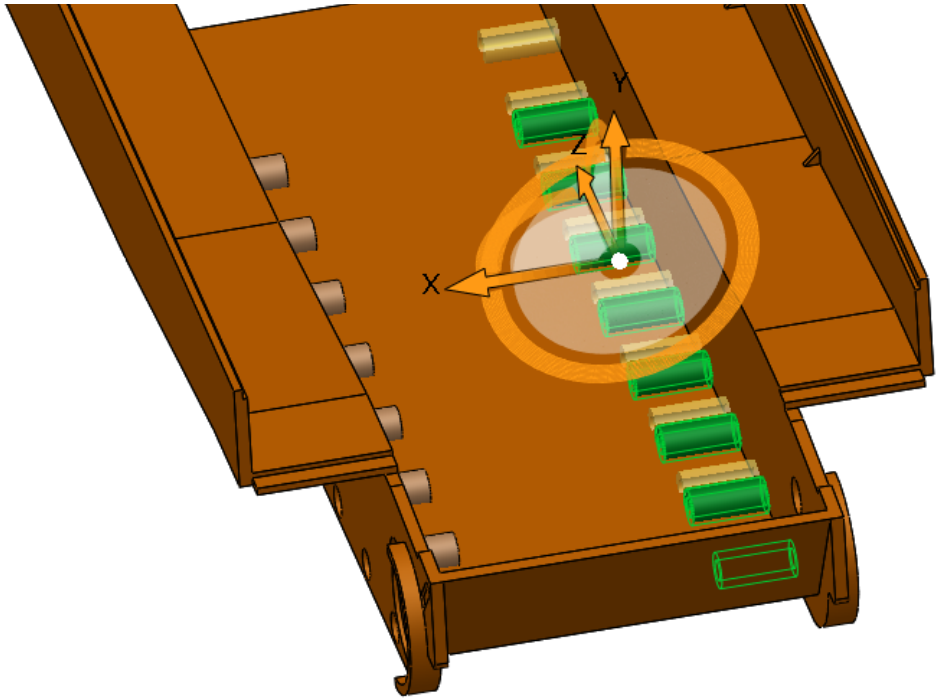


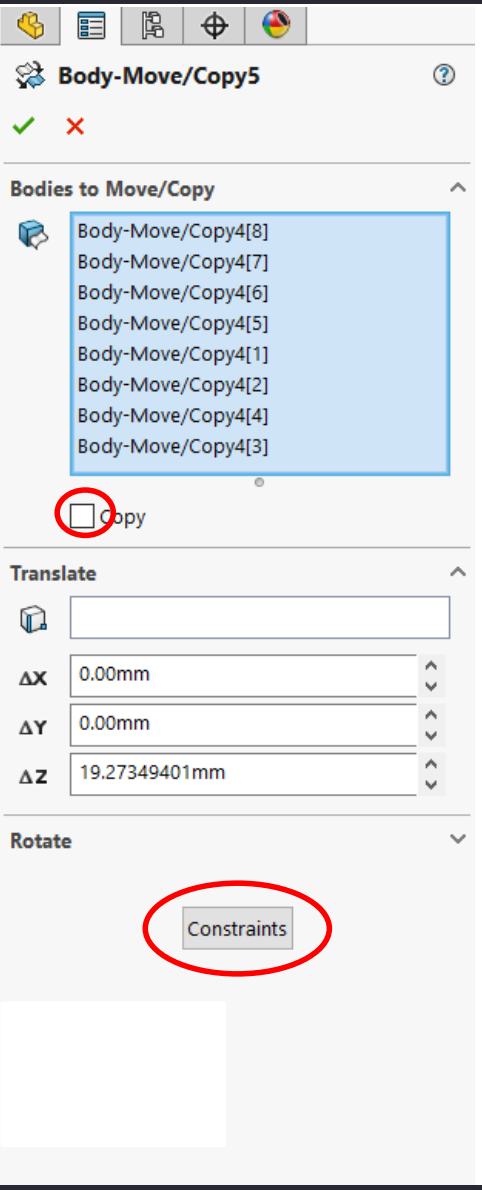
1. Use “Body-Move/Copy”
2. Click on the Constraints to switch it to **Translate/Rotate**
3. Select the end surface of the eight cylinders and the green highlighted surface on the main part
4. Choose *Coincident*
5. Click Ok



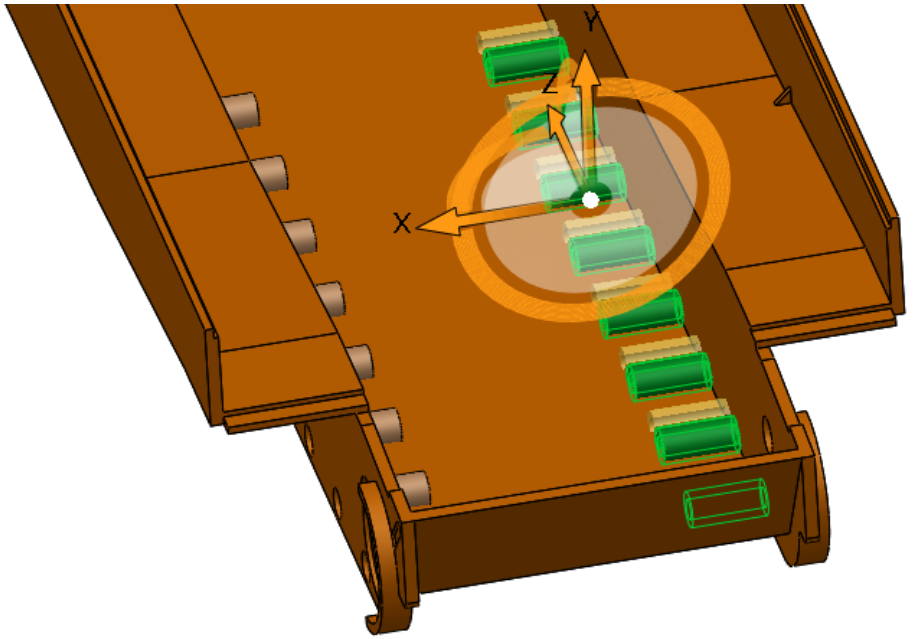


1. Use "Body-Move/Copy"
2. Click on the Translate/Rotate to switch it to **Constraints**
3. Select the eight cylinders
4. Uncheck *Copy*
5. Move them exactly **19.27349** mm in the Z direction

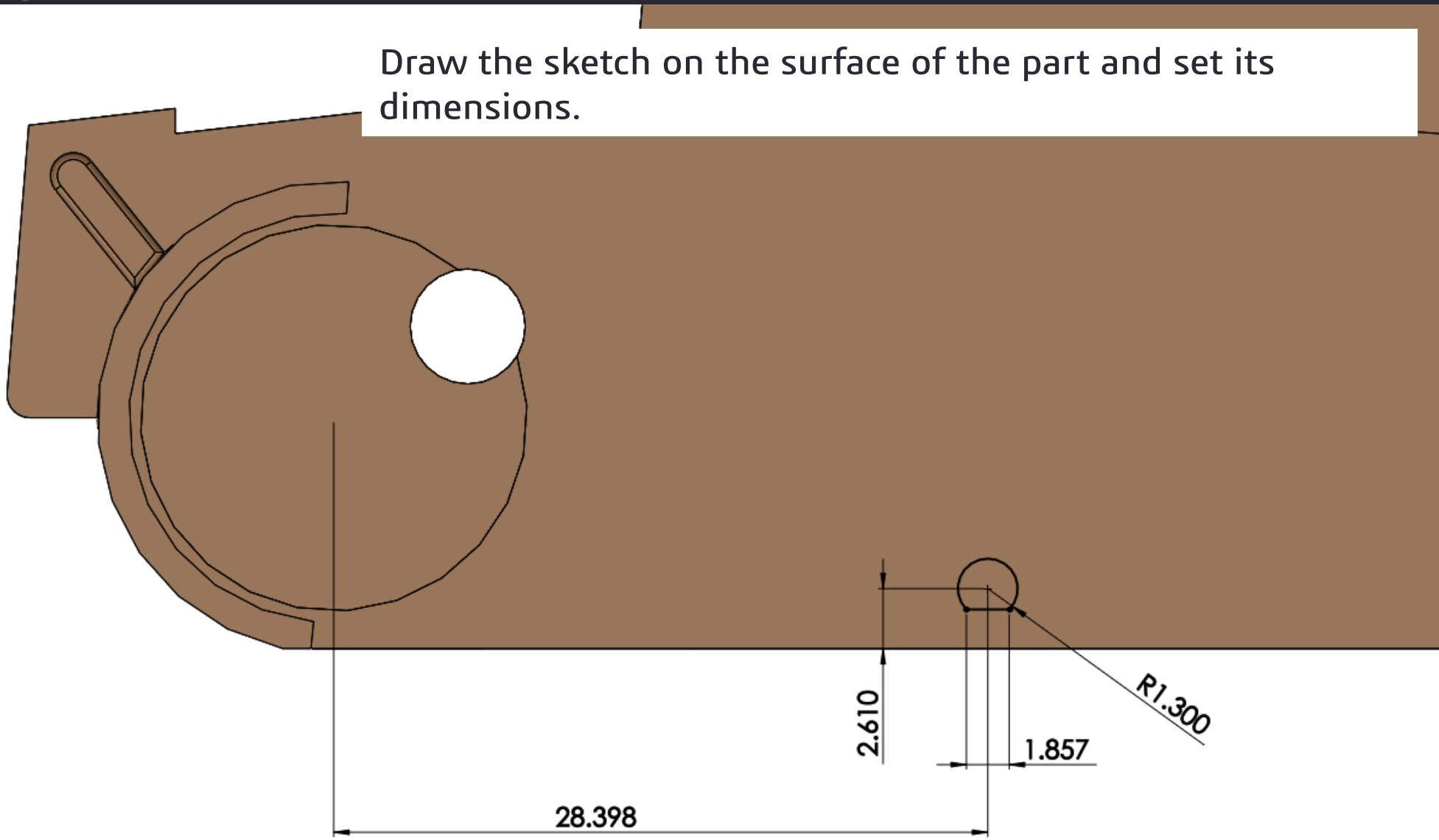


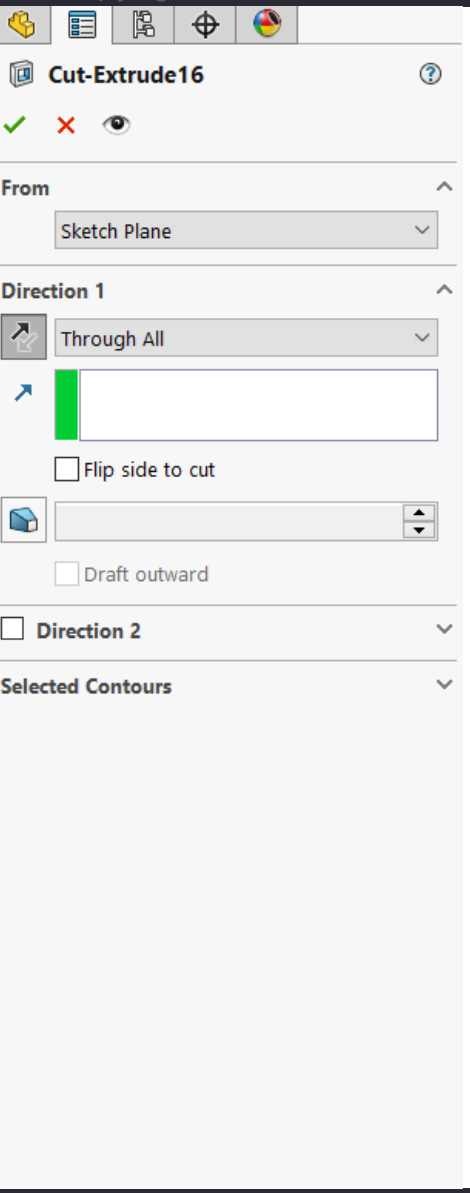


1. Use “Body-Move/Copy”
2. Click on the Translate/Rotate to switch it to **Constraints**
3. Select the eight cylinders
4. Uncheck *Copy*
5. Move them exactly **19.27349** mm in the **Z** direction
6. Click Ok

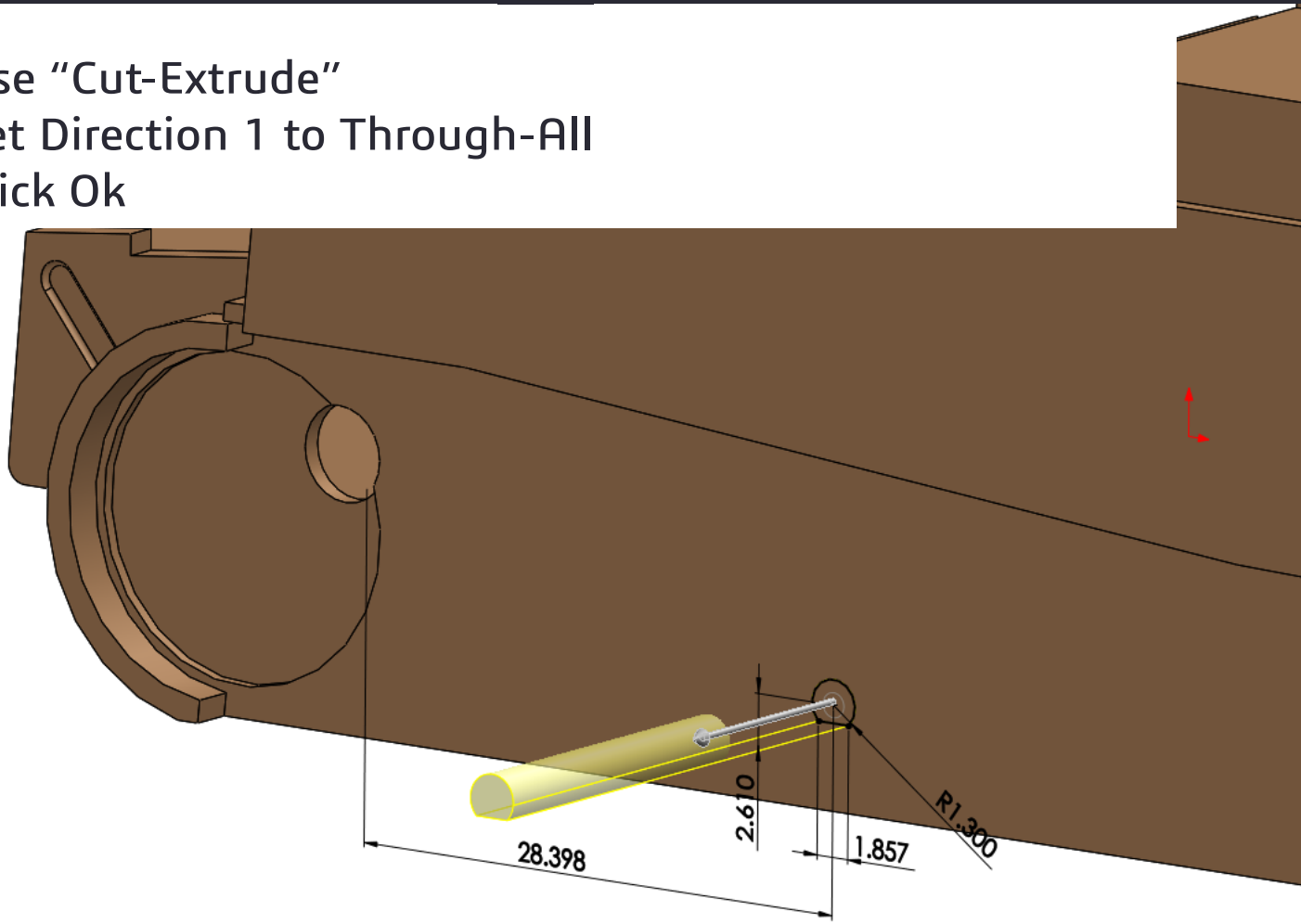


Draw the sketch on the surface of the part and set its dimensions.



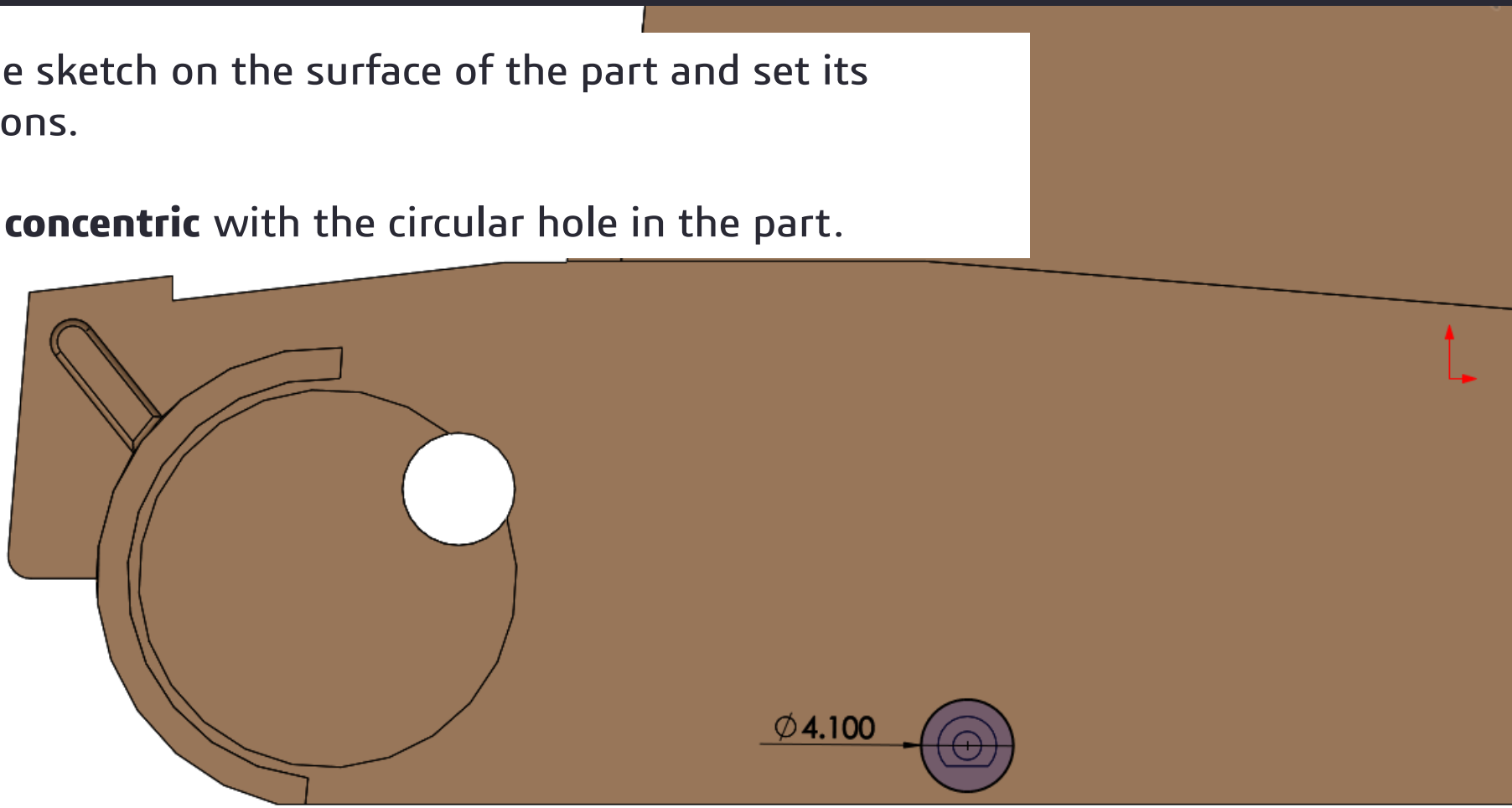


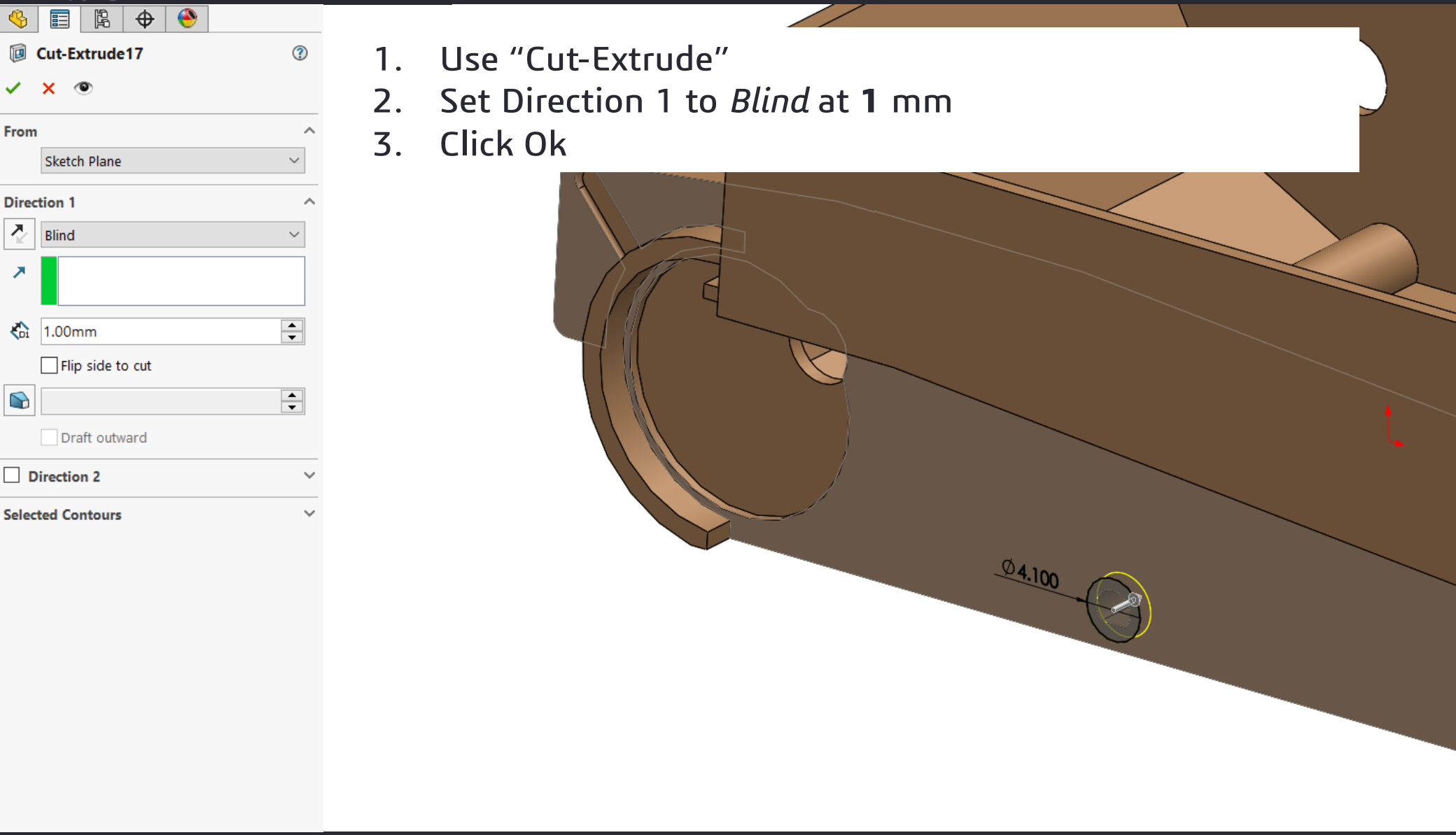
1. Use “Cut-Extrude”
2. Set Direction 1 to Through-All
3. Click Ok



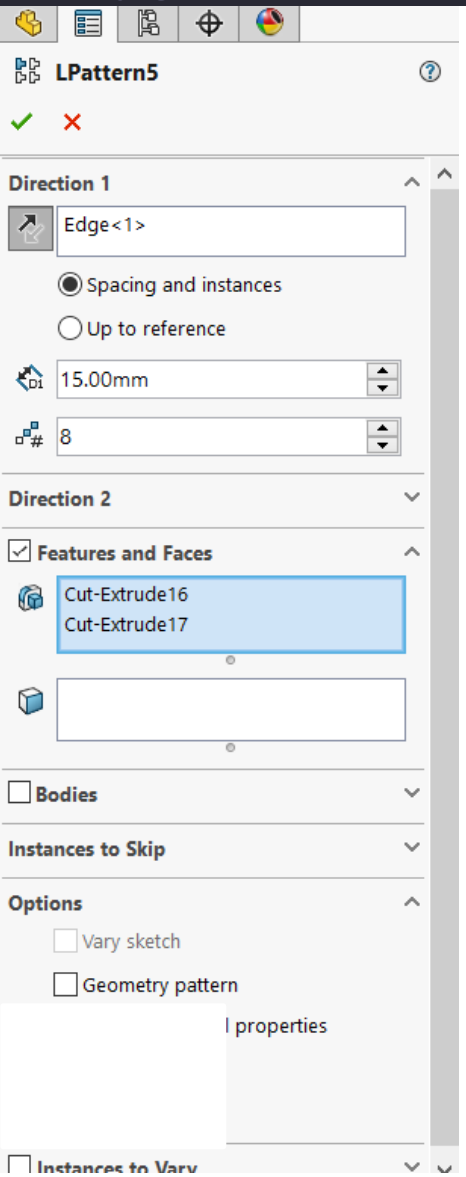
Draw the sketch on the surface of the part and set its dimensions.

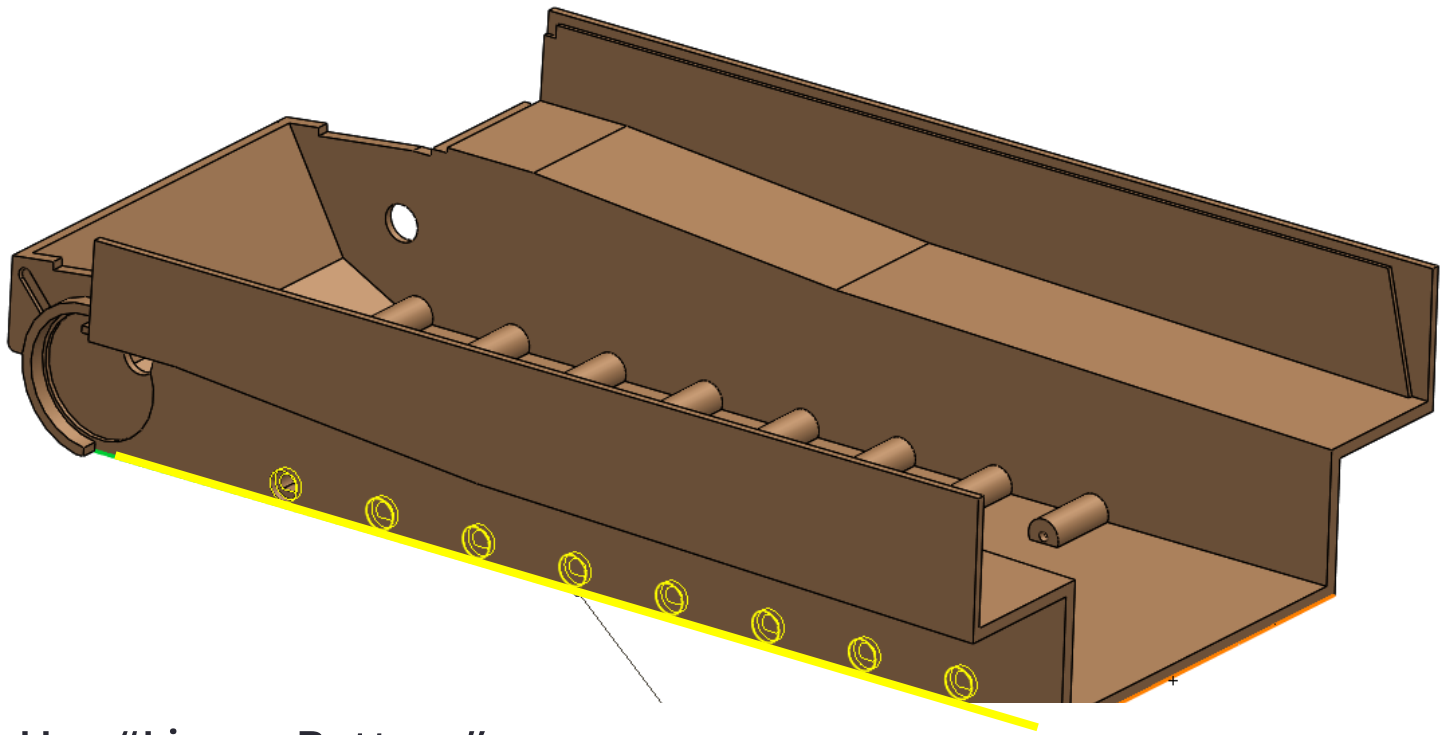
Make it **concentric** with the circular hole in the part.



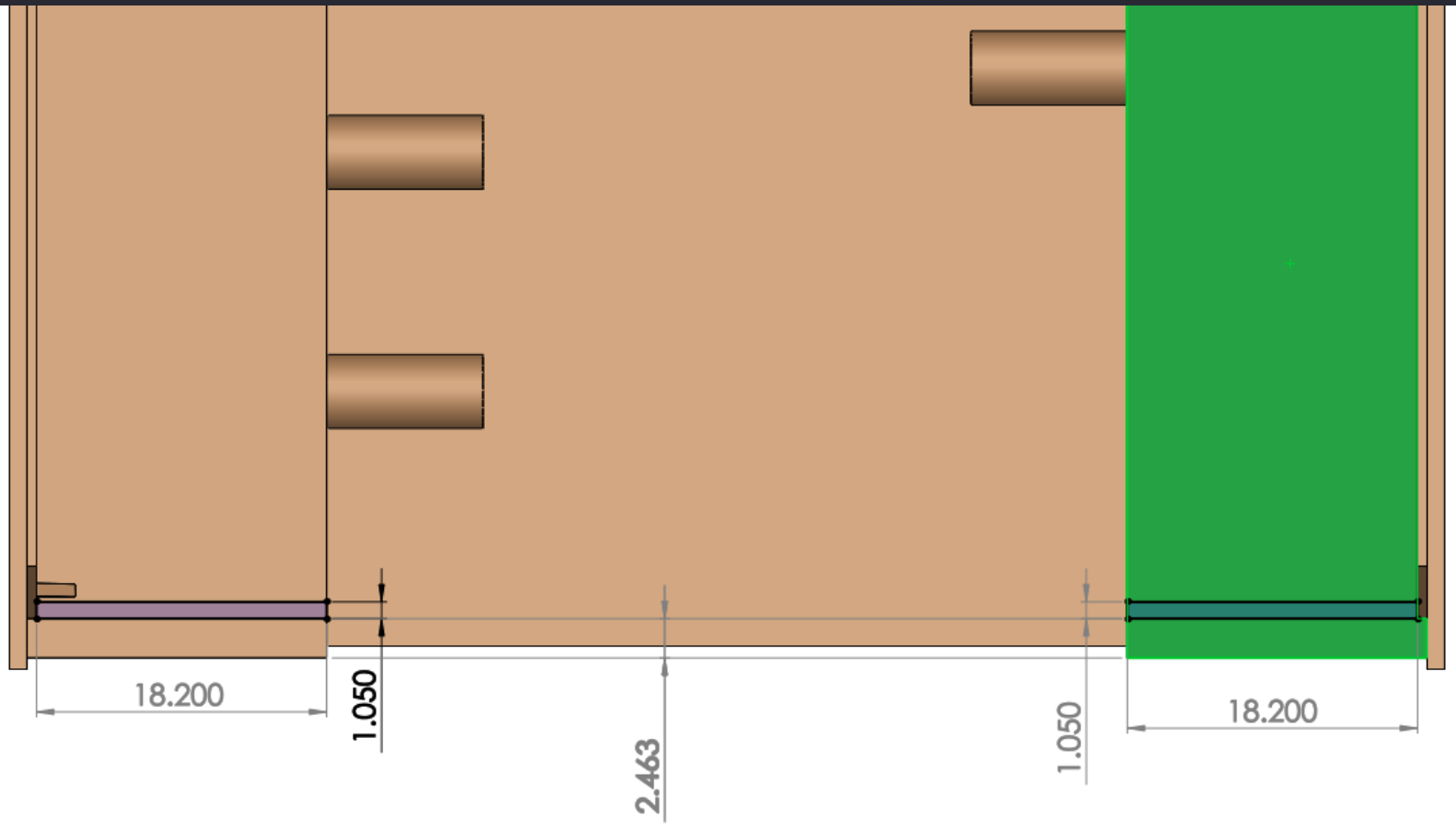


1. Use “Cut-Extrude”
2. Set Direction 1 to *Blind* at **1** mm
3. Click Ok

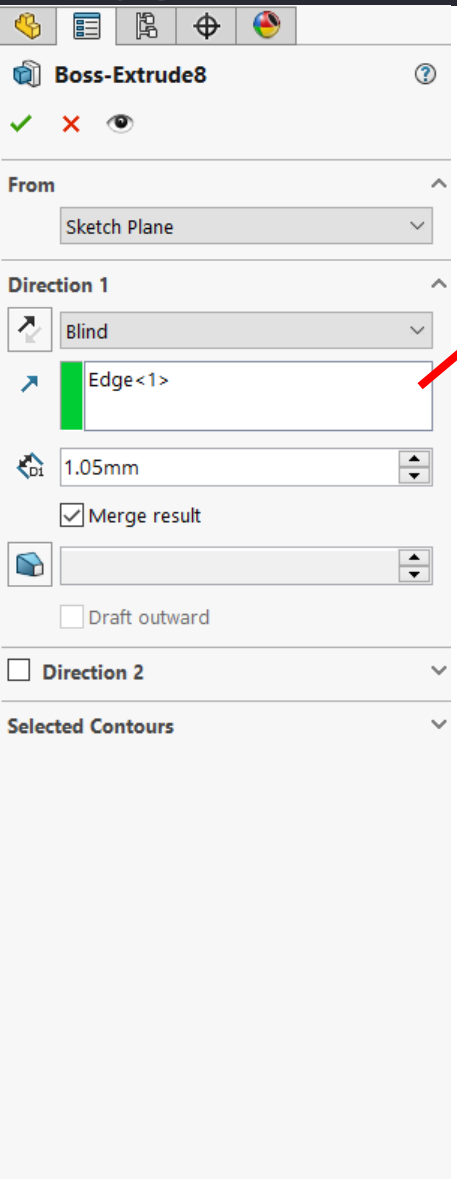




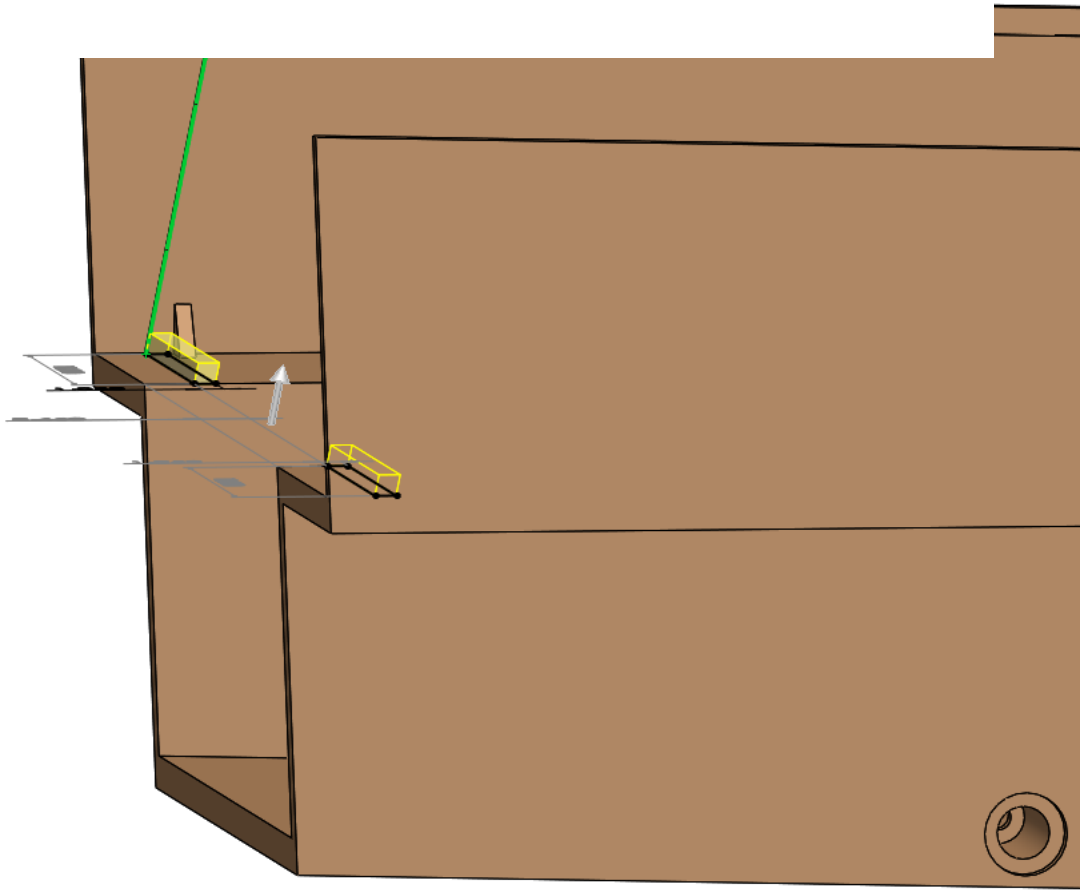
1. Use “Linear Pattern”
2. Select the yellow edge for *Direction 1*
3. Set the distance to **15** mm
4. Set the number of instances to 8
5. Select the cylindrical body on the other side
6. Click Ok

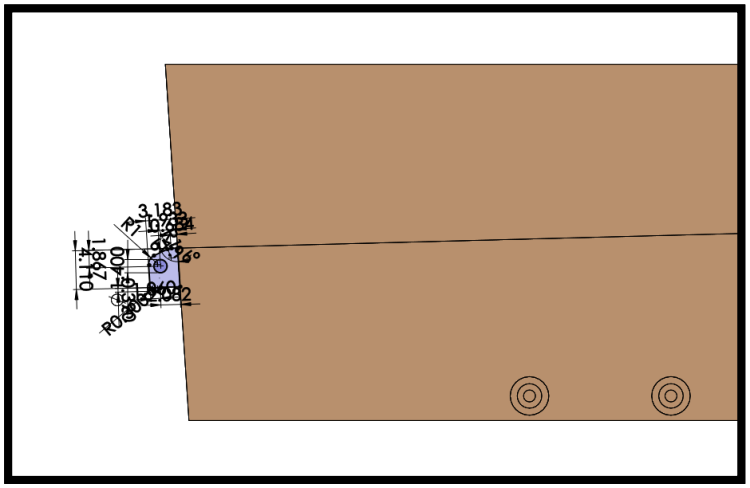


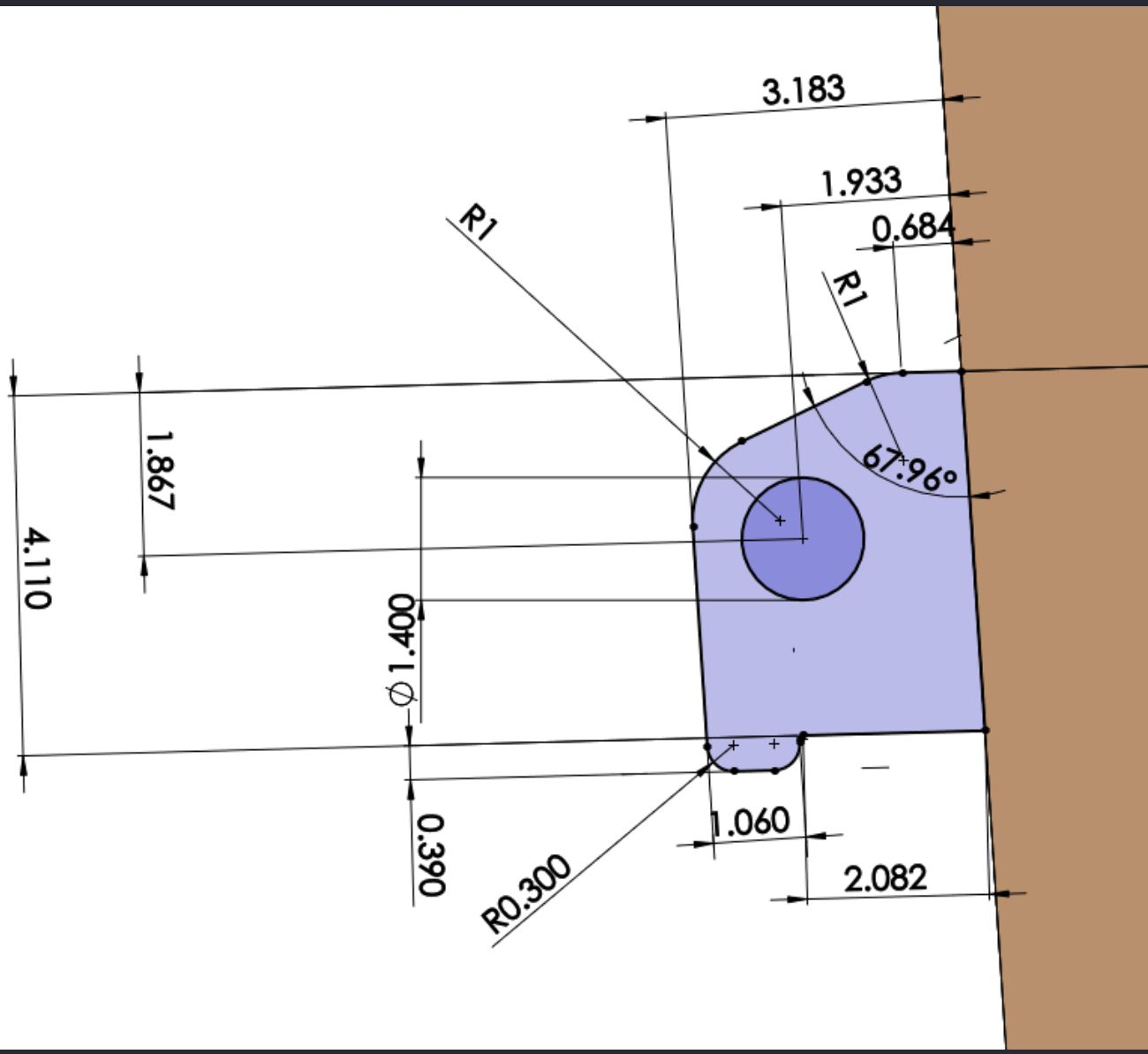
Draw the sketch on the top surface of the part and set its dimensions.



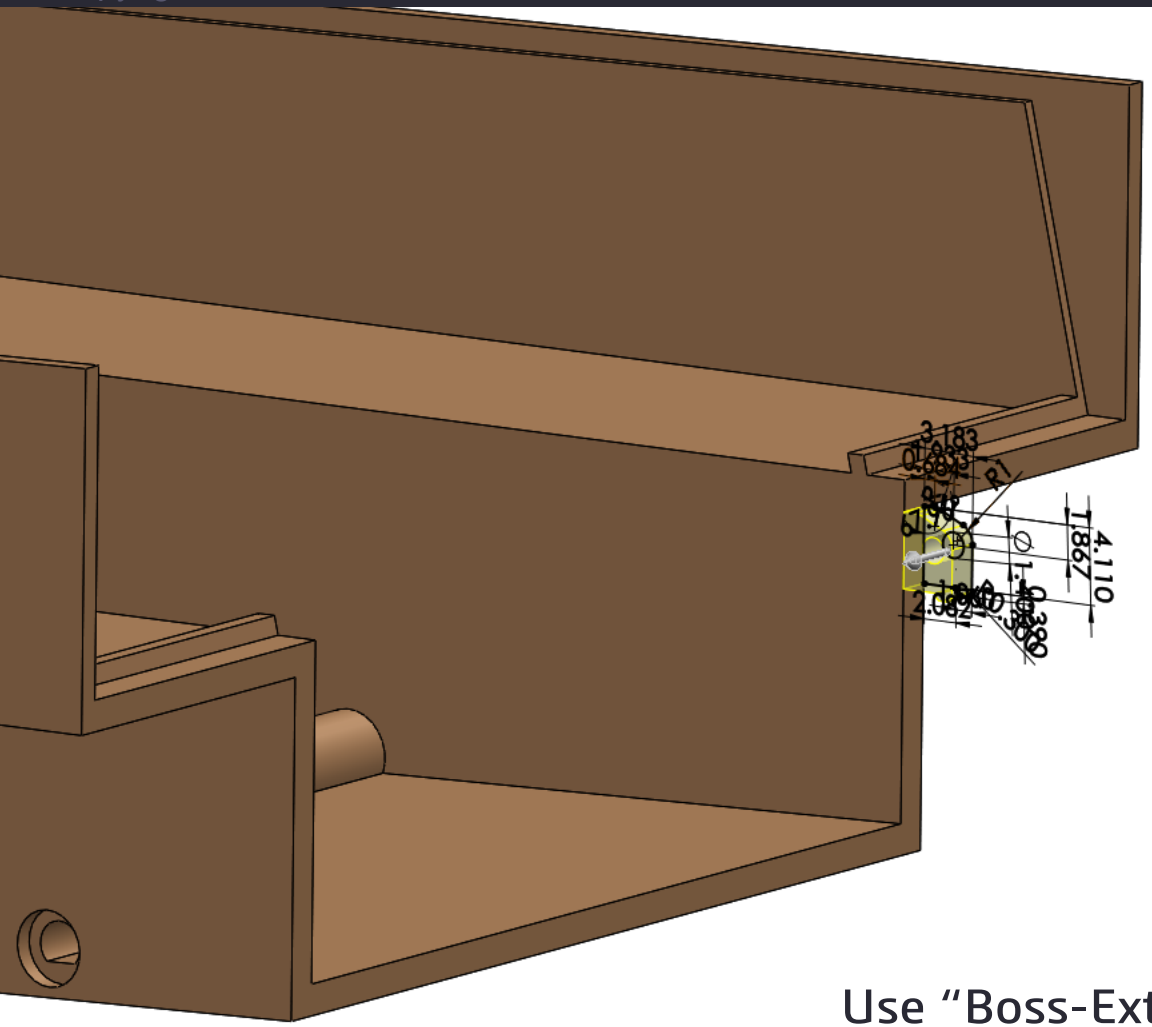
1. Use “Boss-Extrude”
2. Set it to *Blind* at **1.05** mm
3. Select the green highlighted edge
4. Click Ok

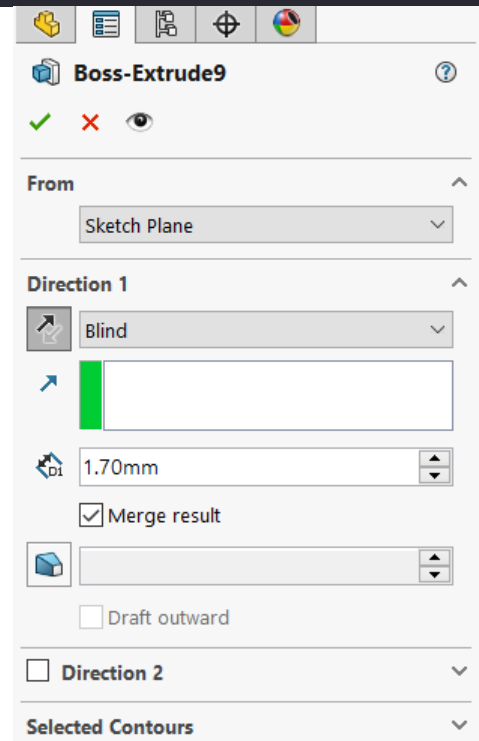






Draw the sketch on the side surface of the part and set its dimensions.

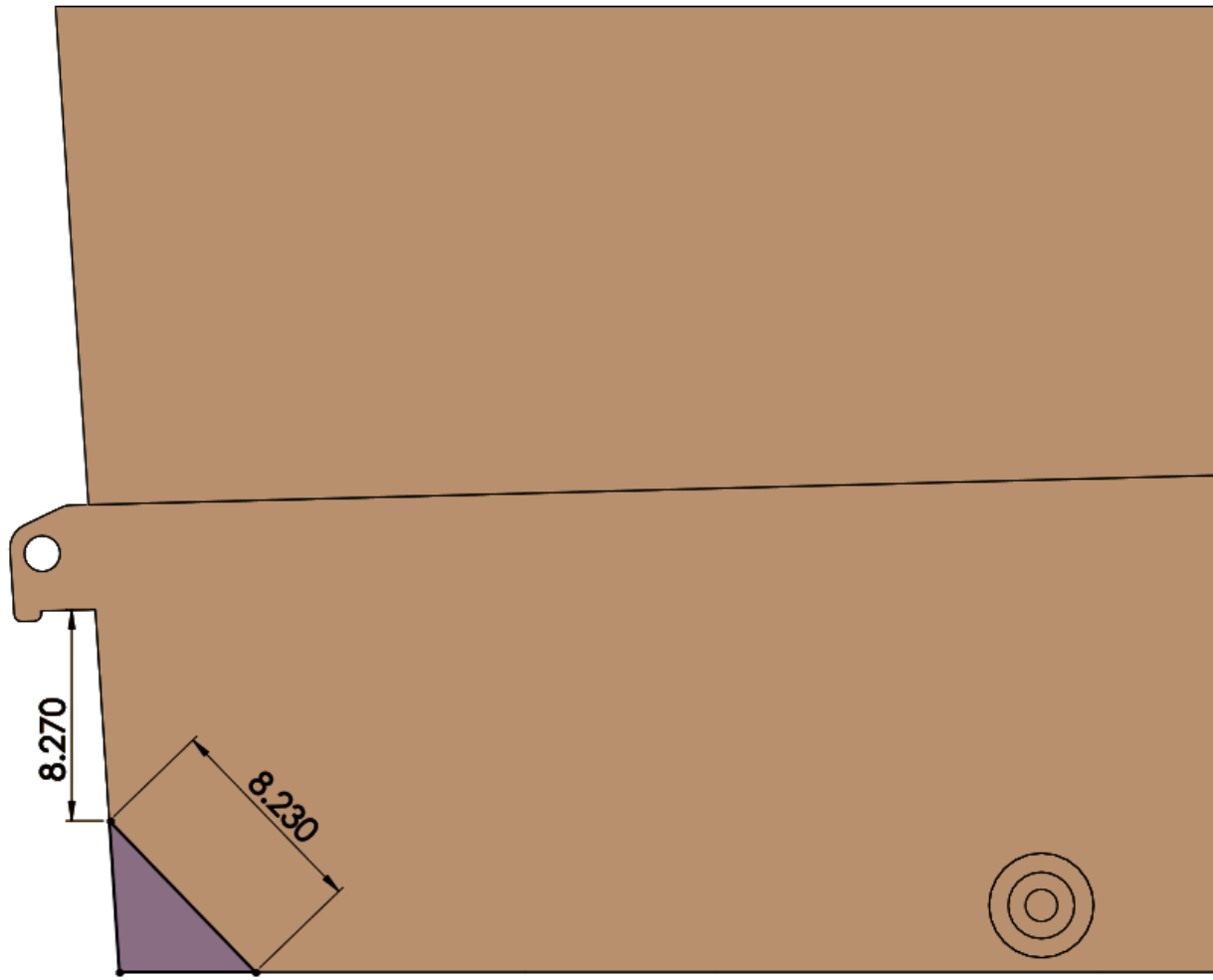


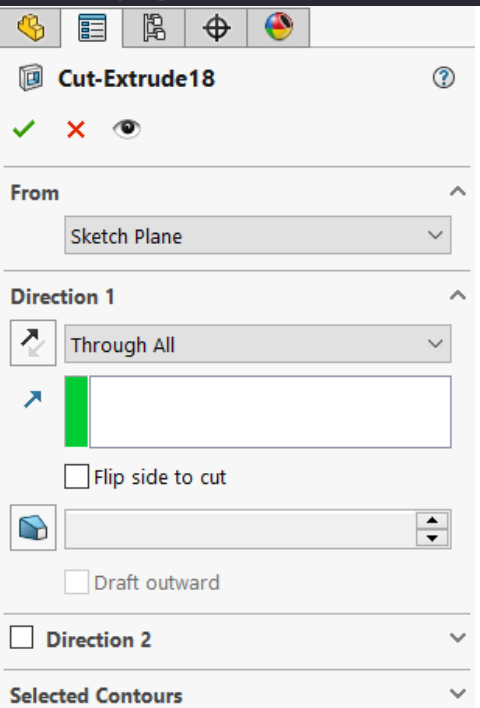


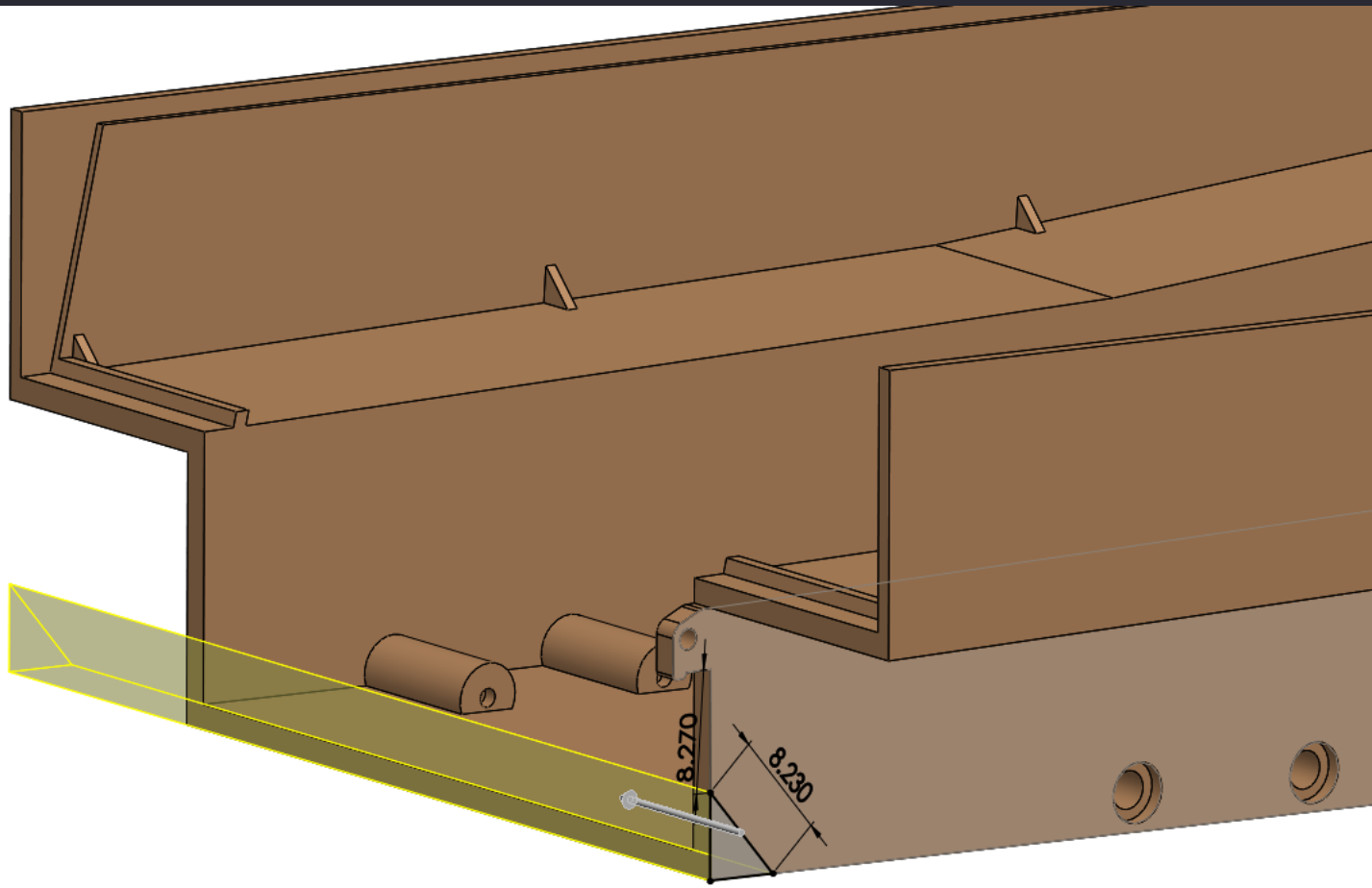
Use “Boss-Extrude” Set it to *Blind* at **1.7** mm and extrude it toward the inner side.

Draw the sketch on the surface of the part and set its dimensions.

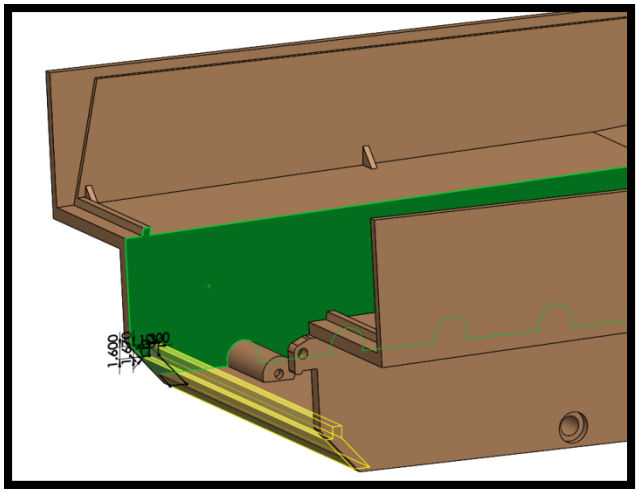
The corners are coincident with the edges of the part.

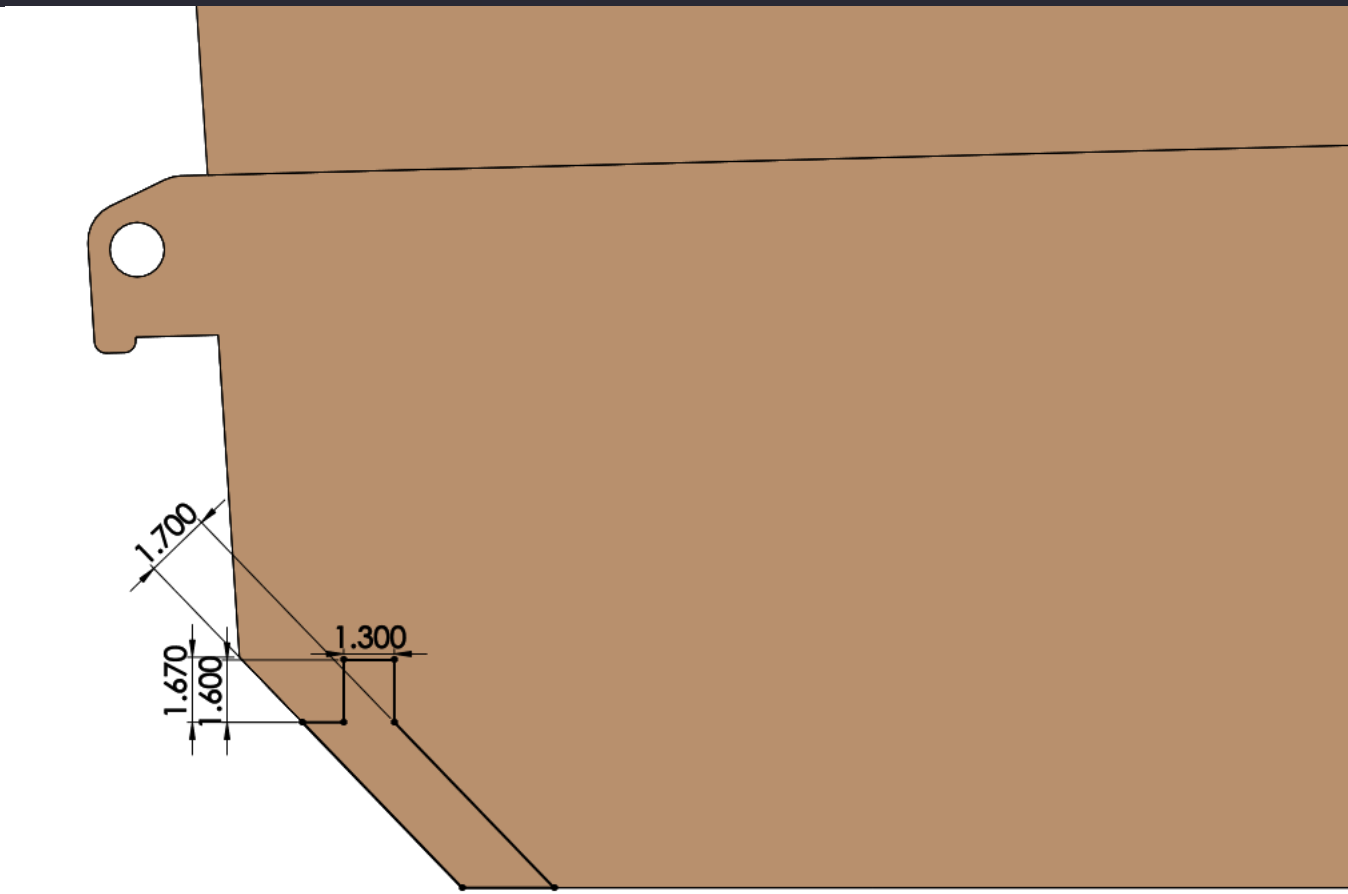




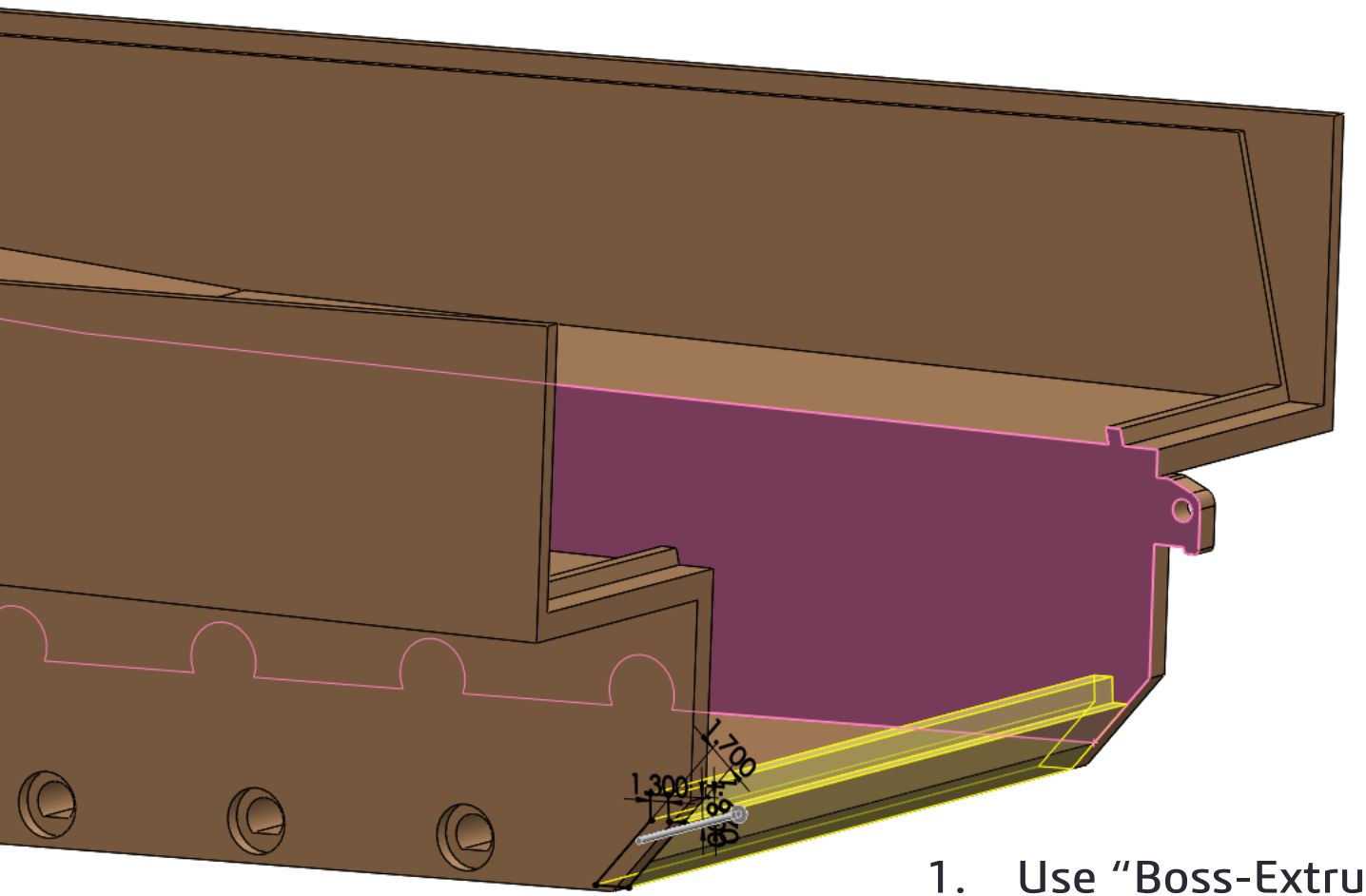


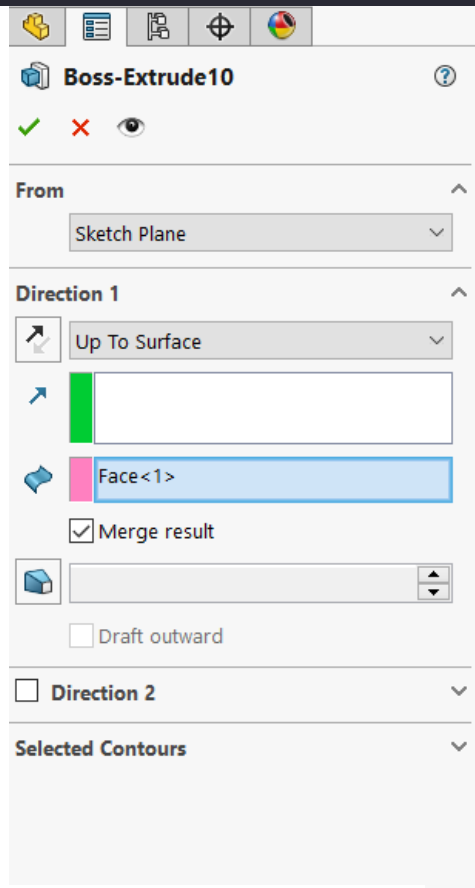
Use "Cut-Extrude", set it to *Through-All* and cut it through the part.





Draw the sketch on the surface of the part and set its dimensions. The corners are **coincident** with the edges of the part.

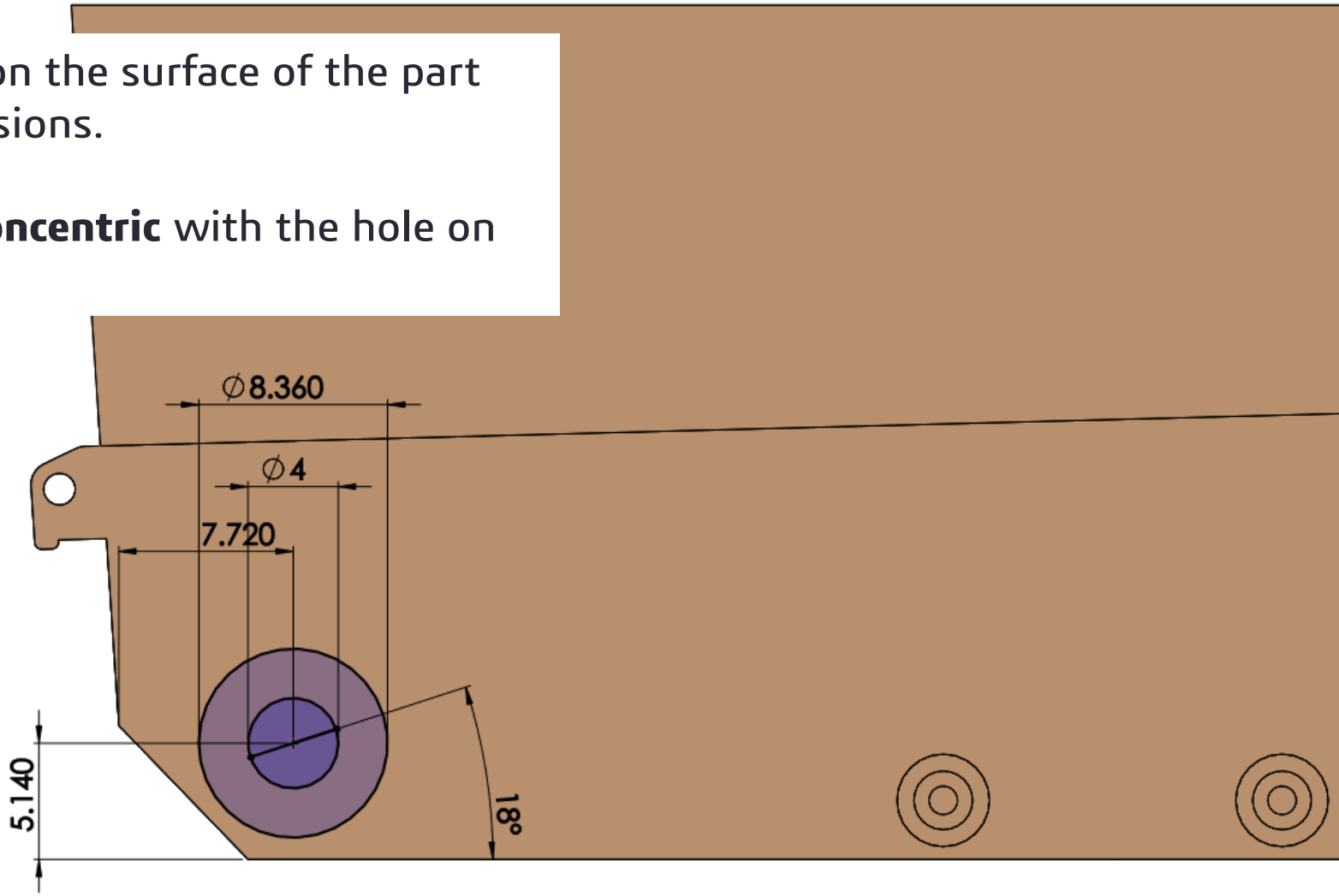


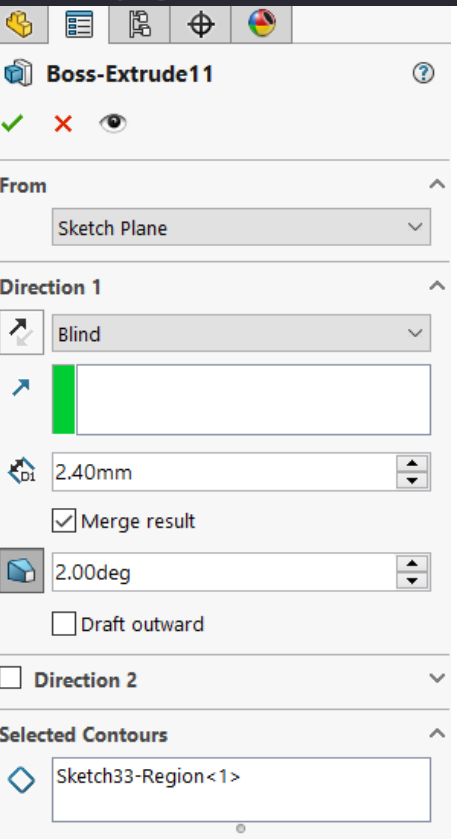


1. Use “Boss-Extrude”
2. Set it to *Up to Surface*
3. Select the Pink surface as the reference
4. Click Ok

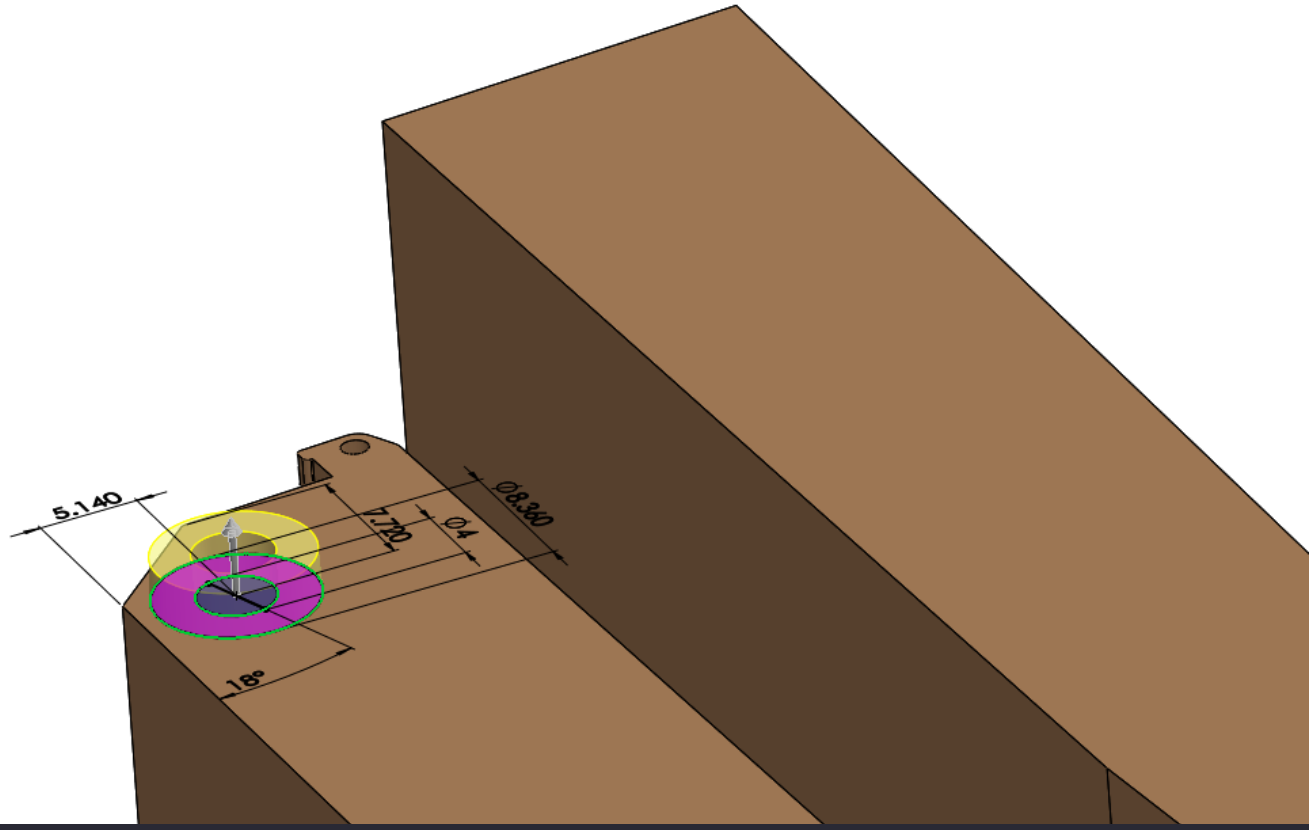
Draw the sketch on the surface of the part and set its dimensions.

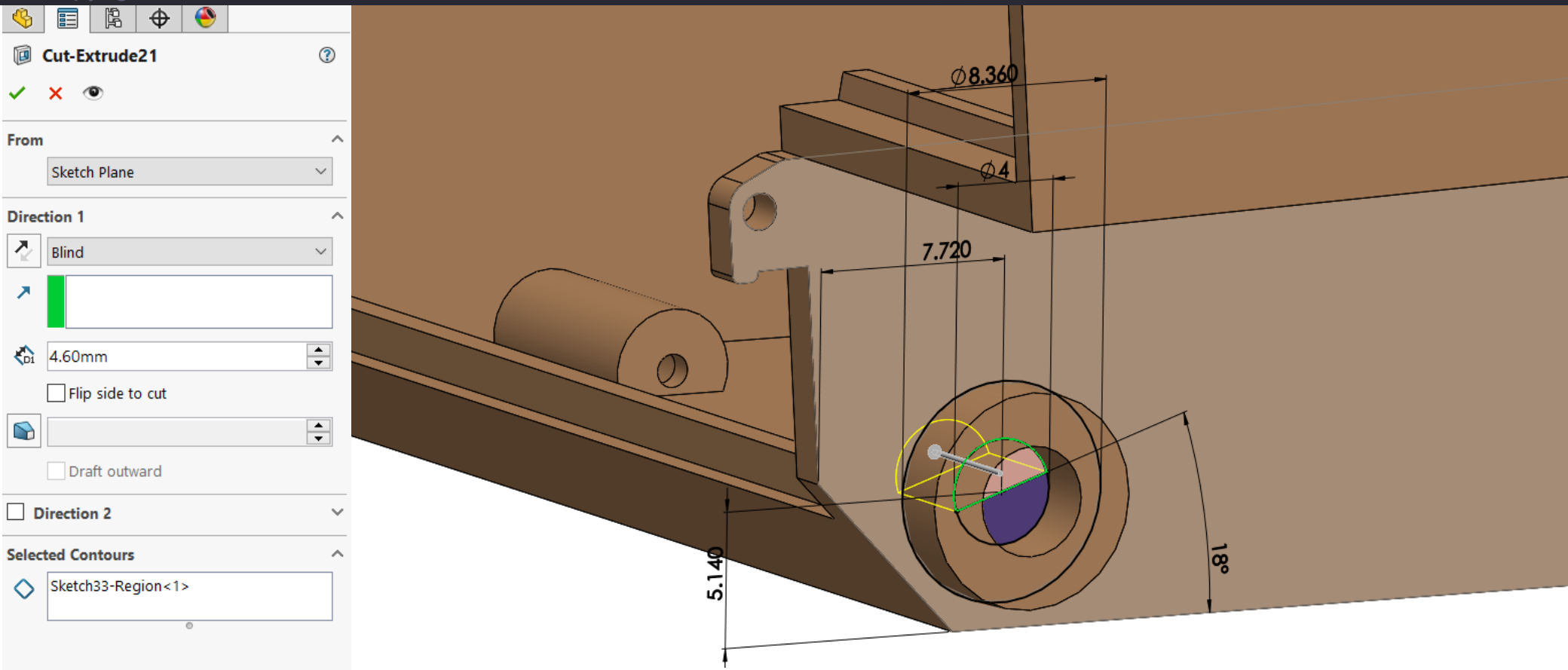
Make the circle **concentric** with the hole on the part.



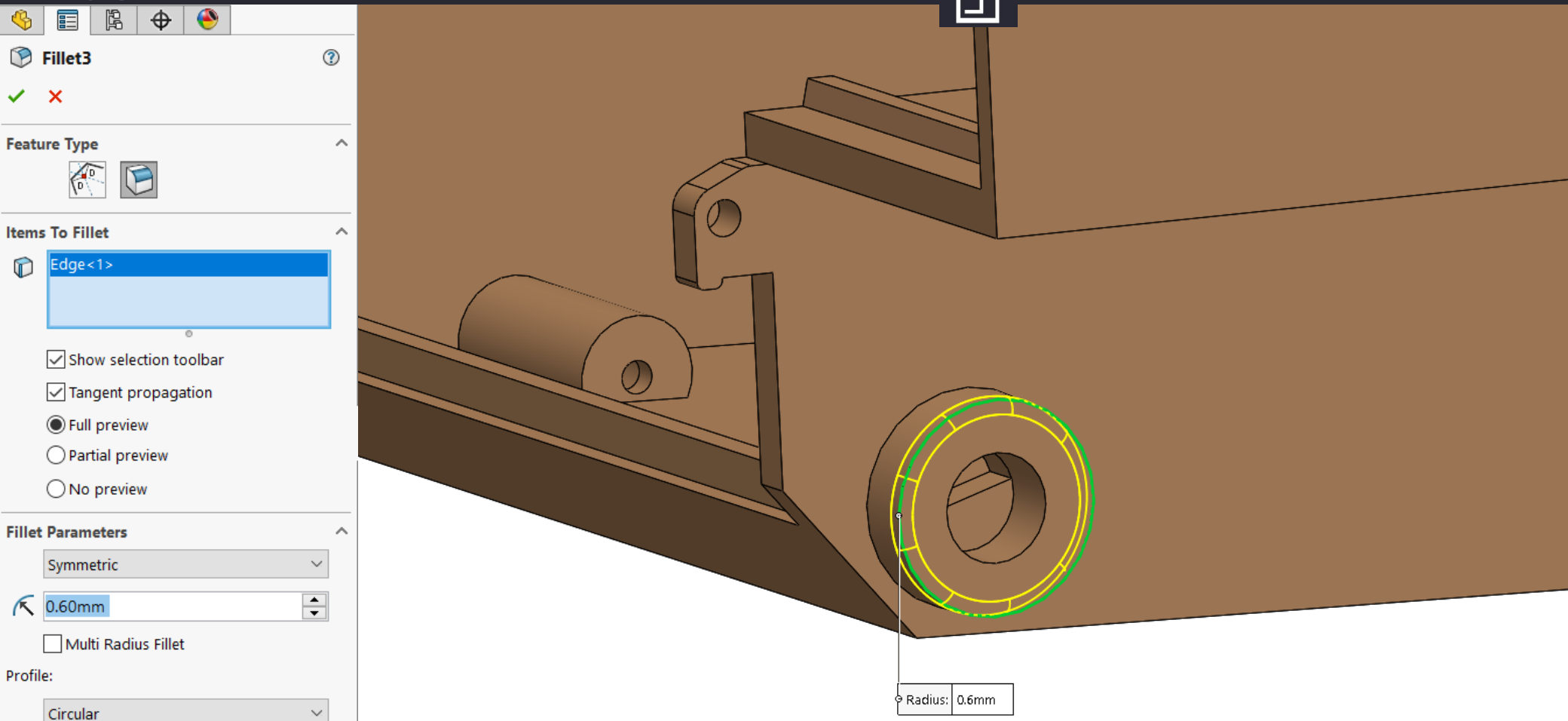


1. Use “Boss-Extrude”
2. Set it to *Up to Surface*
3. Select the Pink surface as the reference
4. Click Ok

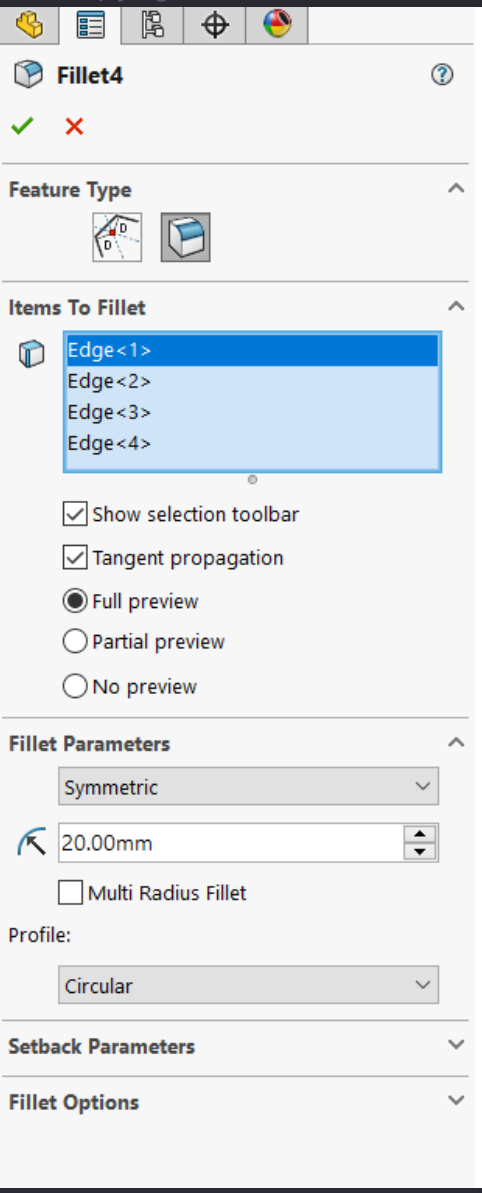


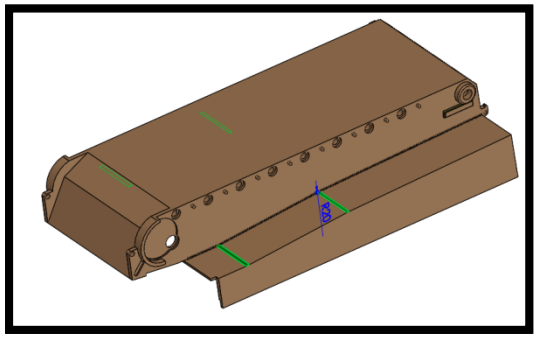


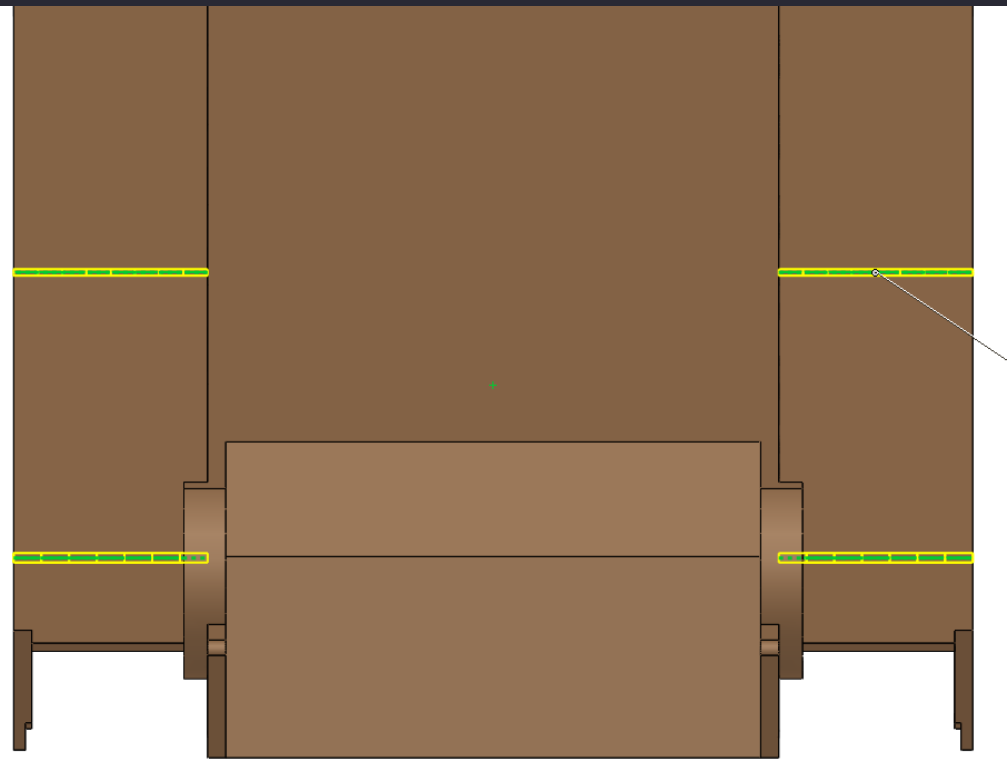
1. Use “Cut-Extrude”
2. Set it to *Blind* at **4.6** mm
3. Select the top half in the Selected Contours
4. Click Ok



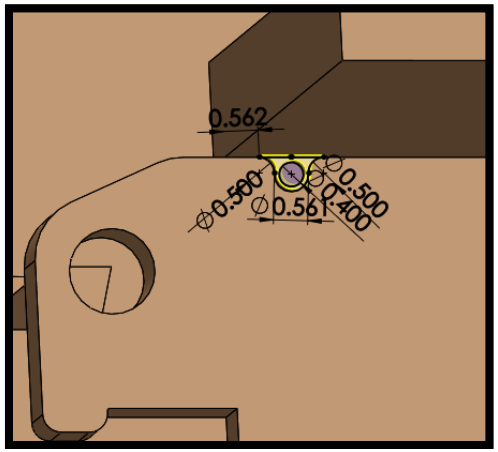
Use “Fillet” and round the highlighted edge with the radius of **0.6** mm







Use “Fillet” and round the highlighted edges with the radius of **20** mm



Draw the sketch on the side surface of the part and set its dimensions.
Make the end points coincident to the edges.

Use “Boss-Extrude” and extrude the sketch for **0.4** mm

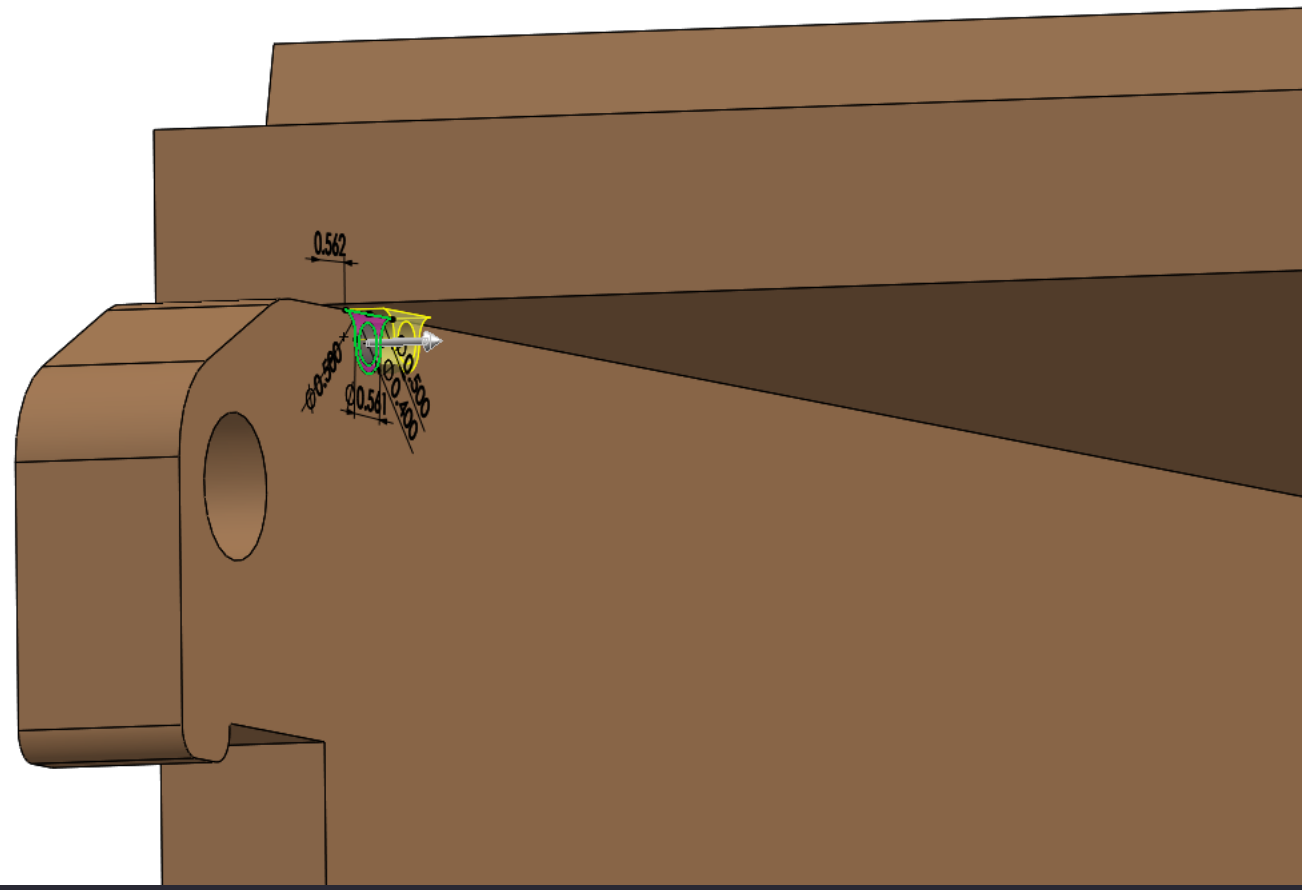
Boss-Extrude12

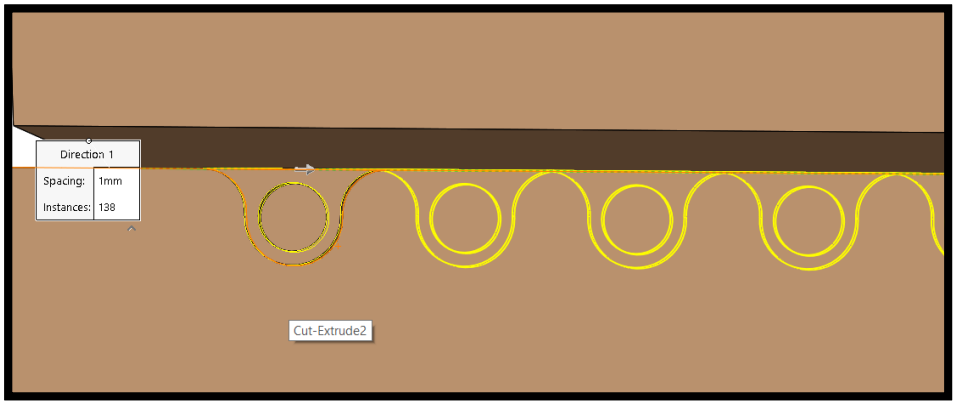
From
Sketch Plane

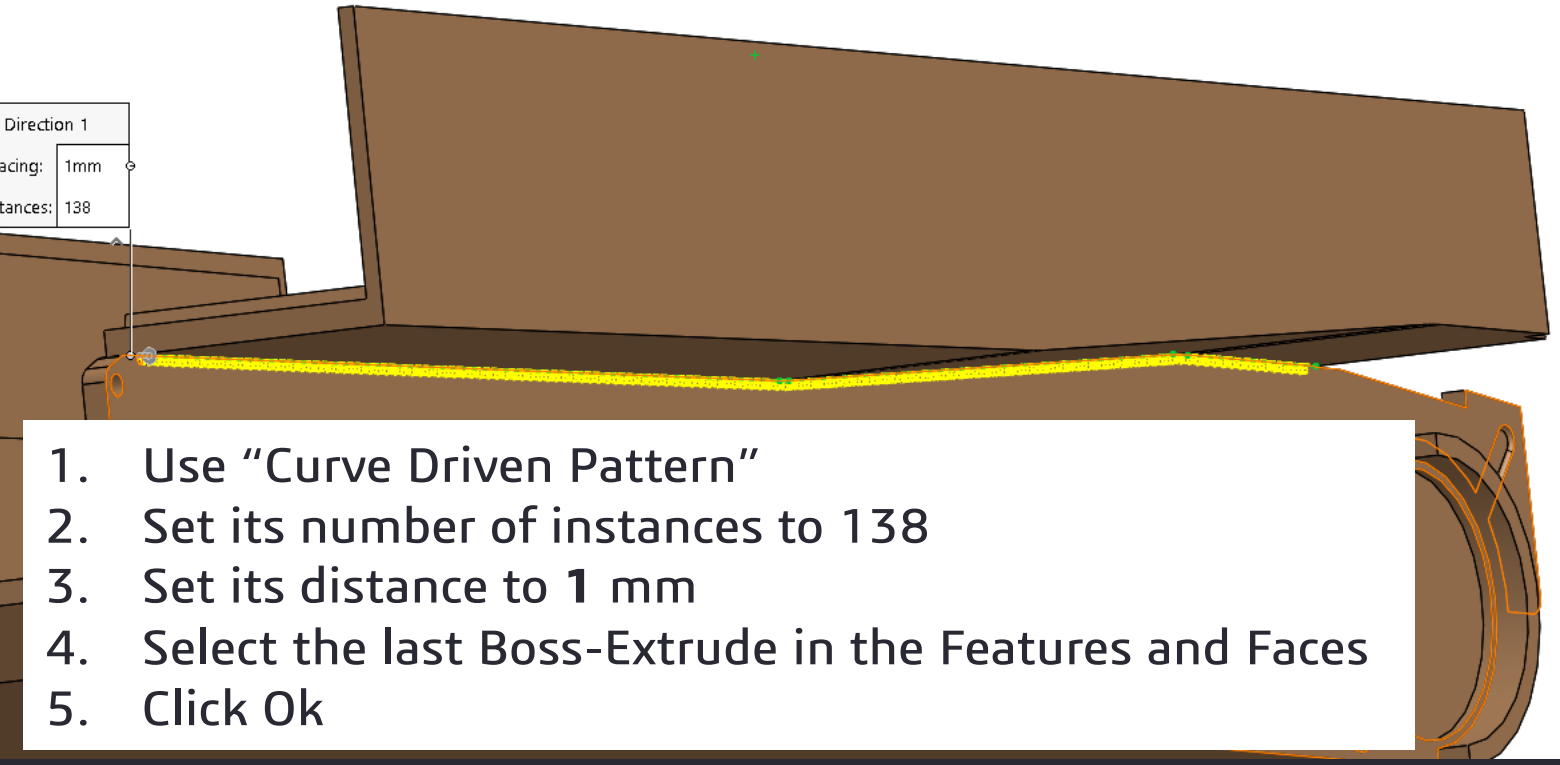
Direction 1
Blind
0.40mm
Merge result
Draft outward

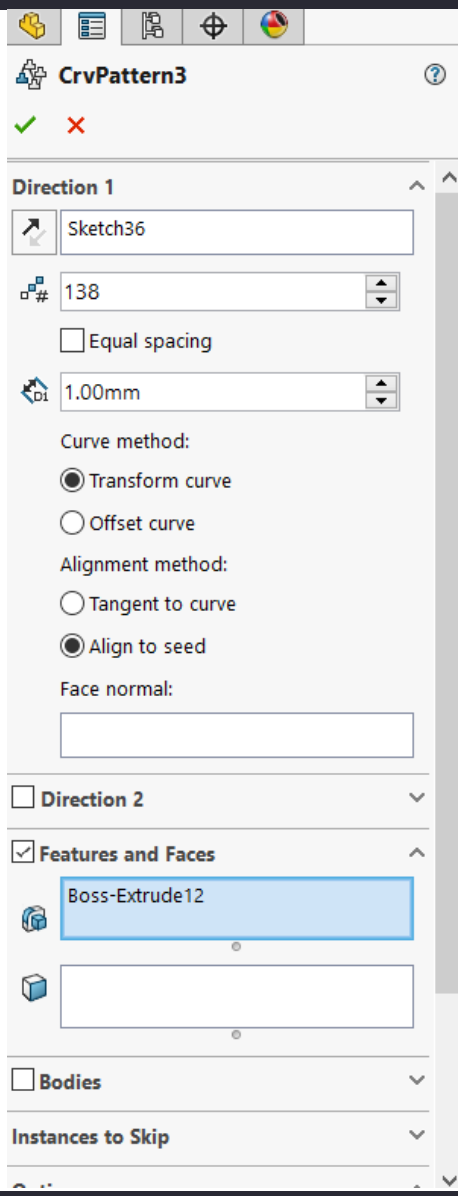
Direction 2

Selected Contours
Sketch37-Region<1>

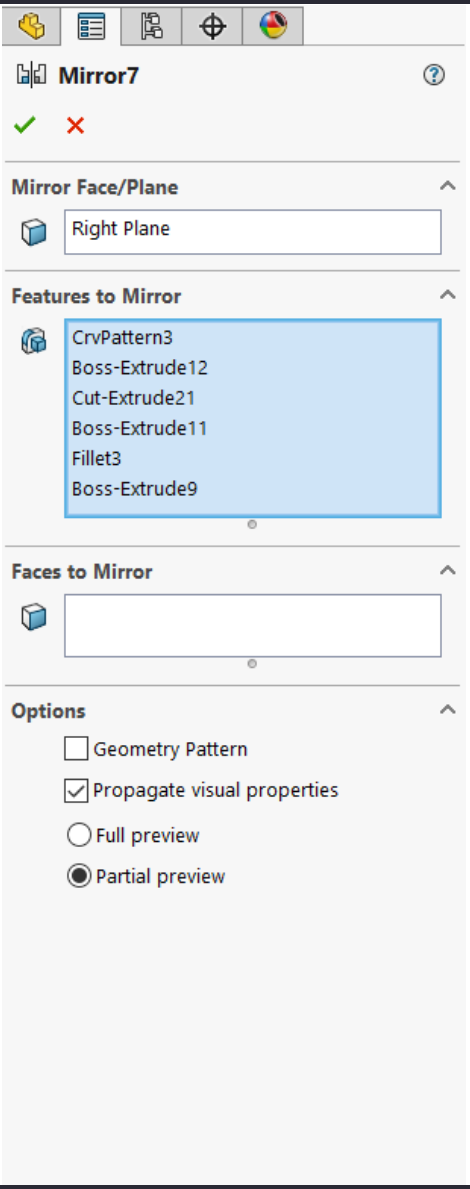




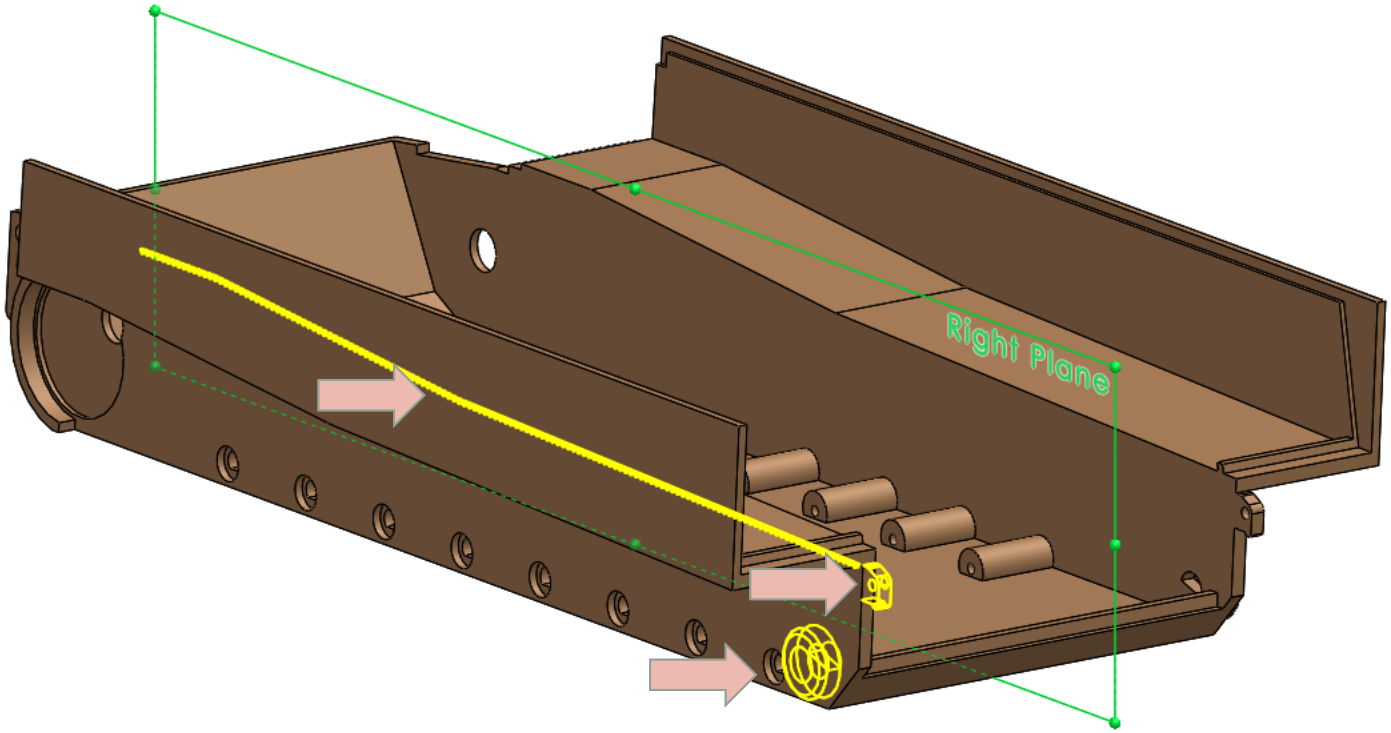




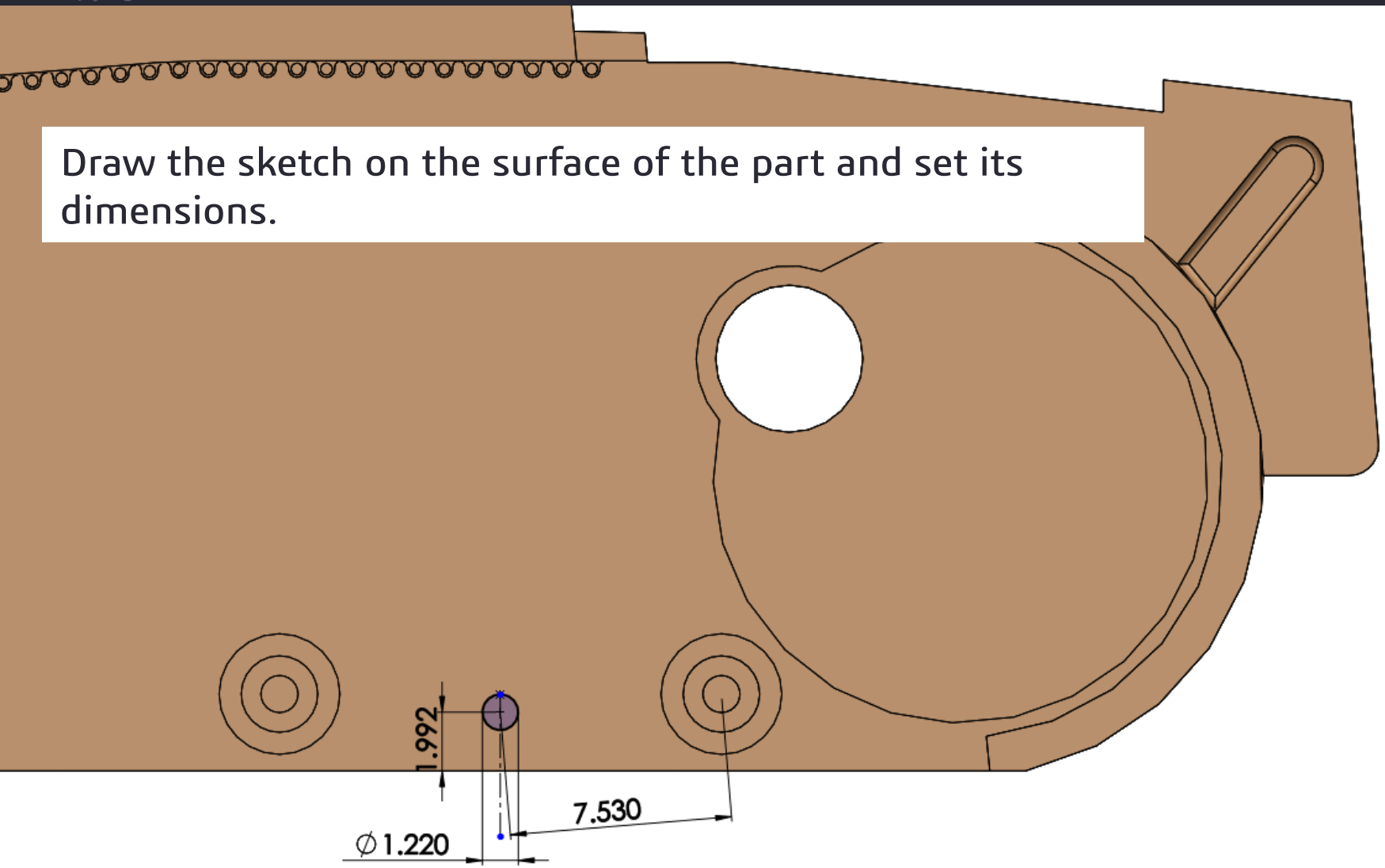
1. Use “Curve Driven Pattern”
2. Set its number of instances to 138
3. Set its distance to **1** mm
4. Select the last Boss-Extrude in the Features and Faces
5. Click Ok



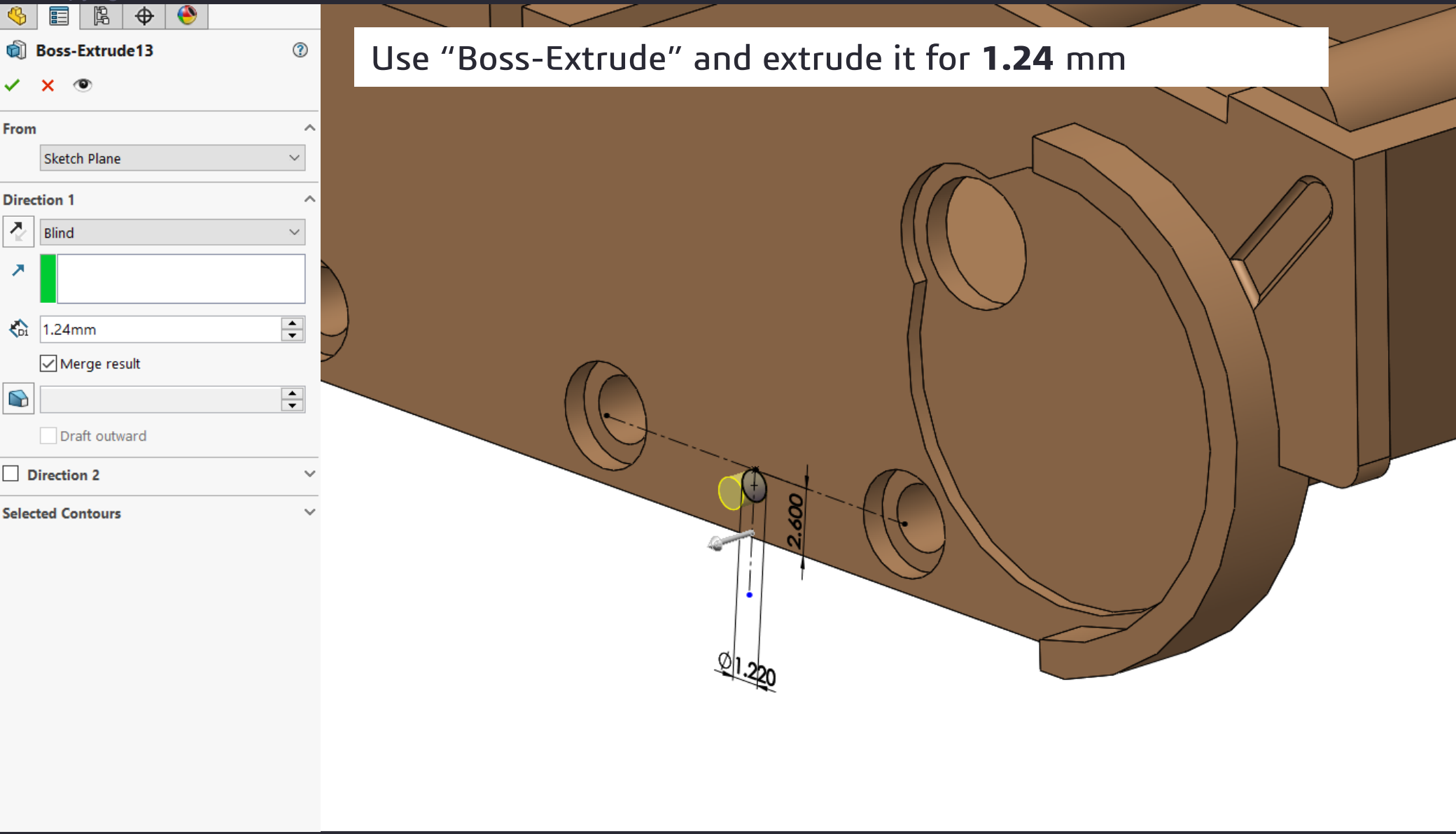
1. Use “Mirror”
2. Select the **Right Plane** as the Mirror Face/Plane
3. Select the highlighted features
4. Click Ok

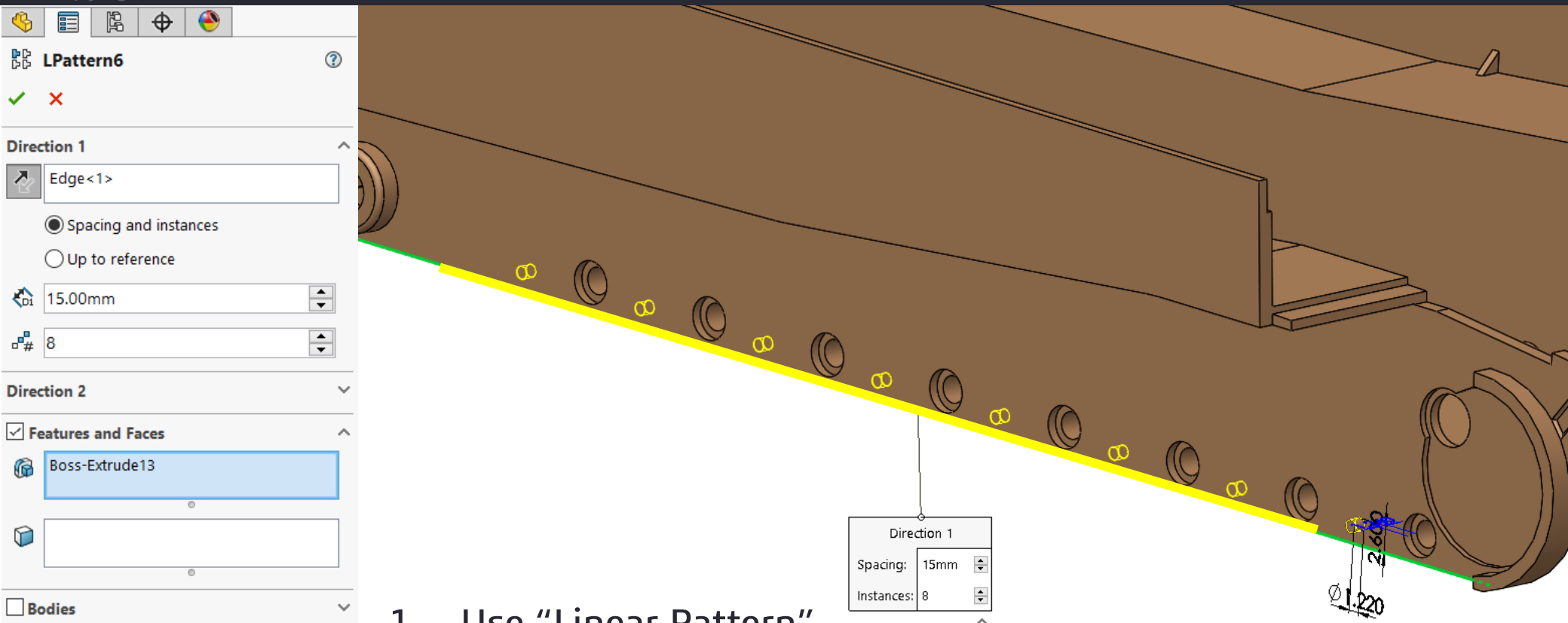


Draw the sketch on the surface of the part and set its dimensions.

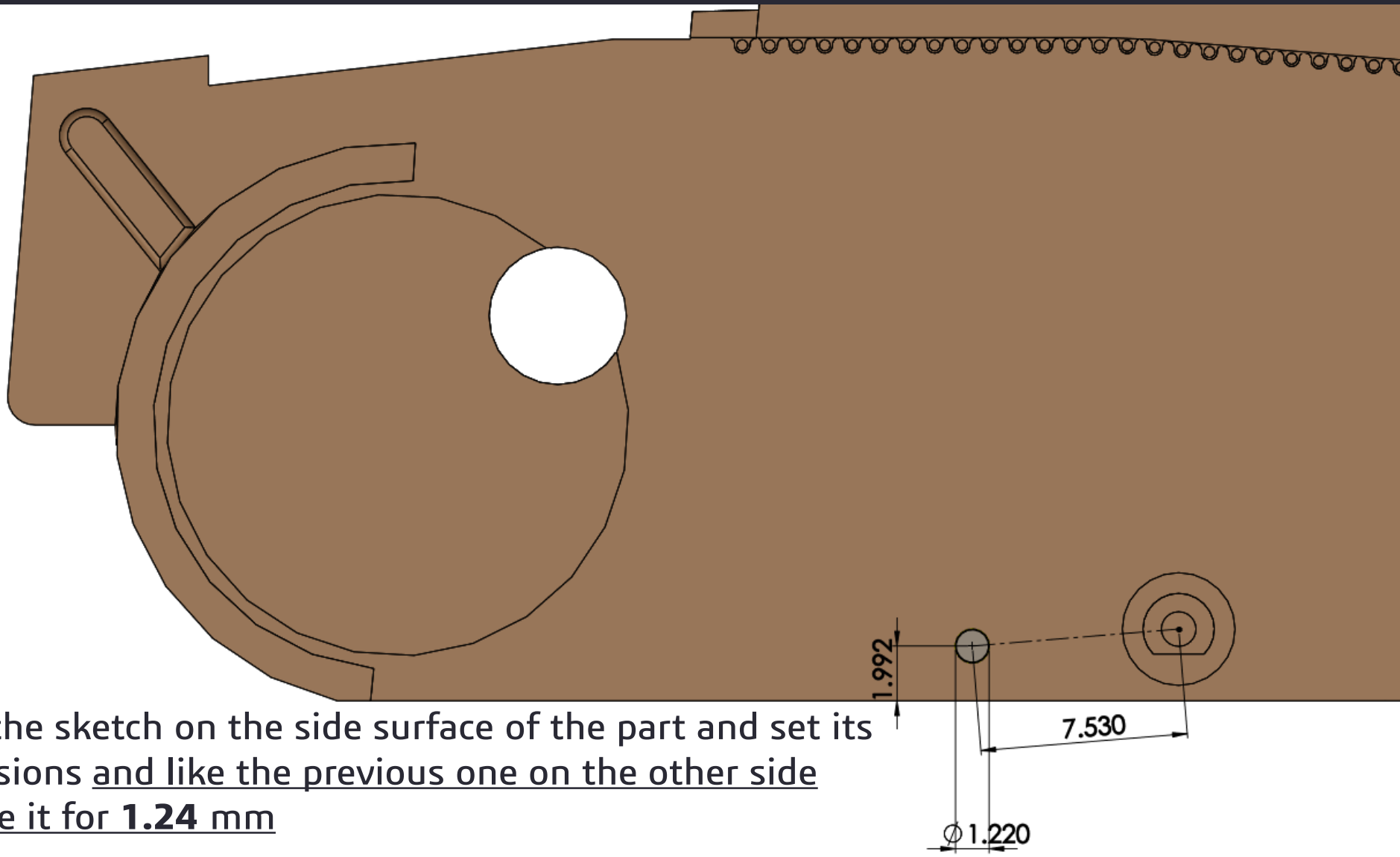


Use “Boss-Extrude” and extrude it for **1.24** mm

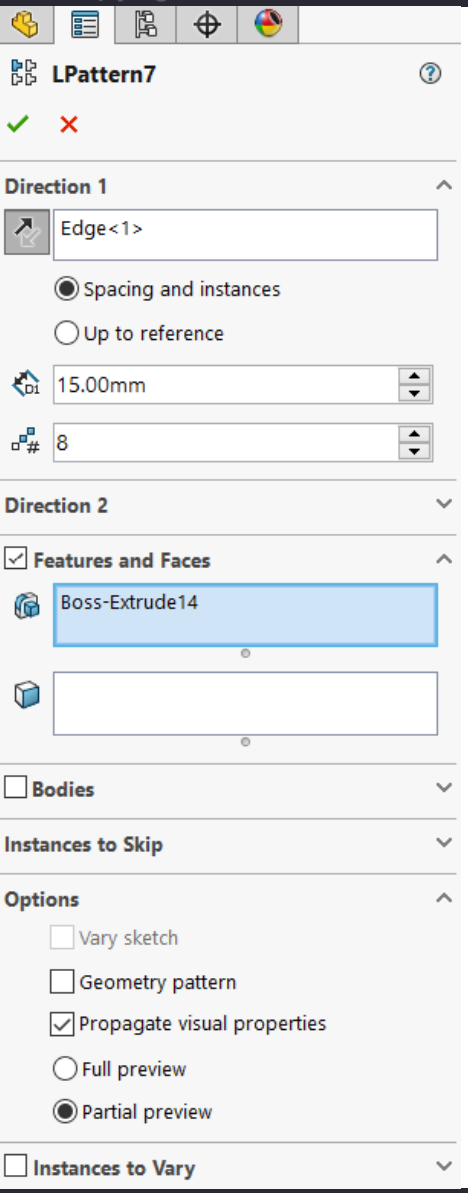


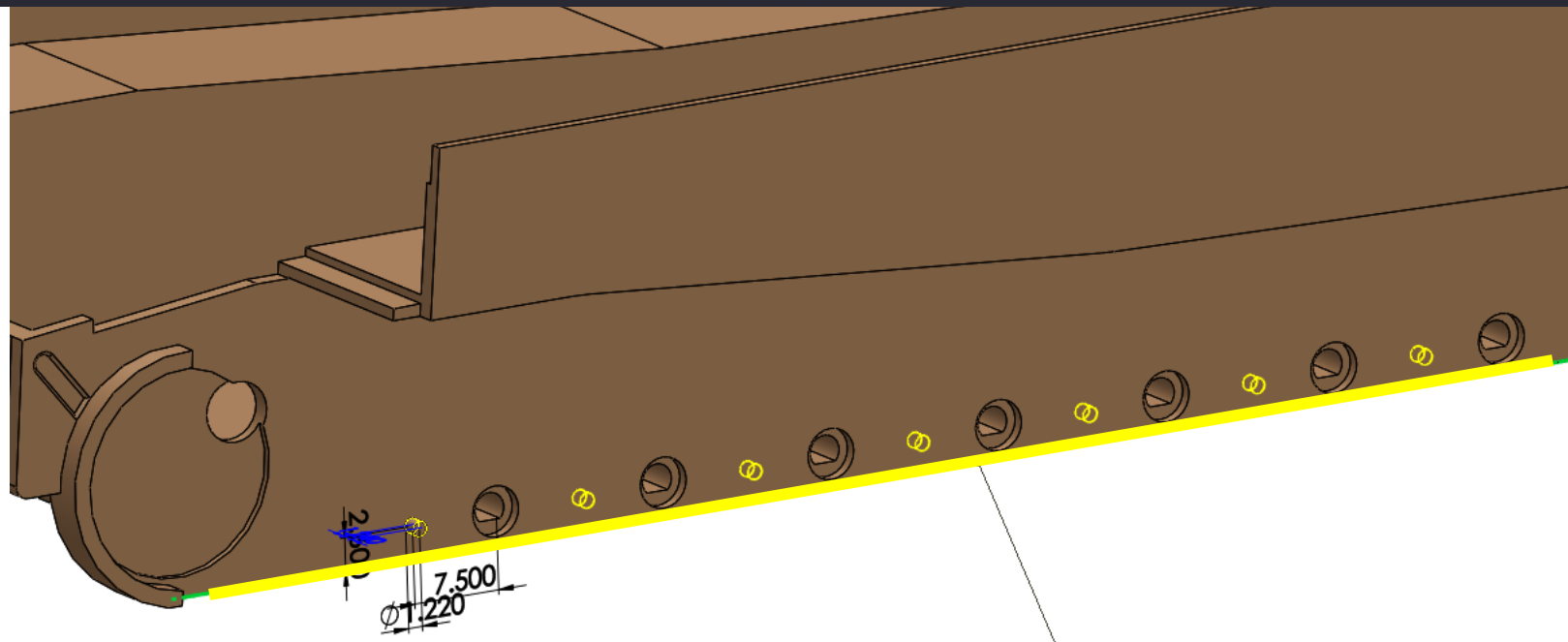


1. Use “Linear Pattern”
2. Select the yellow highlighted edge as *Direction 1*
3. Set the distance to **15** mm
4. Set the number of instances to 8
5. Select the last Boss-Extrude
6. Click Ok

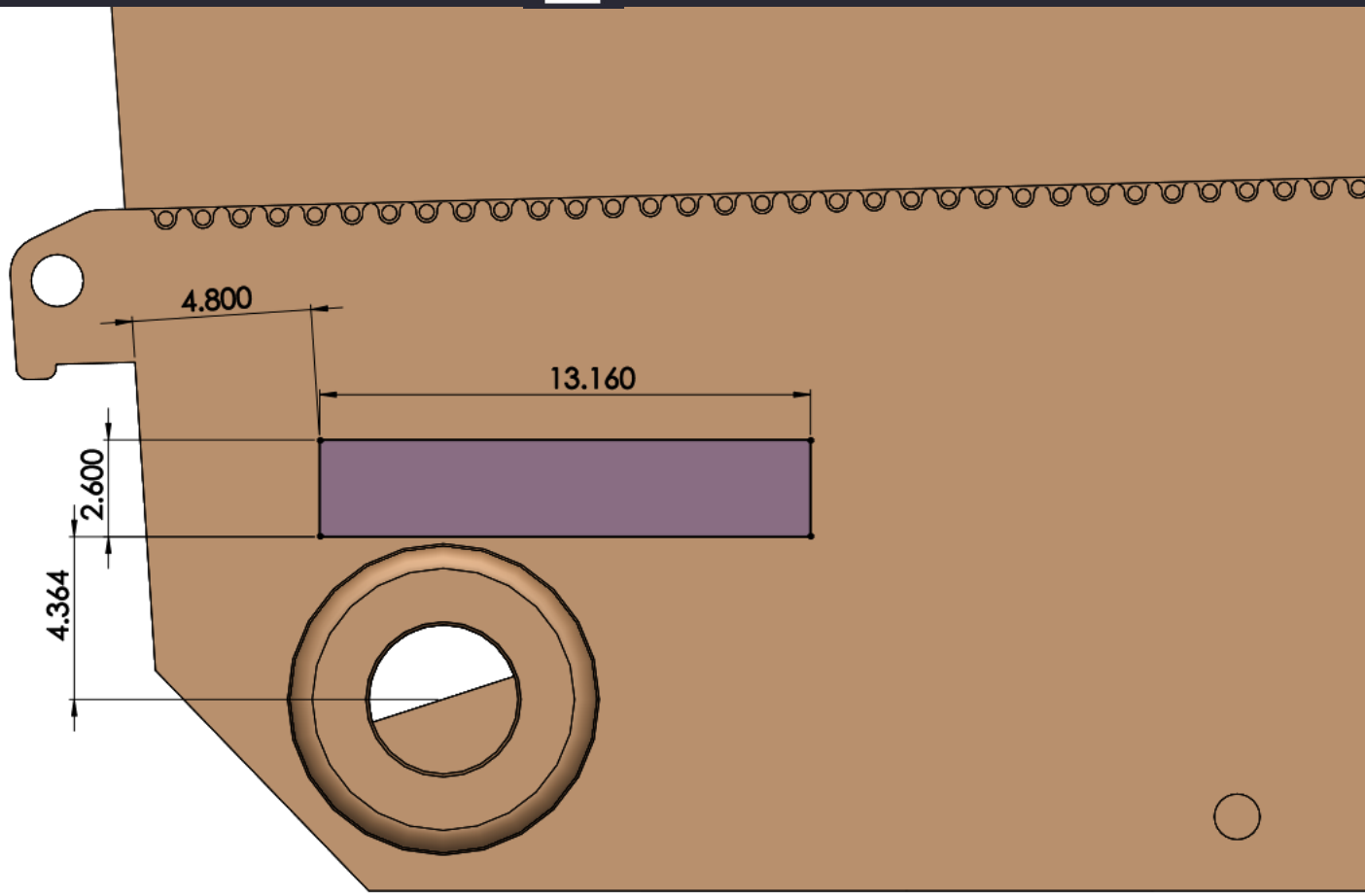


Draw the sketch on the side surface of the part and set its dimensions and like the previous one on the other side extrude it for **1.24** mm

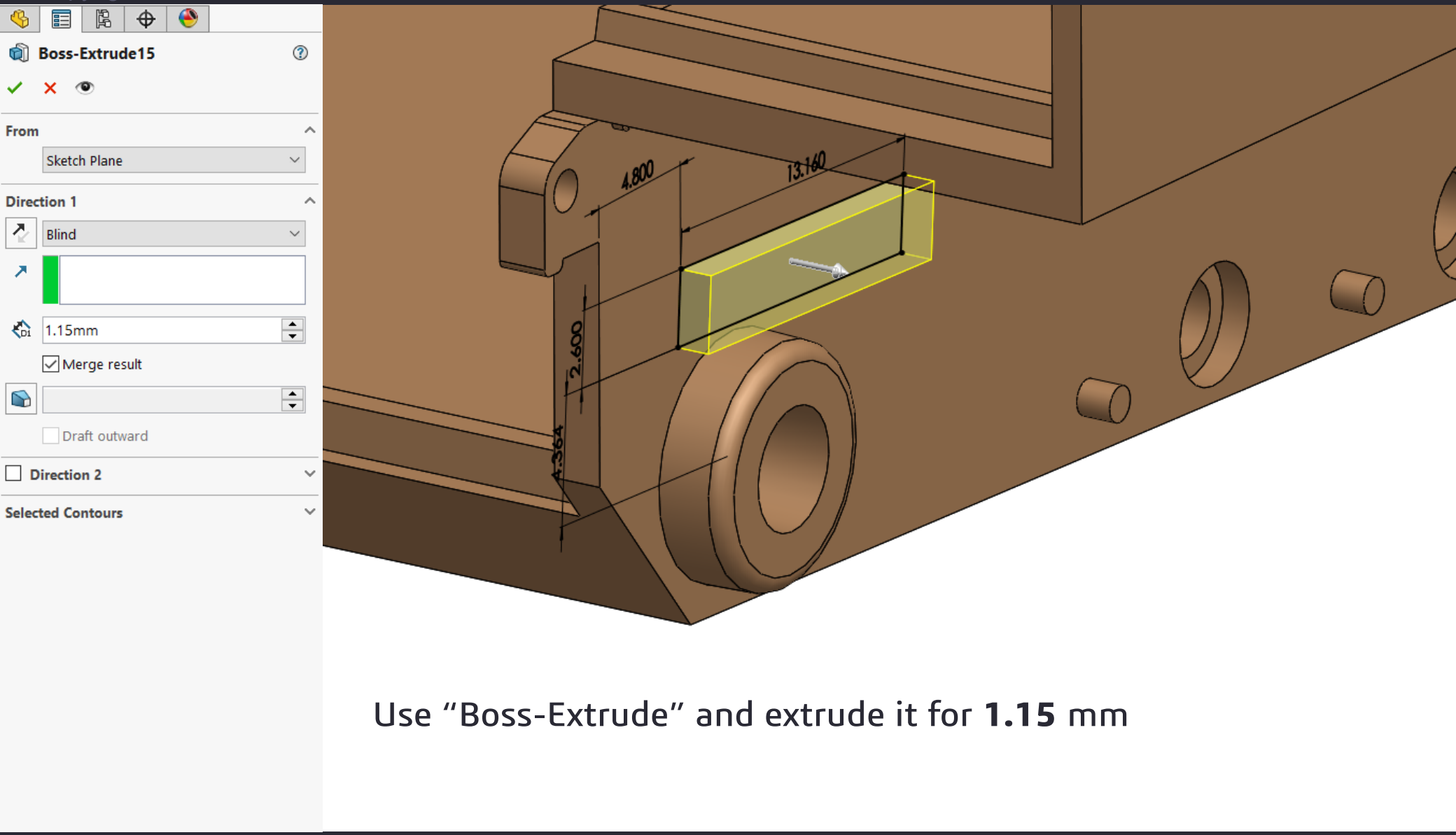




1. Use “Linear Pattern”
2. Select the yellow highlighted edge as *Direction 1*
3. Set the distance to **15** mm
4. Set the number of instances to 8
5. Select the last Boss-Extrude
6. Click Ok

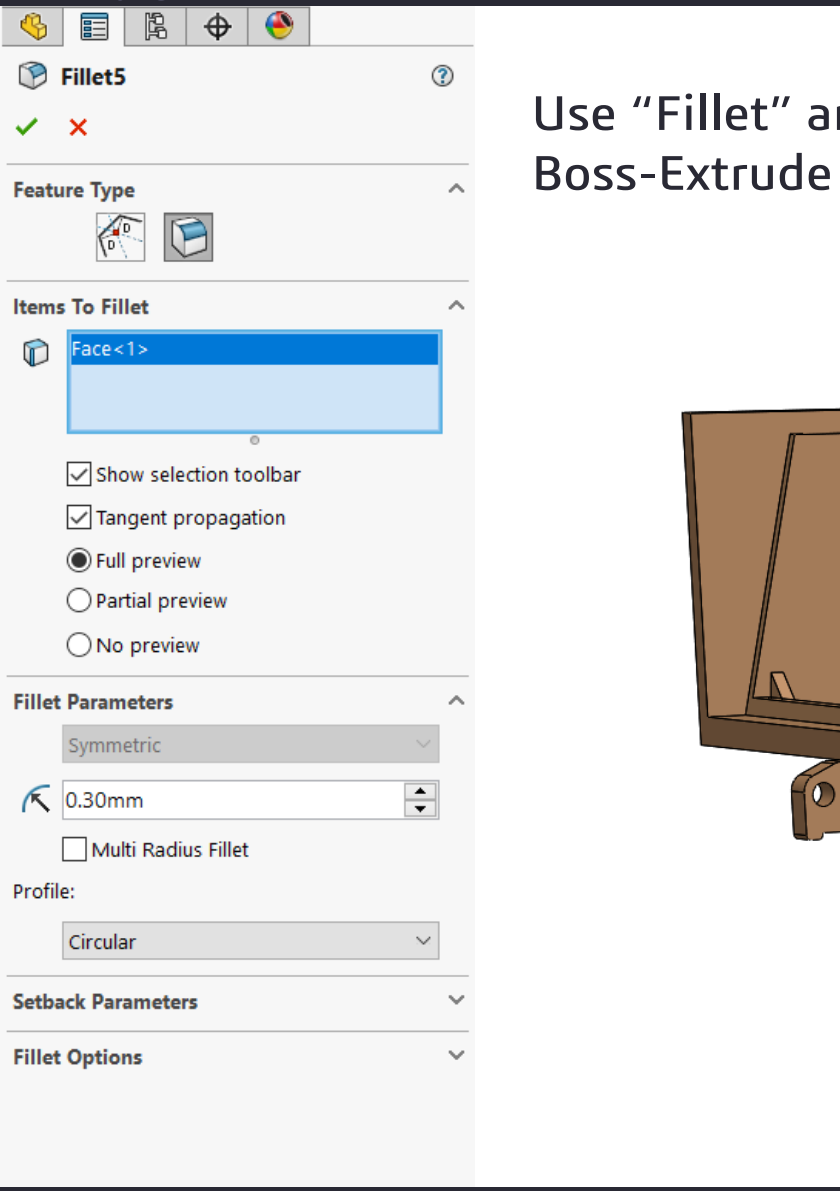


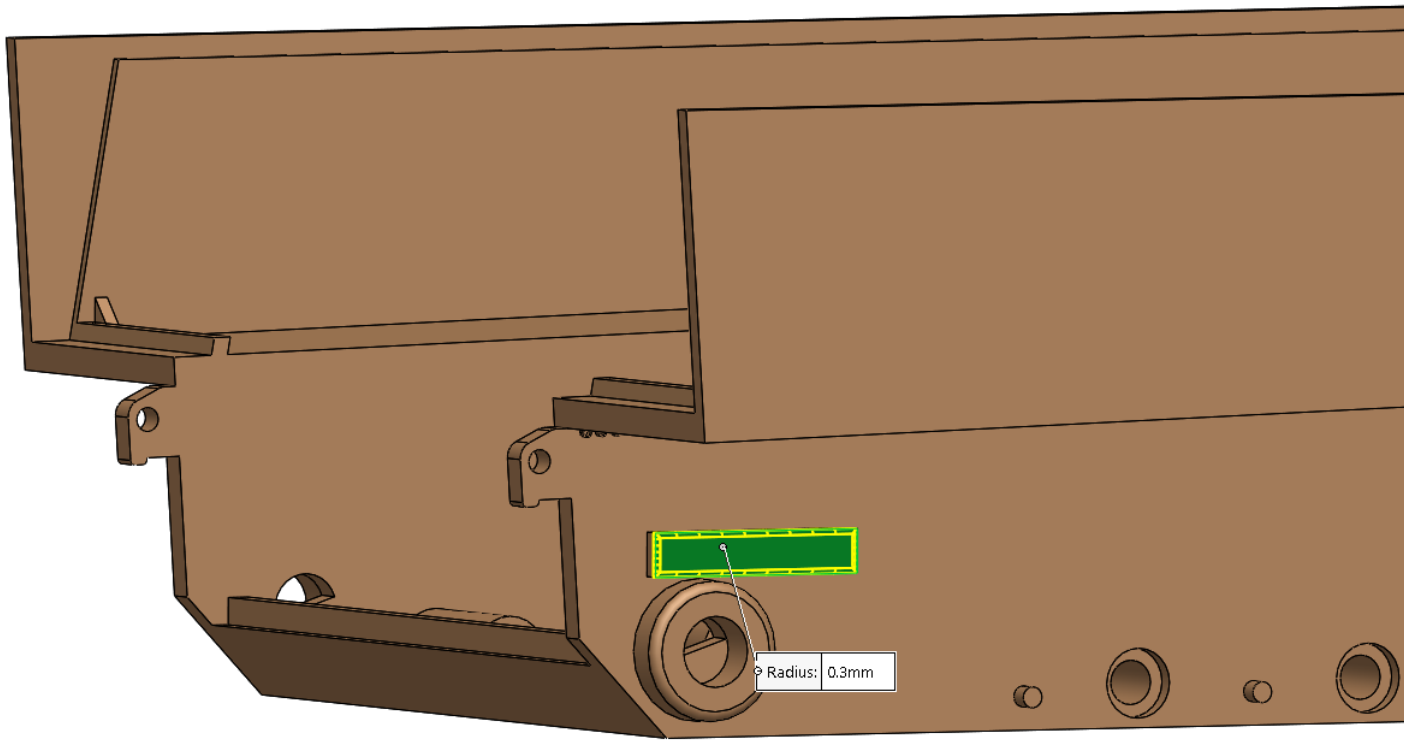
Draw the sketch on the side surface of the part and set its dimensions

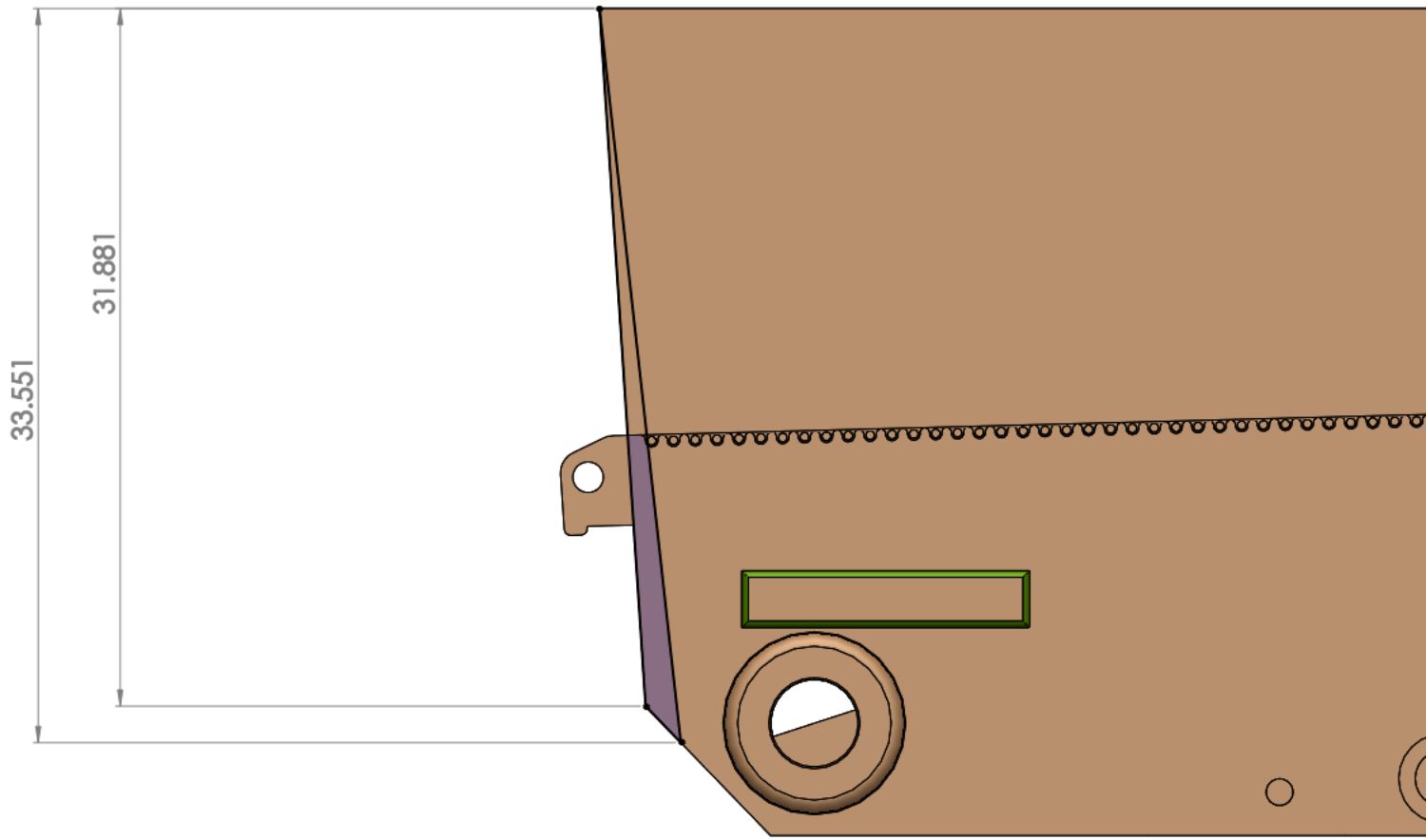


Use “Boss-Extrude” and extrude it for **1.15** mm

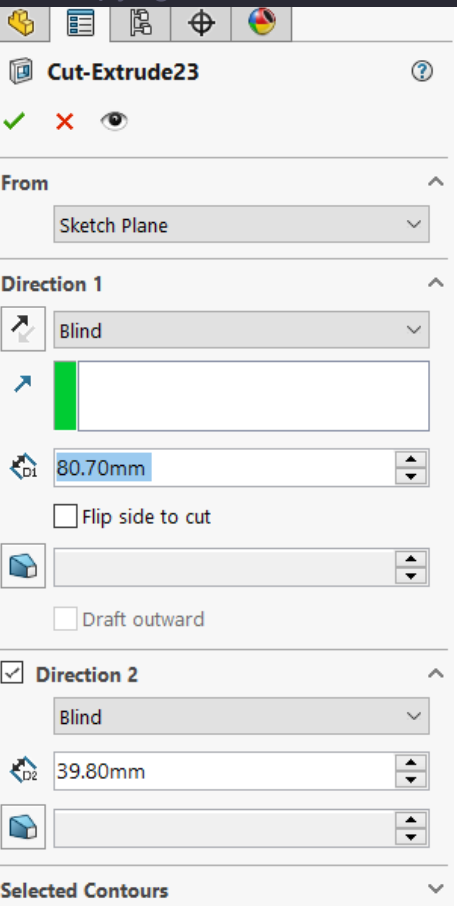
Use “Fillet” and round the edges of the previously created Boss-Extrude for **0.3** mm

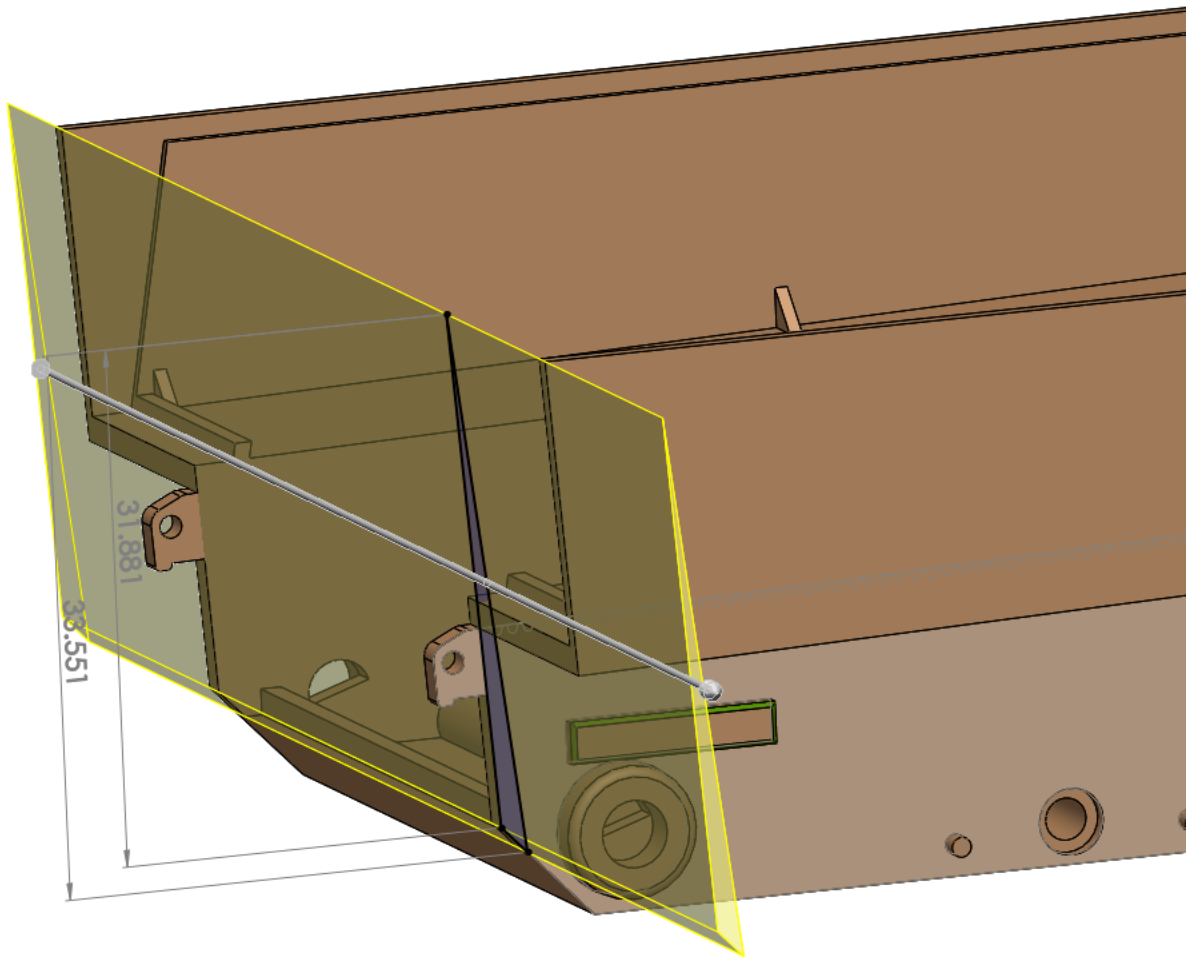




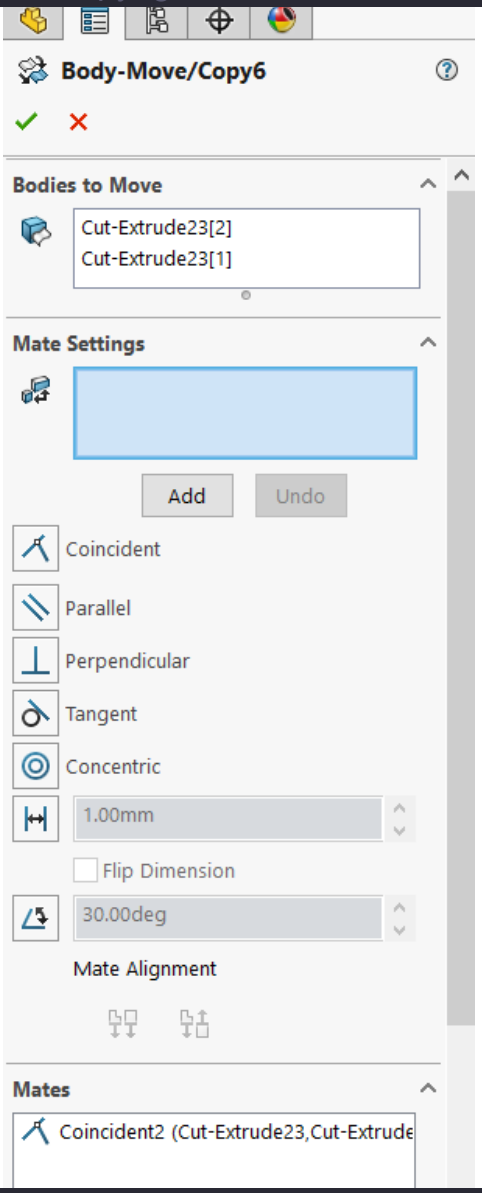


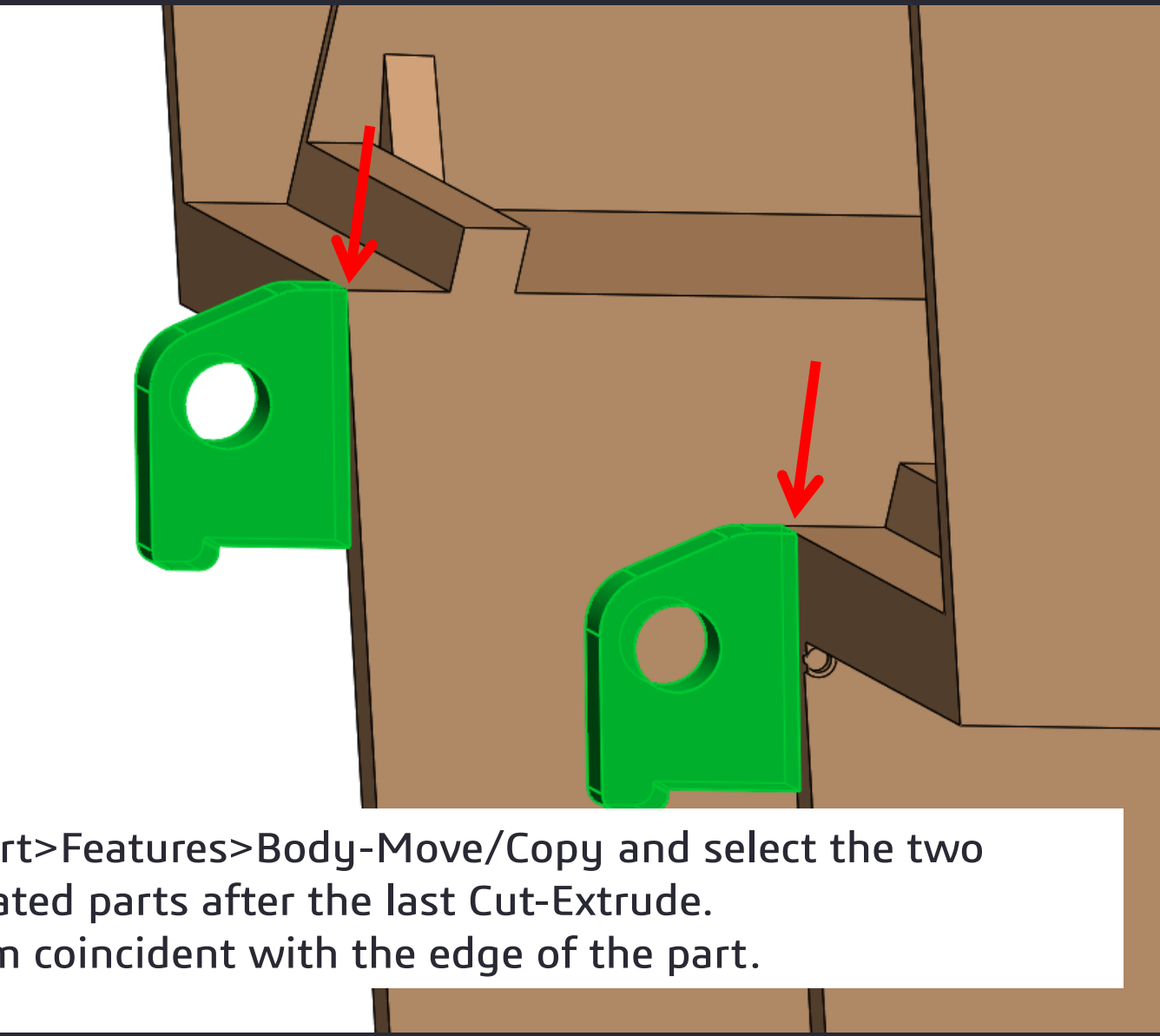
Draw the sketch (Triangle) and on the side surface of the part and set its dimensions . Make sure to make its end point **coincident** with the edges of the part.



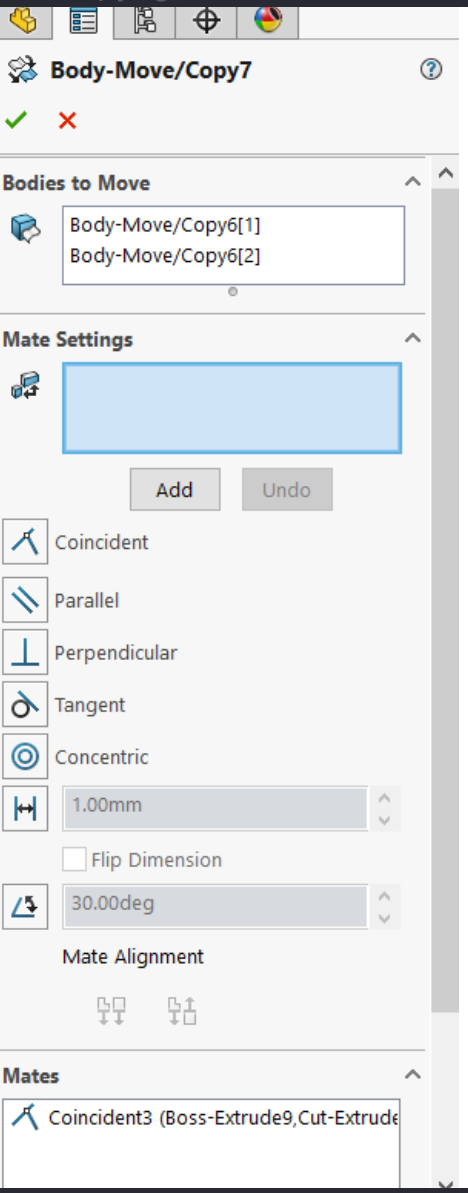


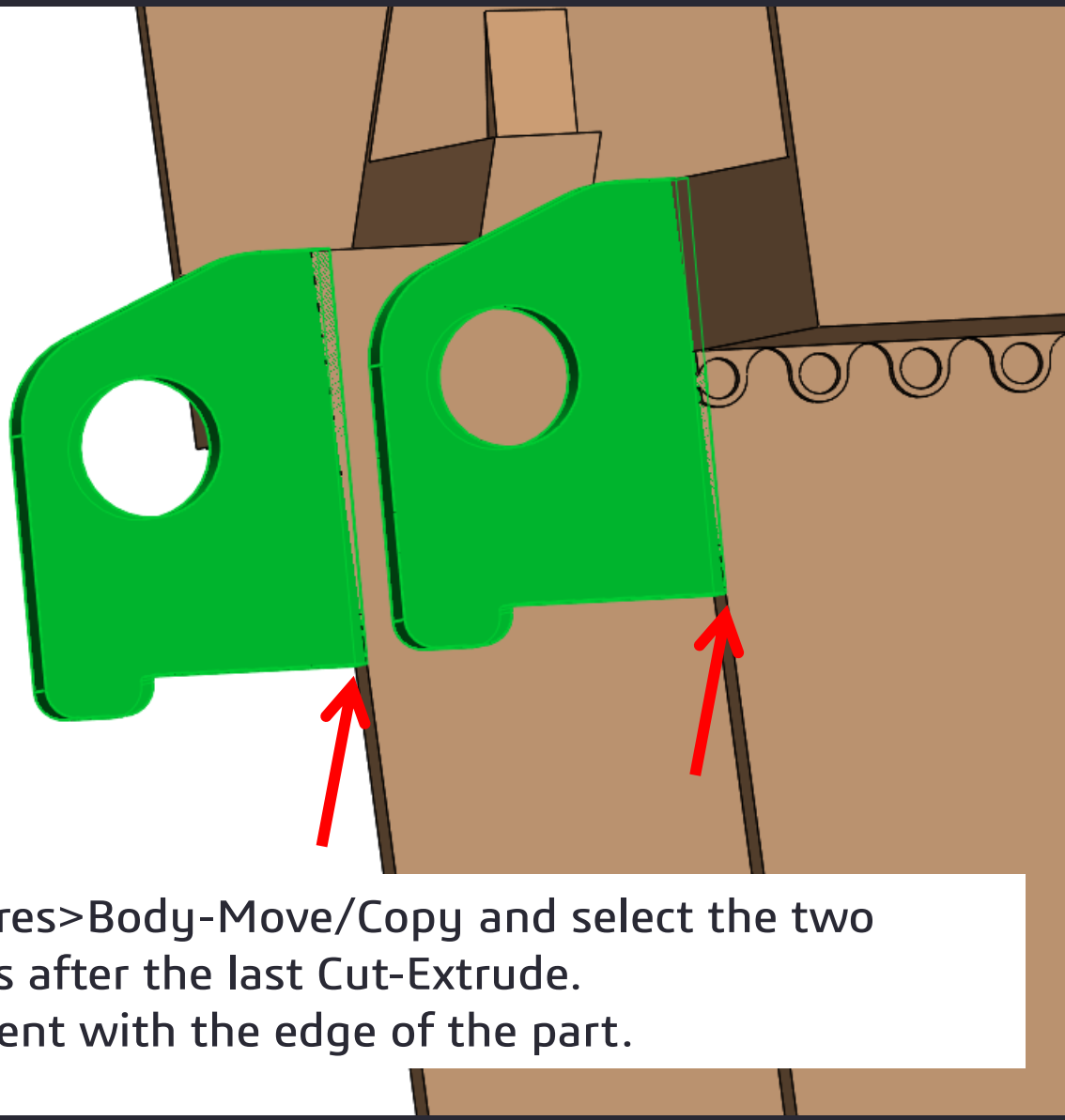
Use “Cut-Extrude”
set Direction 1 to *Blind* and **80.7** mm
Set Direction 2 to *Blind* and **39.8** mm



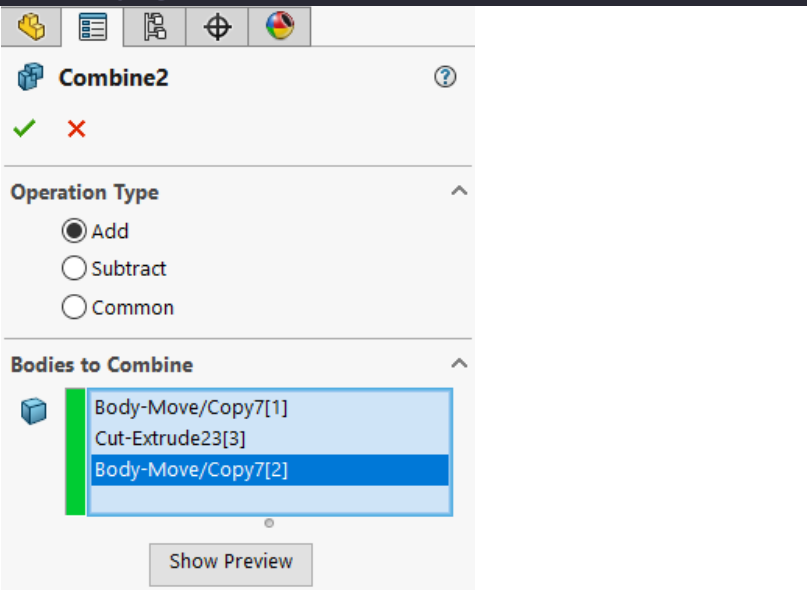


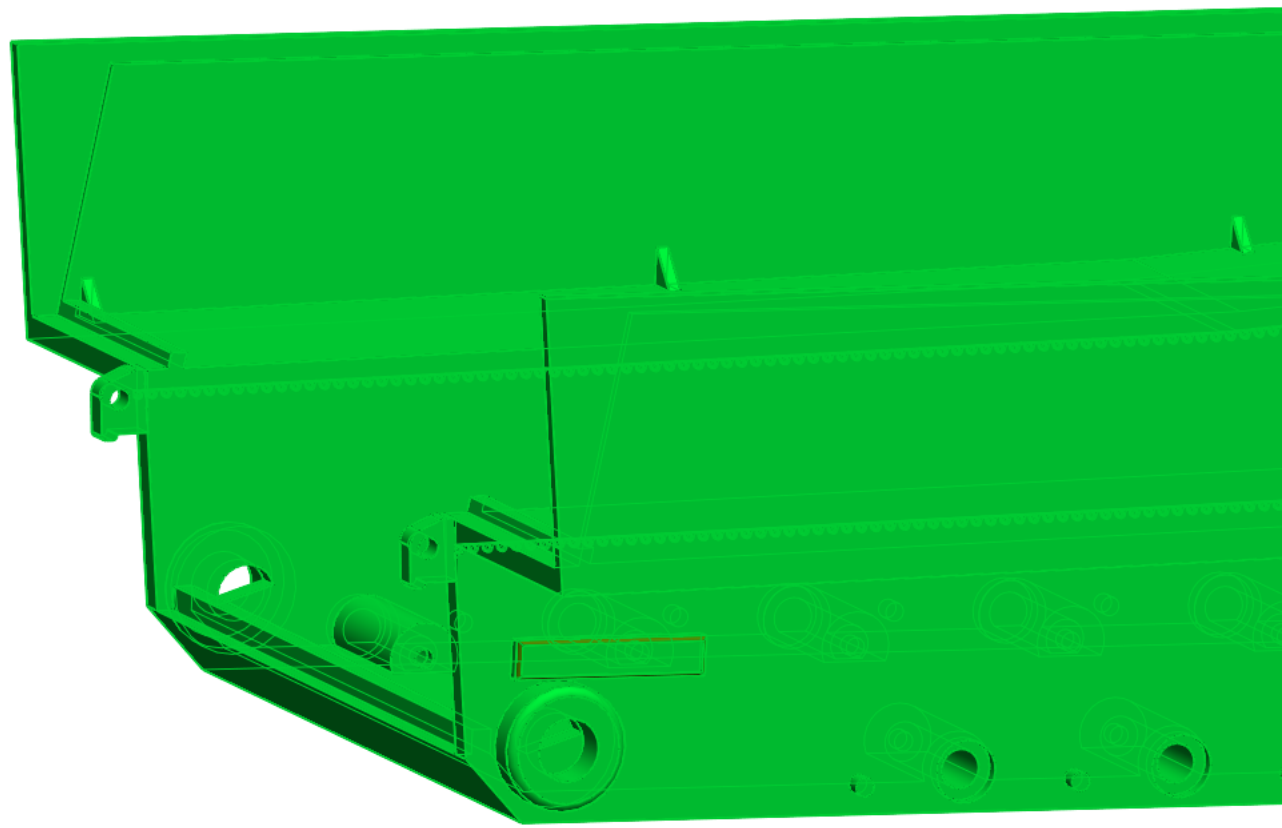
Go to Insert>Features>Body-Move/Copy and select the two newly created parts after the last Cut-Extrude.
Make them coincident with the edge of the part.



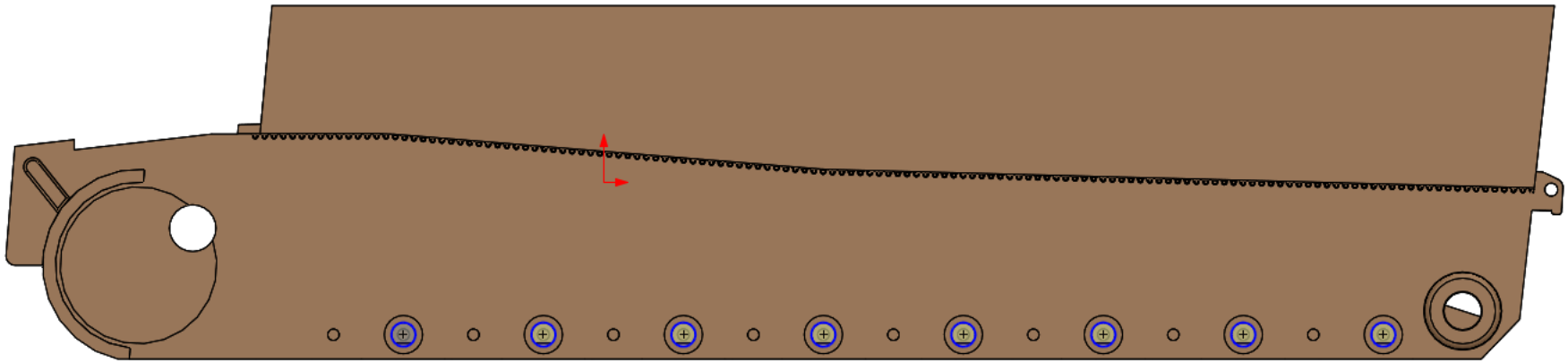


Go to Insert>Features>Body-Move/Copy and select the two newly created parts after the last Cut-Extrude.
Make them coincident with the edge of the part.

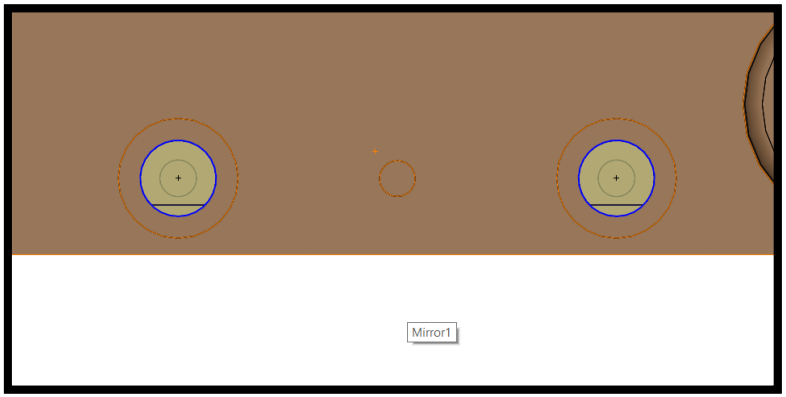


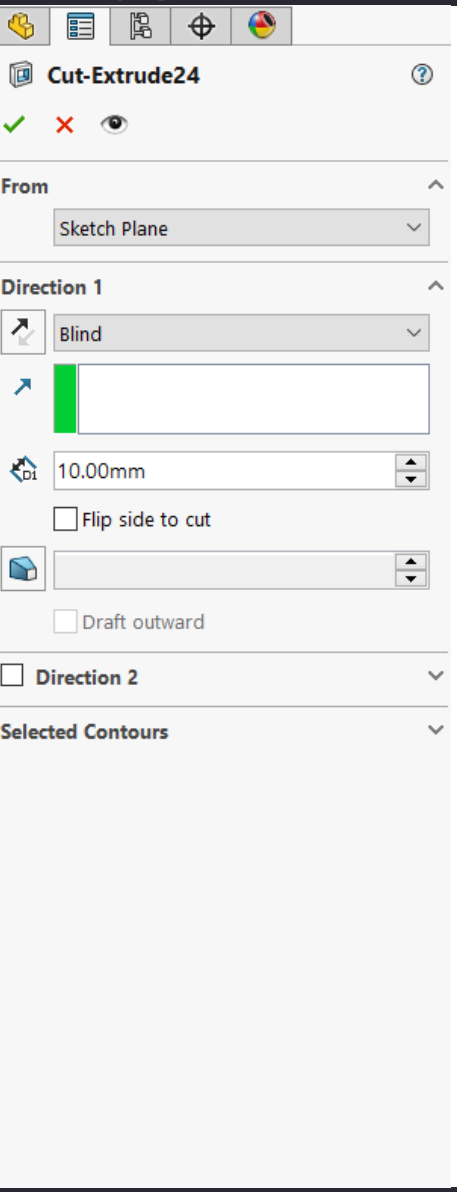


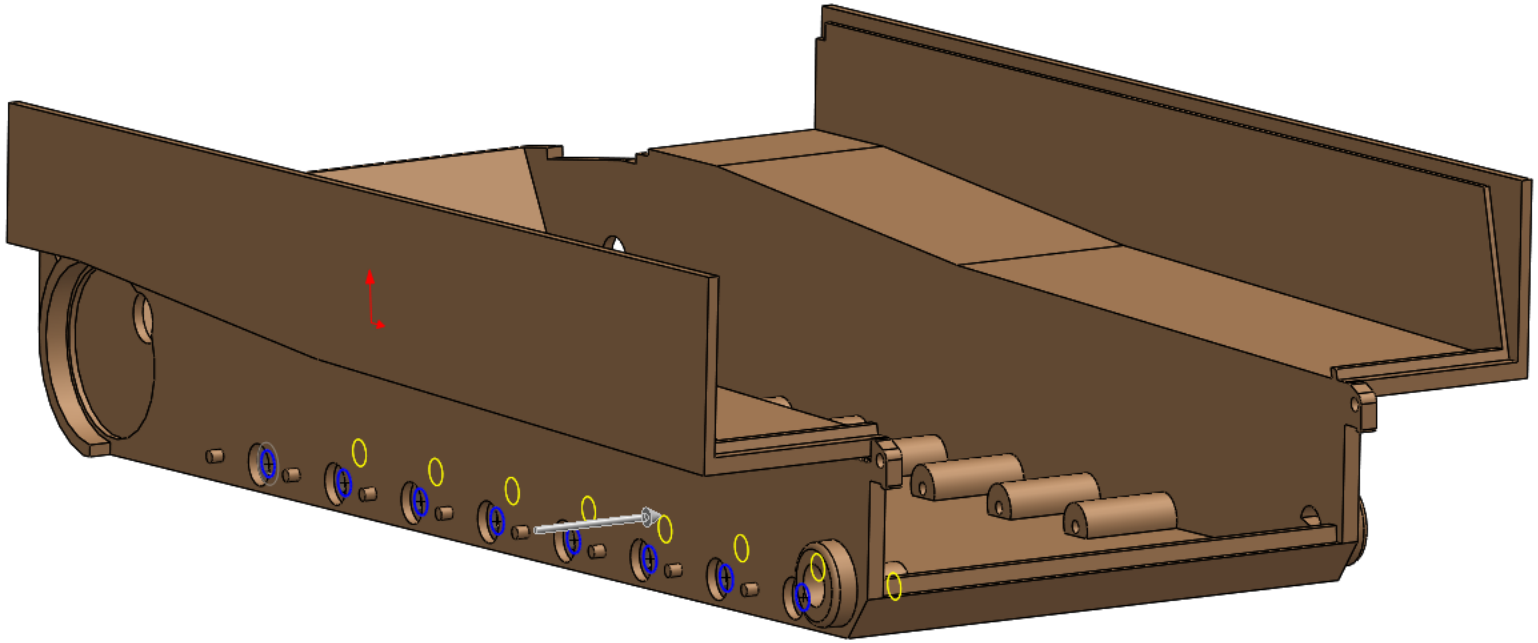
Go to Insert>Features>Combine and add the two newly created parts wit the main part.



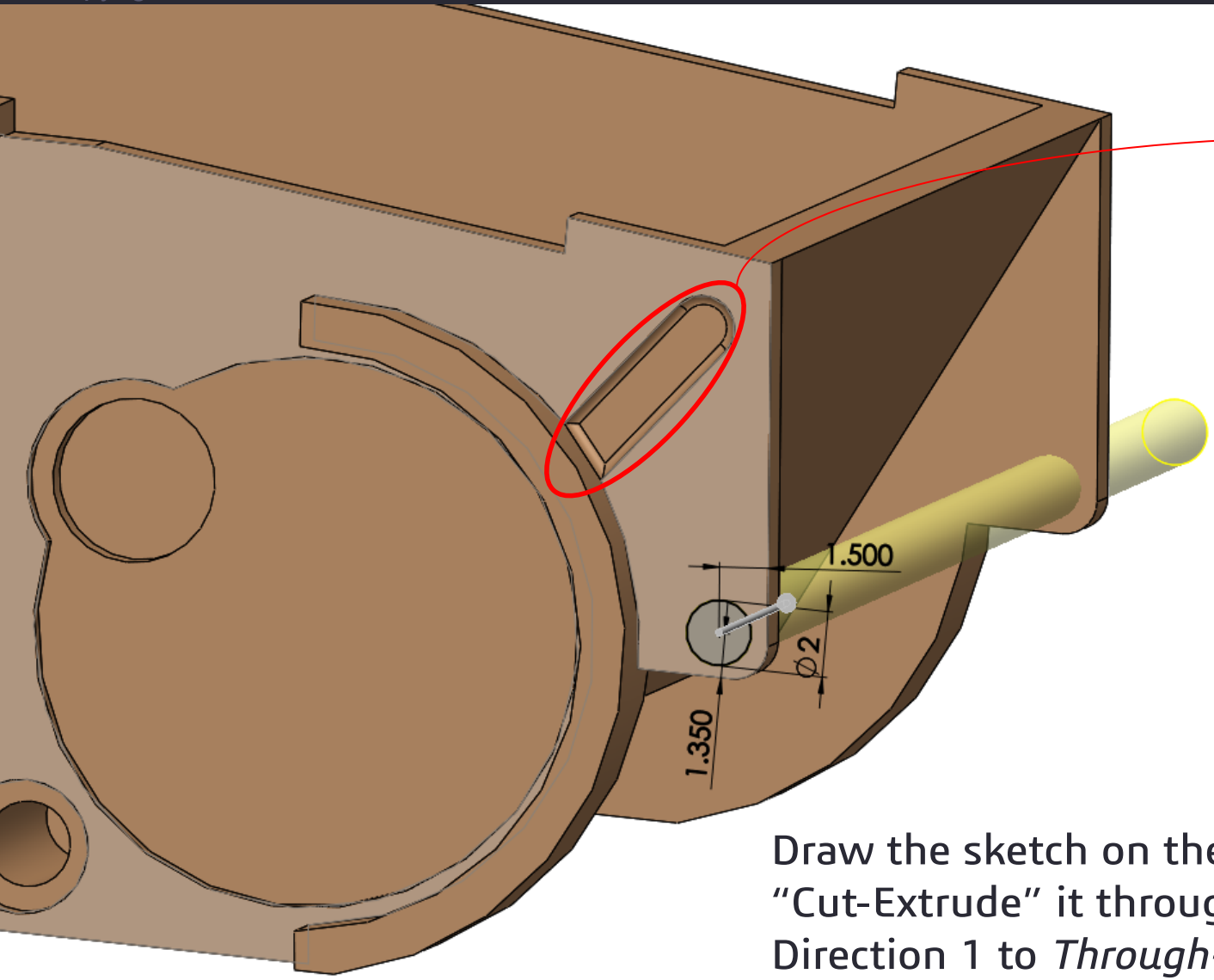
Draw concentric circles on the side surface of the part and make them coradial with the cylindrical holes.



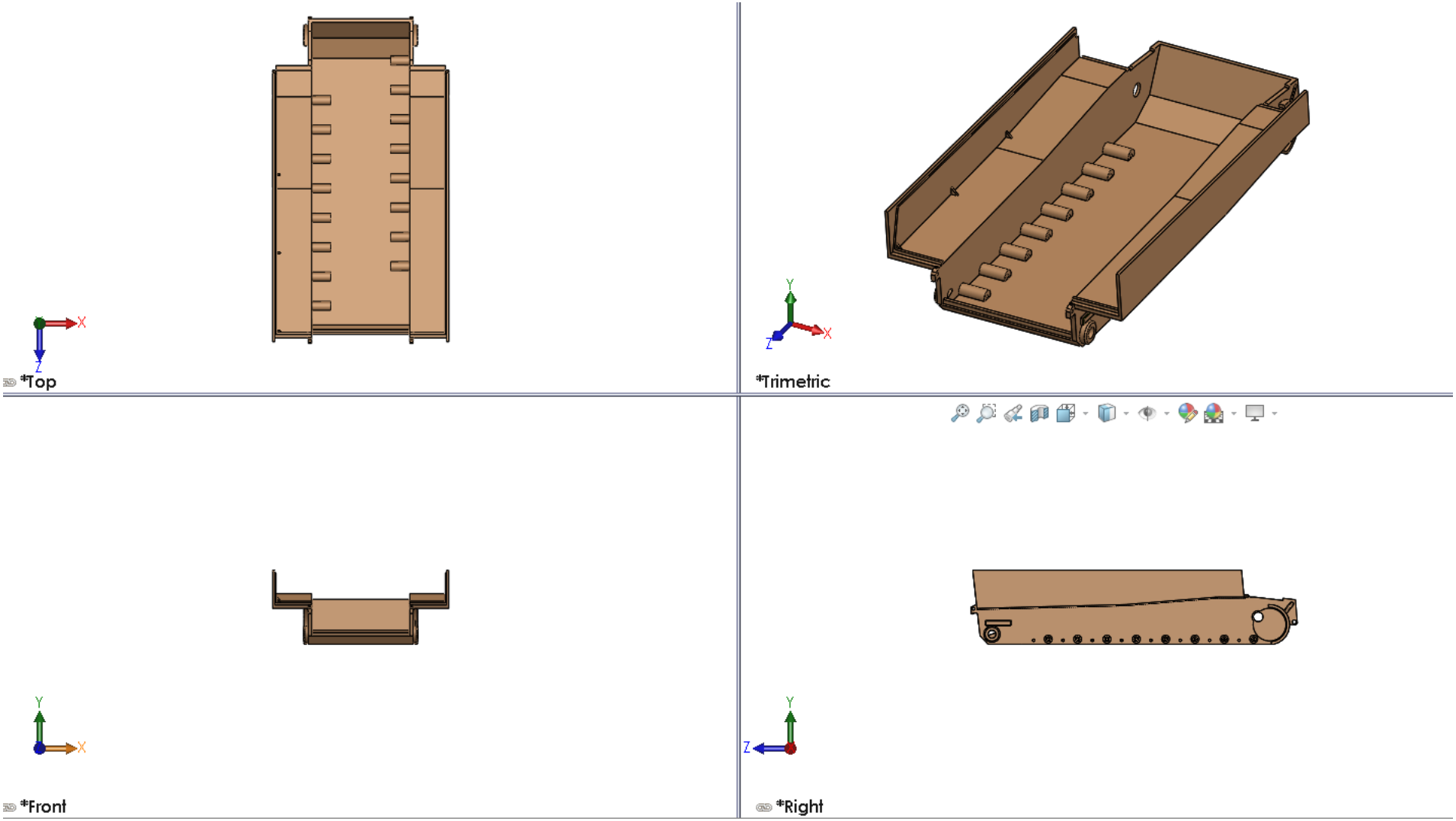




Use “Cut-Extrude”, Set Direction 1 to Blind and cut them for **10** mm into the part.



Draw the sketch on the surface of the part and "Cut-Extrude" it through the part by setting the Direction 1 to *Through-All.*



SAVE YOUR PART

TIGER I

SOLIDWORKS TUTORIALS