# SOLIDWORKS TUTORIALS TIGER TANK I PDF TUTORIAL

Part: Chassis Difficulty: Medium Pages: 88



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# PART NAME: Chassis



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# **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** 

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Ì	Boss-Extrude1	?
<b>~</b>	×	
From		^
	Sketch Plane	$\sim$
Direc	tion 1	^
2	Blind	$\sim$
*		
Ci	37.80mm	▲ ▼
		▲ ▼
	Draft outward	
D	irection 2	~
Selected Contours		

Use "Boss-Extrude" and extrude the sketch for **37.80** 



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

The two green lines are parallel.

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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.

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### Cut-Extrude2 ? × 0 From $\sim$ Sketch Plane Direction 1 2 Blind ▲ ▼ Kai. 18.20mm Flip side to cut ▲ ▼ Draft outward Direction 2 Selected Contours $\sim$ Sketch3-Region<1> $\circ$

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



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### 0 🕀 4 G Mirror1 ? 🗸 🗙 Mirror Face/Plane **Right Plane** Features to Mirror Cut-Extrude2 Faces to Mirror Options Geometry Pattern Propagate visual properties Full preview Partial preview

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



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



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

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



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- 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







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# Your Part so far...



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





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🍕 🗐 🛱 🔶	
🗊 Plane1	
✓ ×	
Message	^
Fully defined	
First Reference	^
Front Plane	
N Parallel	
Perpendicular	
Coincident	
<b>1</b>	$\sim$
0	
Mid Plane	
Second Reference	^
Vertex<1>	
K Coincident	
Project	
0	~
Third Reference	^

Options

Flip normal

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



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🌯 🗐 🖹 🗘 🔶	
🗊 Plane2	?
✓ ×	
Message	^
Fully defined	
First Reference	^
Plane1	
N Parallel	
Perpendicular	
🔨 Coincident	
0	< >
5.00mm	~ ~
✓ Flip offset	
Mid Plane	
Second Reference	^
Third Reference	^
Options	^

Flip normal

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



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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.



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Direc	tion 1
2	Edge<1>
	Spacing and instances

O Un to reference	

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Direc	tion 2		~
🗹 Fe	atures and Faces		~
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Ø			
		0	
Bo	odies		~
nsta	nces to Skip		~
Optio	ons		/
	Vary sketch		
	Geometry pattern		

Propagate visual properties

Full preview

Partial preview

Instances to Vary

- Use "Linear Pattern" 1
- Select the green highlighted edge as *Direction 1* 2.

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- Set the distance to **40** mm 3.
- Set the number of instances to 3 4.
- Select the Rib 5.
- Click Ok 6.



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

All circles are the same size.



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4				
P	Fillet1	?	Use "Fillet", set its radius to 1	mm and round the two
1	×		highlighted edges	
Feat	ture Type	^		
Item	ns To Fillet	^		
	Edge<1> Edge<2>			
	O Partial preview			
	○ No preview			
Fille	<b>t Parameters</b> Symmetric	~		
R	1.00mm	▲ ▼		
<b>D</b> (	Multi Radius Fillet			
Pror	Circular	~		
Setb	pack Parameters	~		
Fille	et Options	~		Radius: 1mm

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



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<ul> <li>♥ ● ●</li> <li>♥ ● ●</li> <li>♥ Cut-Extrude7</li> <li>✓ × ●</li> <li>From <ul> <li>Sketch Plane</li> </ul> </li> <li>Direction 1</li> <li>♥ Through All</li> <li>♥ Flip side to cut</li> <li>♥ Flip side to cut</li> </ul>		<ol> <li>Use "Cut-Extr</li> <li>Set it at <i>Throw</i></li> <li>Select the larg</li> <li>Click Ok</li> </ol>	ude", Jgh All Jer area in the Selected Contours
Draft outward	~		1.046
Sketch12-Region<1>			

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		Cut-Extrude8	D
2.400		✓ X ⑨	
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· · · · · · · · · · · · · · · · · · ·		Sketch Plane ~	
		Direction 1	^
		Blind ~	
		× .	]
		🚯 1.70mm	]
		Flip side to cut	
		Draft outward	
		Direction 2	~
		Selected Contours	^
		Sketch12-Region<1>	
		0	J
Use "Cut-Extrude",			
Set it at <i>Blind</i>			
Select the smaller area in the Selected Contours			

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Partial preview

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G Mirror2

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Mirror Face/Plane				
	Right Plane			
Features to Mirror				
G	Cut-Extrude8			
	0			
Faces to Mirror ^				
$\bigcirc$				
	0			
Options ^				
Geometry Pattern				
Propagate visual properties				
O Full preview				

- Use "Mirror" 1
- Select the *Right Plane* as the *Mirror Face/Plane* 2.

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- Select the Cut-Extrude from the last step 3.
- And mirror it to the other side. 4.



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



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<ul> <li>Image: Sketch Plane</li> <li>Image: Sketch Plane</li> <li>Image: Sketch Plane</li> </ul>	<sup>3</sup> Use "Cut-Extrude" set it to All and cut it through the	<i>Through-</i> part.
Direction 1          Through All         Through Contemporation         Flip side to cut         Draft outward		
Selected Contours		







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.

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✓ × ●	G	Use "Cut-Extrude", set Direction 1				
From Sketch Plane	~	to Blind at <b>U.66</b> mm and cut it into the part.				
Direction 1	~					
C.66mm	<b>*</b>					
Draft outward	▲ ▼					
Direction 2						
Selected Contours	~					

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

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	Cut-Extrude11	(?)	
~	× (1)		
From	I	^	
	Sketch Plane	$\sim$	
Direc	tion 1	^	
2	Blind	$\sim$	
*			
<b>₹</b> Di	0.66mm	▲ ▼	
	Flip side to cut		
		▲ ▼	
	Draft outward		
	Virection 2		
Selec	ted Contours		
			•
			L.

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



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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.

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Boss-Extrude2

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Sketch Plane

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From

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Direction 1

Blind

2.23mm

Merge result

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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.

Draft outward Direction 2  $\sim$ Selected Contours  $\sim$ Sketch16-Region<1>

0.500 **SOLIDWORKS TUTORIALS** For queries about this tutorial INFO@SOLIDWORKSTUTORIALS.NET



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



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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.

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		BD LPattern3	?
		✓ ×	
		Direction 1	~ ′
		Edge<1>	
		Spacing and instances	
		○ Up to reference	
		🗞 15.00mm	▲ ▼
		₽# 8	▲ ▼
		Direction 2	~
<b>4</b> ₿=		Features and Faces	^
	Direction 1	Cut-Extrude14	
Sp	acing: 15mm 😜	0	
Ins	ances: 8 🜩		
		0	
1.	Use "Linear Pattern" feature	Bodies	~
2.	Select the yellow edge as <i>Direction 1</i>	Instances to Skip	~
3.	Set the distance to <b>15</b> mm	Options	^
Λ	Sat the number of instances to 8	Vary sketch	
4.		Propagate visual properties	
5.	Select the two recently Cut-Extrudes in the	Full preview	
	Features and Faces	Partial preview	
6.	Click Ok	Instances to Vary	~ 、
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### Make them *concentric* with the circular hole

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🜒 Boss-Extru	ide5 ⑦		
🗸 🗙 👁			
From	^		
Sketch Plan	e ~		
Direction 1	^		
Blind	$\sim$		
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9.80mm			
Draft out	▼.		4.80
Discretion 2			
selected Contour	s Ý		
1.	Use "Boss-E	ktrude"	
2	Set Direction	1 it to Blind at <b>9.8</b> mm	
2. 7			
5.		ge Results	
4.	Click Ok		

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Direction 1

15mm

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Spacing:

Instances: 8

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**Click Ok** 6.

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- 2. Select the Right Plane for Mirror Face/Plane
- 3. Select the four yellow highlighted features to mirror

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Bodie	s to Move/Copy		
8	Boss-Extrude5		1
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	LPattern4[2]		
	LPattern4[3]		
	LPattern4[4]		
	LPattern4[5]		
	LPattern4[6]		
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	•		
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Transl	ate		^
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ΔY	0.00mm	4	
۸Z	0.00mm	$\uparrow$	
		~	
Rotate	2		~
	Constraints		

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- 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





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<b>[</b> 4.	[

₩ C C C C C C C C C C C C C C C C C C C	Face<1 Face<2	> 2>			
		Add	Undo		
1	Coincide	ent			
N F	Parallel				
T t	Perpend	icular			
0	Fangent				

Concentric

30.00deg

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Mate Alignment

Flip Dimension

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K Coincident1 (Body-Move/Copy2,Shell1)

Translate/Rotate

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Mates

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- 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



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#### 🗟 🖪 🕅 🔶 Body-Move/Copy5

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### Bodies to Move/Copy4[8] Body-Move/Copy4[7] Body-Move/Copy4[6] Body-Move/Copy4[5] Body-Move/Copy4[1] Body-Move/Copy4[2] Body-Move/Copy4[4] Body-Move/Copy4[3]

Trans	late	/
ΔХ	0.00mm	\$
ΔΥ	0.00mm	\$
ΔZ	19.27349401mm	¢
Rotat	e	```

Constraints

- 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 <u>Z</u> direction



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#### Body-Move/Copy5

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Translate

ΔX

ΔY

ΔZ

Rotate

0.00mm

0.00mm

19.27349401mm

Constraints

**%** 

Bodies to Move/Copy Body-Move/Copy4[8] Body-Move/Copy4[7] Body-Move/Copy4[6] Body-Move/Copy4[5] Body-Move/Copy4[1] Body-Move/Copy4[2] Body-Move/Copy4[4] Body-Move/Copy4[3]

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- 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 <u>Z</u> direction
- 6. Click Ok



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



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<ul> <li>♦ E R + ●</li> <li>♥ Cut-Extrude16</li> <li>✓ × ●</li> <li>From</li> <li>Sketch Plane</li> </ul>	(2)	1. Us 2. Se 3. Cl	se "Cut-Extrude" et Direction 1 to Through-All lick Ok
Direction 1  Through All  Flip side to cut  Draft outward			
Direction 2 Selected Contours	~		28.398

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SW

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

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





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Image: Second Structure   Image: Second Structure   Image: Sketch Plane   Image: Sketch Plane <		1. 2. 3.	Use "Cut-Extrude" Set Direction 1 to Blind at 1 mm Click Ok	

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<ul> <li>LPattern5</li> <li>X</li> <li>Direction 1</li> <li>Edge&lt;1&gt;</li> <li>Spacing and instances</li> <li>Up to reference</li> <li>15.00mm</li> <li>15.00mm</li> <li>8</li> <li>Direction 2</li> </ul>	🗐 🖹 🕁 🥯	
<ul> <li>X</li> <li>Direction 1</li> <li>Edge&lt;1&gt;</li> <li>Spacing and instances</li> <li>Up to reference</li> <li>15.00mm</li> <li>a</li> <li>a</li> <li>b</li> <li>c</li> <li>a</li> <li>a</li> <li>b</li> <li>c</li> <li>c</li> <li>c</li> <li>d</li>     &lt;</ul>	attern5 ⑦	
Direction 1 Edge <1> Spacing and instances Up to reference 15.00mm 8 Direction 2		
Edge <1> Spacing and instances Up to reference 15.00mm 8 Direction 2	n 1 ^ ^	
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Vary sketch 3. Set the distance to 15 mm	Vary sketch	3. Set the distance to <b>15</b> mm
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5. Select the cylindrical body on the other s	l properties	5. Select the cylindrical body on the other side
6 Click Ok		6 Click Ok
	and the Marcia	



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

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#### Boss-Extrude8 ? X 0 From Sketch Plane Direction 1 진 Blind Edge<1> + Coi 1.05mm Merge result \* Draft outward Direction 2 Selected Contours

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



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

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toward the inner side.

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

The corners are coincident with the edges of the part.



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Cut-Extrude18	?		
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*			
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Draft outward			
Direction 2 ~			
Selected Contours ~			



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

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







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

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



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- Soss-Extrude11
  ✓ × ●
- From ~ Sketch Plane Direction 1 7 Blind Л \* 2.40mm Merge result + 2.00deg Draft outward Direction 2 Selected Contours Sketch33-Region<1>  $\Diamond$
- 1. Use "Boss-Extrude"
- 2. Set it to Up to Surface
- 3. Select the Pink surface as the reference
- 4. Click Ok



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- 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

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Setback Parameters $\checkmark$	
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### Use "Boss-Extrude" and extrude the sketch for **0.4** mm



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



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Partial preview

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- Select the yellow highlighted edge as Direction 1
- Set the distance to **15** mm
- Set the number of instances to 8
- Select the last Boss-Extrude
- Click Ok 6.

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Direction 1

15mm

Spacing:

Instances:

\*

\*





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

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Use "Boss-Extrude" and extrude it for 1.15 mm

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Use "Fillet" and round the edges of the previously created



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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.

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Di	80.70mm	<b>•</b>
	Flip side to cut	
		<b>•</b>
	Draft outward	
	Direction 2	^
	Blind	$\sim$
₹D2	39.80mm	<b>•</b>
		▲ ▼
Selec	ted Contours	~



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

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newly created par	ts after the last Cut-Extrude.
Mates ^ Make them coinci	dent with the edge of the part.
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Go to Insert>Features>Combine and add the two newly created parts wit the main part.

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



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

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# t them for **10** mm into the part.

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### SAVE YOUR PART



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