



Solid Edge: How to Design a Part using Synchronous Technology

VIRTUAL LAB EXERCISE GUIDE

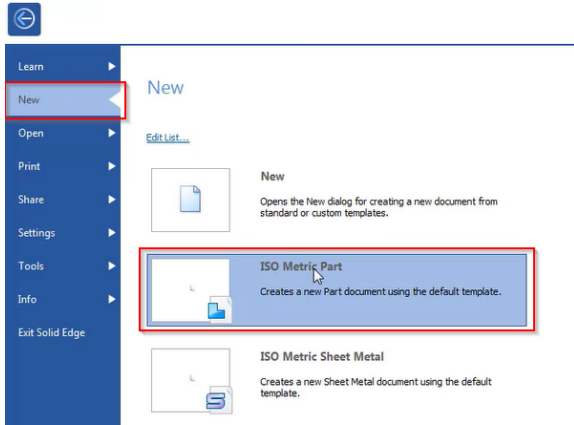
Part Design: How to Design a Part using Synchronous Technology

Introduction

In this tutorial we will create a coupling arm using synchronous modeling techniques in Solid Edge.

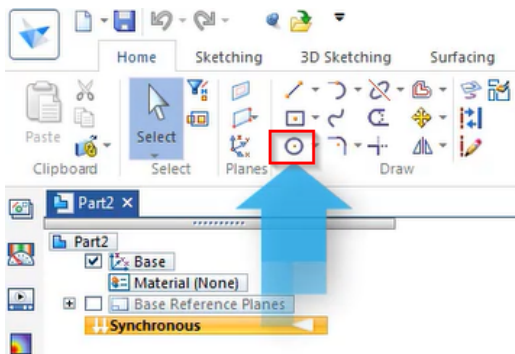
Synchronous Technology in Solid Edge makes sketching and modeling fast and intuitive and gives the user the freedom to modify geometry using simple push-and-pull methods. This tutorial shows you how to work with an imported sketch, make modifications to the sketch and use the sketch for adding features to the geometry.


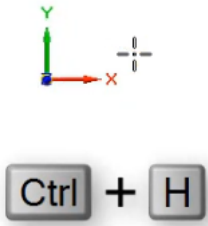
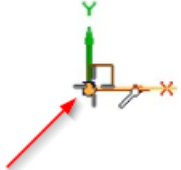
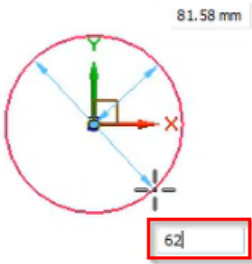
- 1 Click the Solid Edge Application Button and from the New menu, select ISO Metric Part.



Note: When first starting Solid Edge, if a screen appears asking you to choose a theme. Choose the balanced option.

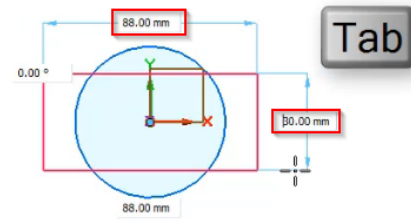
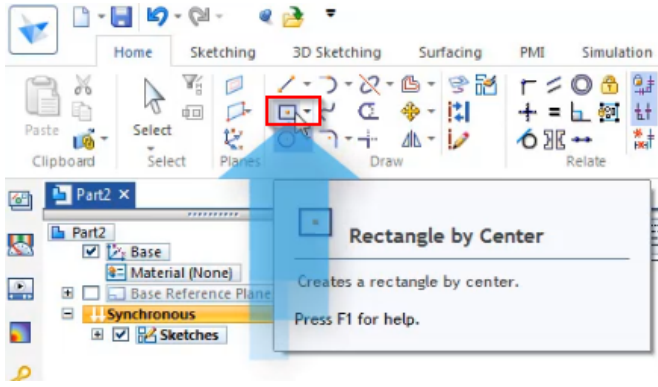
- 2 From the ribbon bar, start the Circle by Center Point command and hover the mouse cursor between the X and Y axis to highlight the plane.



3	Click the lock icon or press F3 to lock the plane for sketching.	
4	Press <Ctrl> + H to orient the selected plane parallel to the screen.	
5	Click at the origin to start drawing the circle.	
6	Type in 62 as the diameter and press <Enter>.	

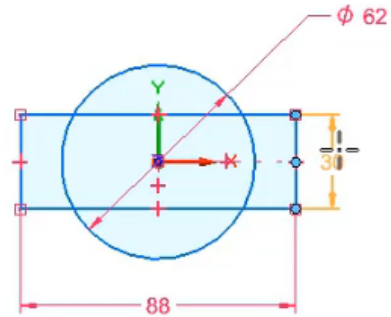
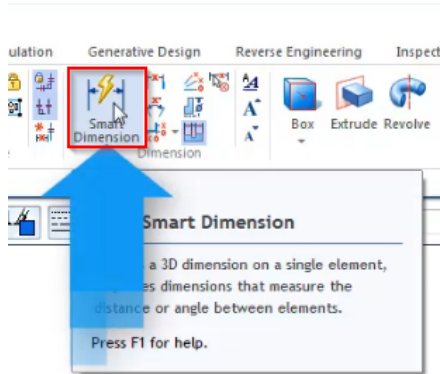
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Start the Rectangle by Center command and draw a rectangle of width 88, height 30 and angle 0. Hover over the plane and hit F3 to lock to it. Press the tab key after specifying each dimension.



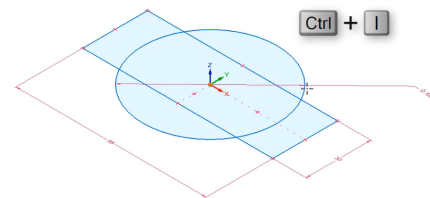
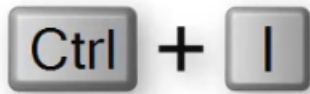
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Start the Smart dimension command and select the circle and the rectangle sides to apply dimensions.



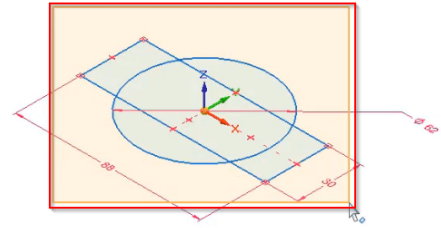
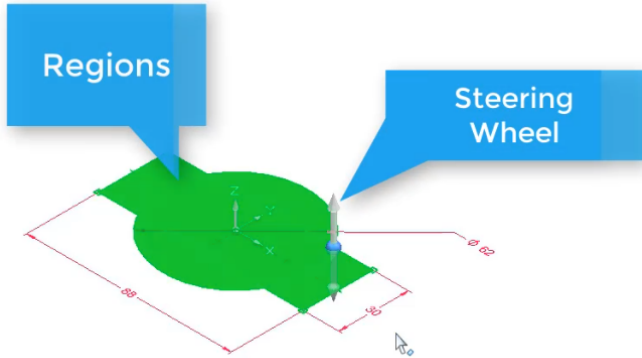
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Press <Ctrl> + I to return to the Isometric view.



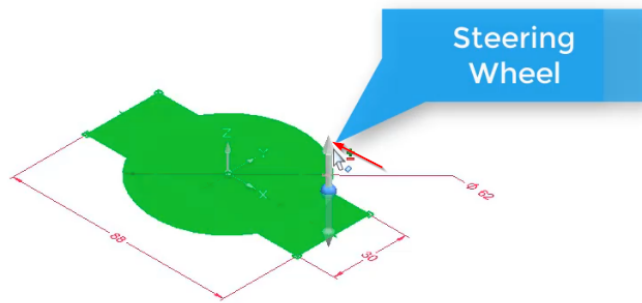
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Click select and drag around the sketches to select them. This creates regions and the steering wheel appears.

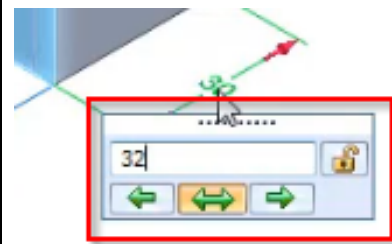
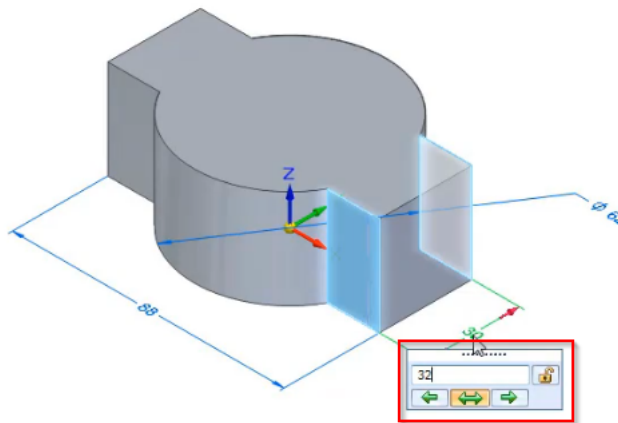
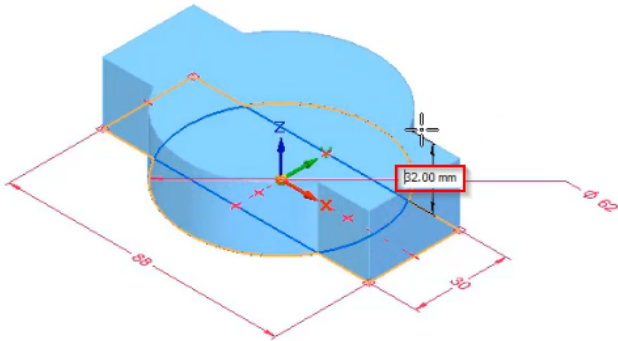


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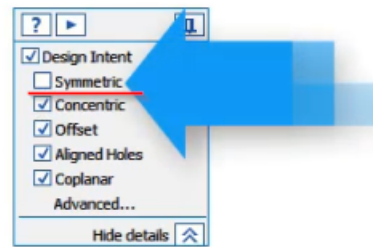
Drag the steering wheel to create the solid.

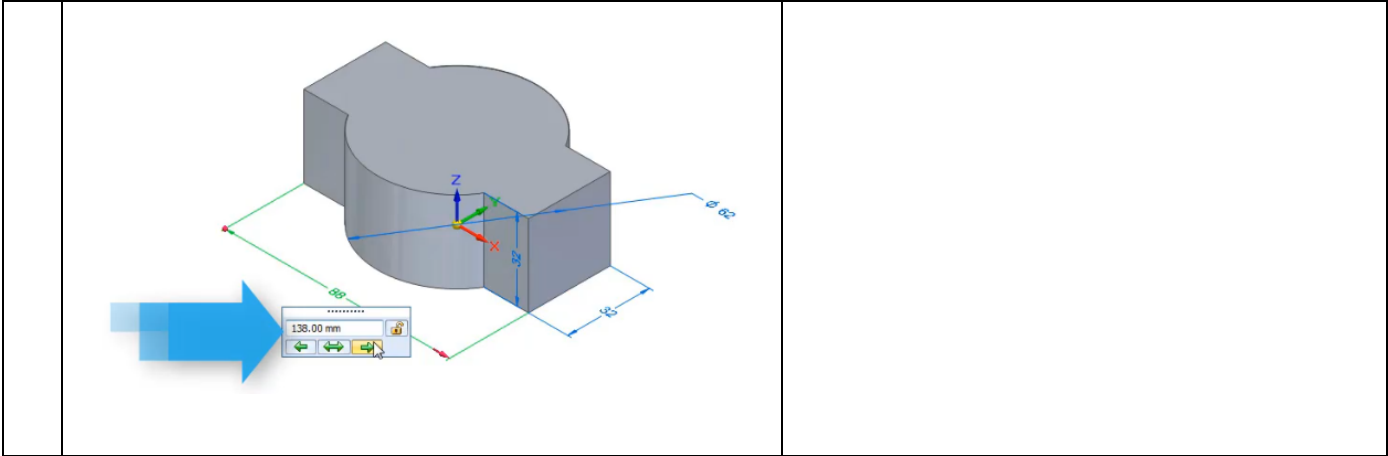


12 Select the height of the rectangle and change it to 32.
Note the model update with the change.

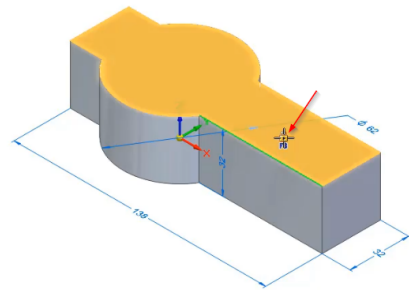
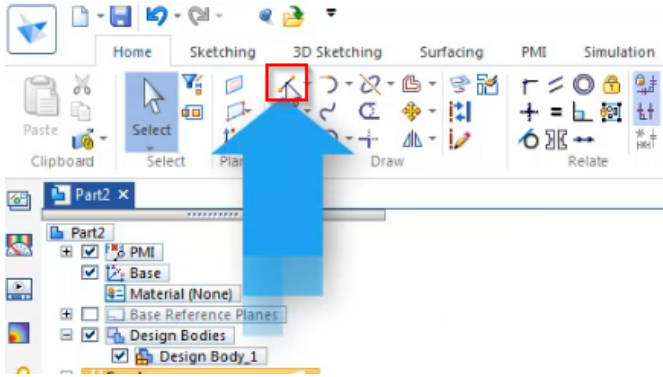


13 Click on the length dimension, and in the dialog box that appears, turn off Symmetric design intent. Click on the right arrow below the dimension, then change the length of the dimension to 138.

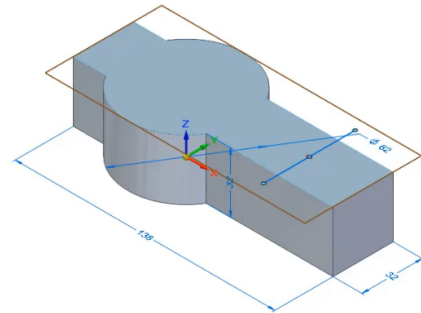
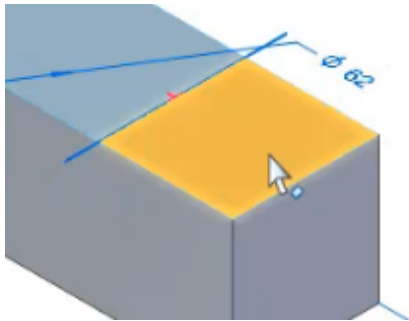




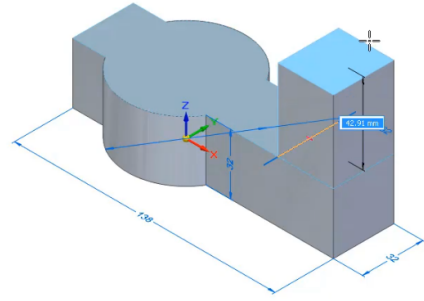
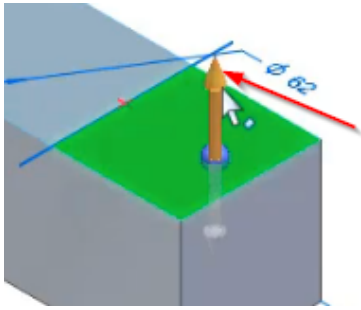
14 Start the line command and lock on to the top face of the model.



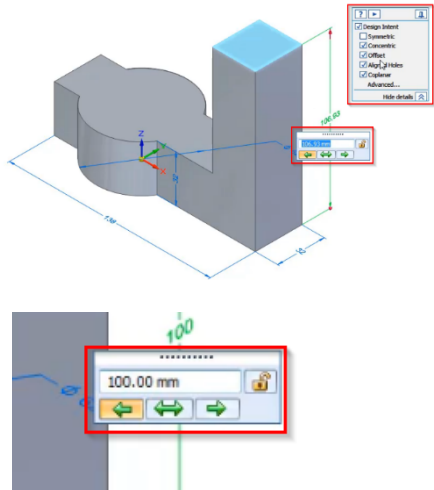
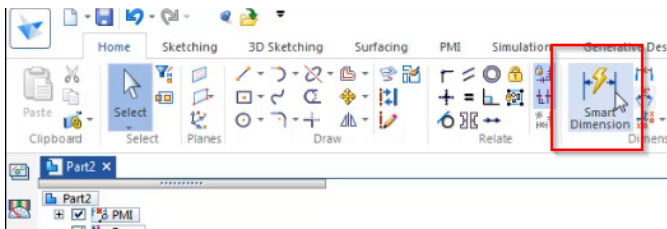
15 Draw a 90 degree line that stretches across the bottom segment of your model, then click enter. Click "Select" then click inside the region formed by the line and the adjacent edges.



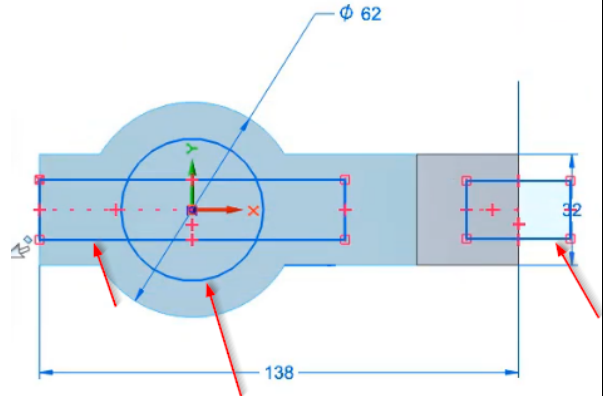
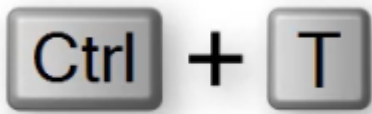
16 Drag the selection up to form the arm feature.



17 Apply a Smart dimension again to the arm height and change it to 100.

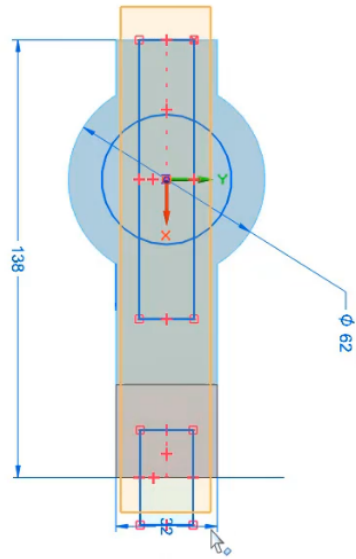
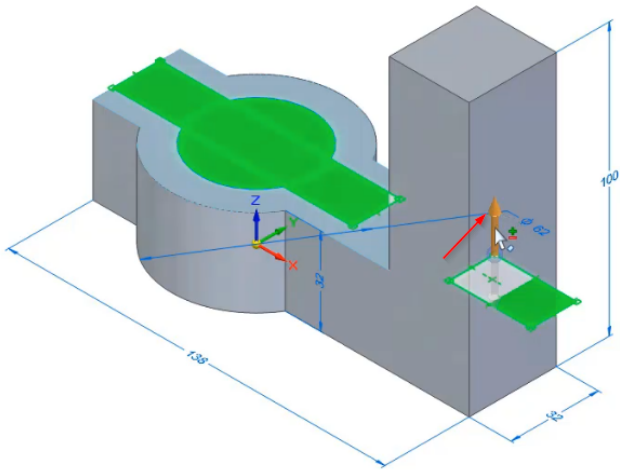
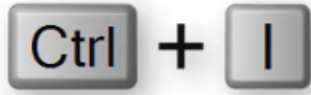


18 Hit Ctrl + T to orient the drawing to view the top face, select the Circle by Center Point command, then click on the lock icon to lock to the top face. Then, draw a circle slightly smaller than your first circle, starting at the same center point. Then draw two rectangles as shown in the picture.



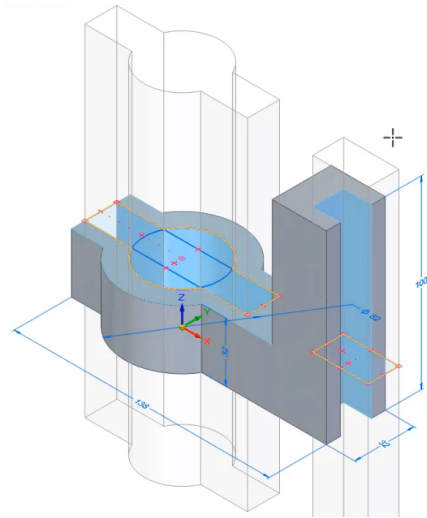
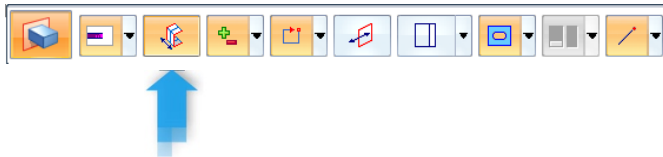
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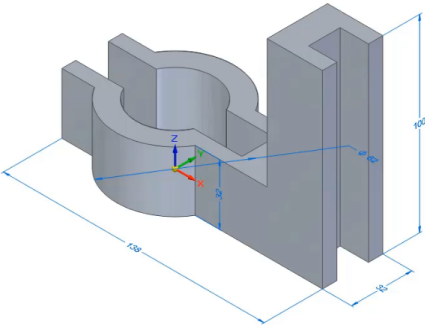
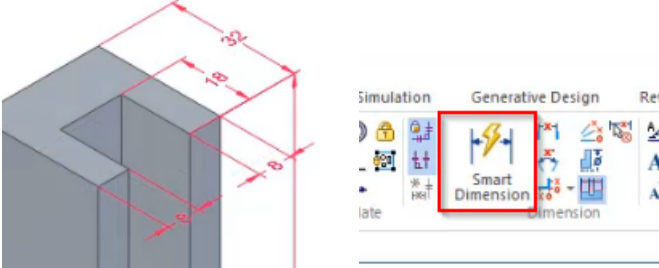
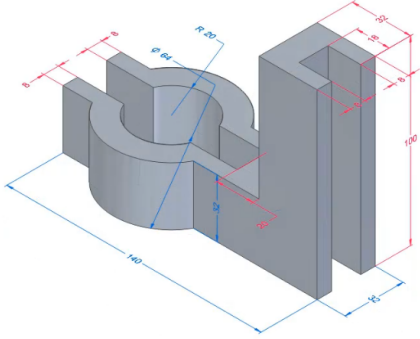
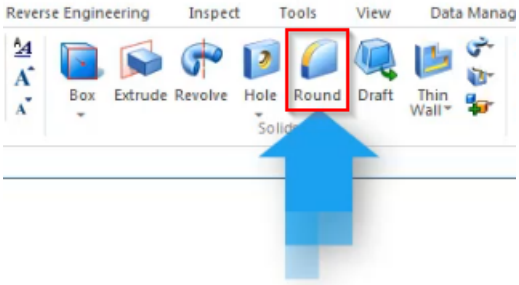
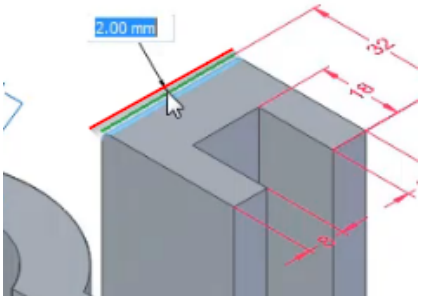
Click Select and drag around the sketches to select them (using Shift + click to select and deselect as necessary) then hit Ctrl + I to return to the Isometric view.



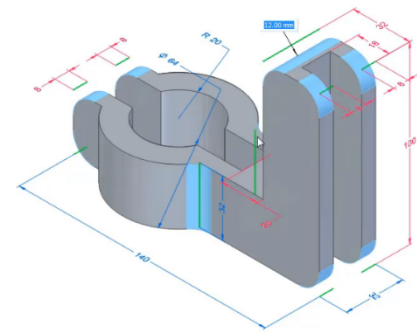
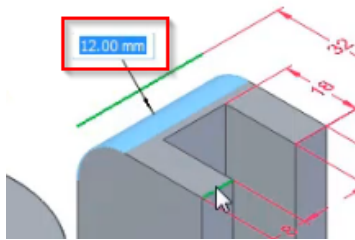
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Drag the steering wheel handle downwards, then press <Shift> once to change to the symmetric mode. You can also choose the Symmetry option in the command ribbon to activate symmetric mode

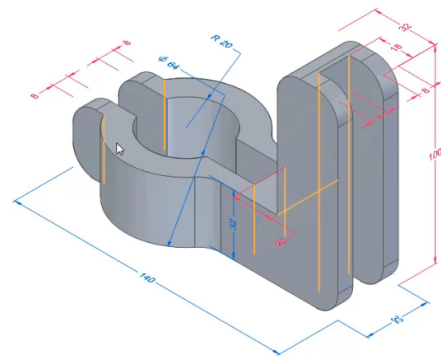
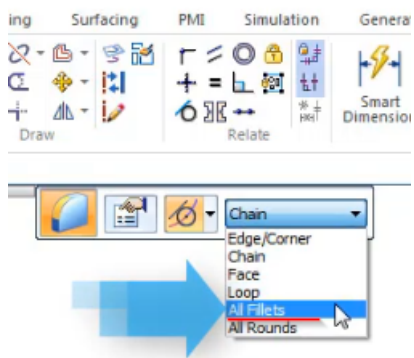


<p>21</p>	<p>Drag through your solid body to create several cutout features in one operation</p>	
<p>22</p>	<p>Apply Smart dimensions to all newly created features. Change the dimensions if they don't match with this model.</p> 	
<p>23</p>	<p>Start the Round command and select an edge.</p> 	

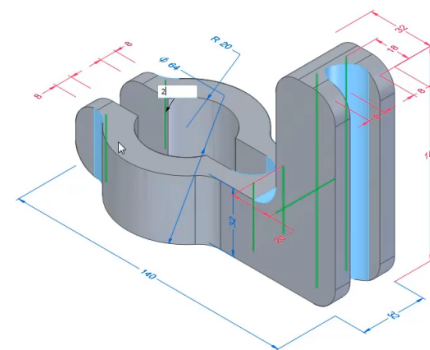
24 Specify radius as 12 and pick edges to create round features.



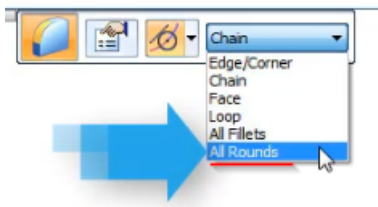
25 Change the option to All fillets and click on the model.



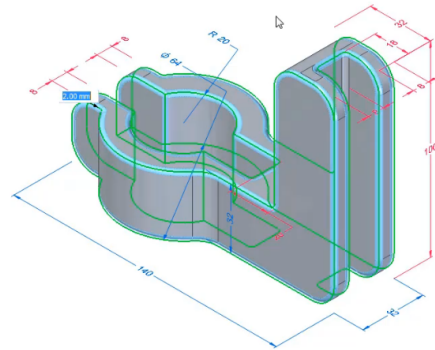
26 Change the radius to 2 and right-click to apply the fillets.



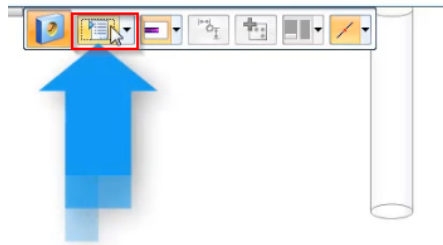
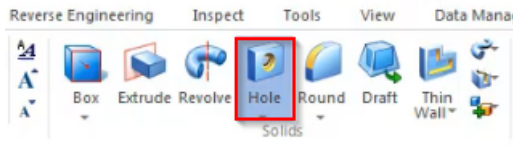
27 Change the option to All Rounds and click on the model again.



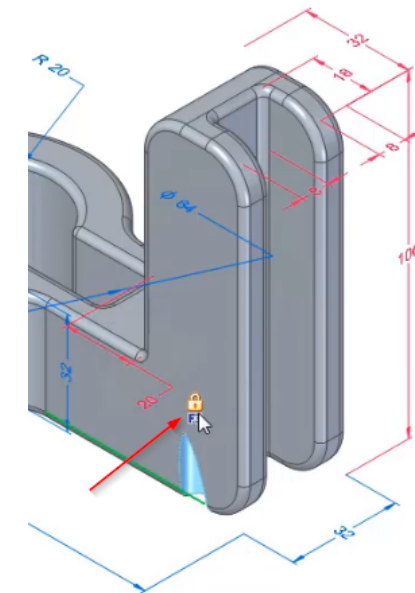
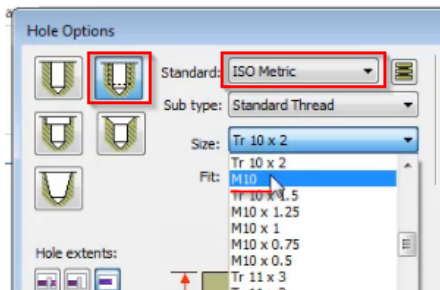
Right-click to apply all rounds.

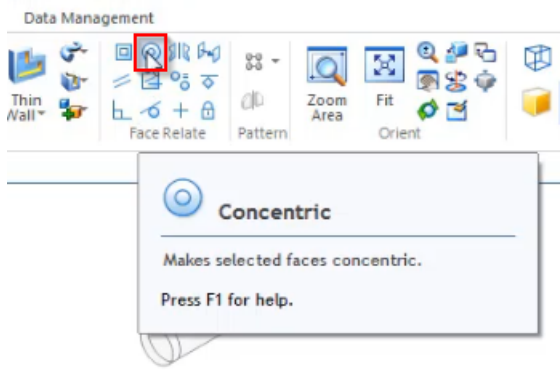
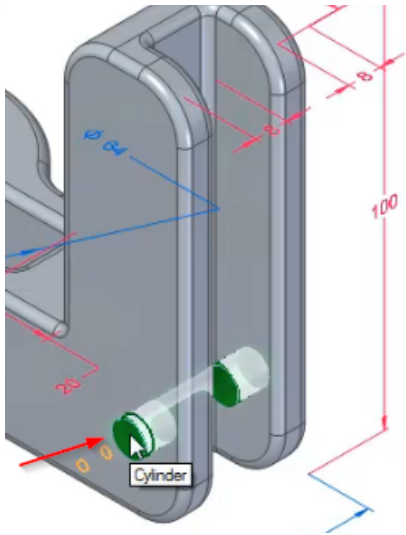
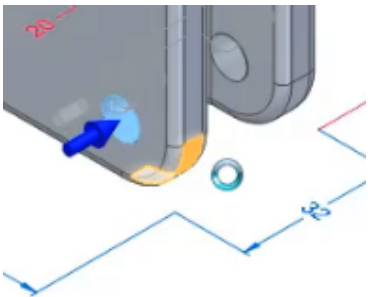
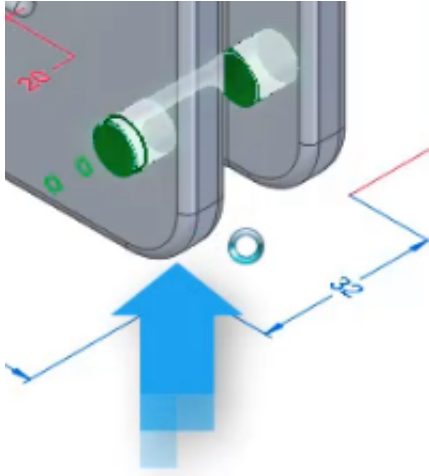
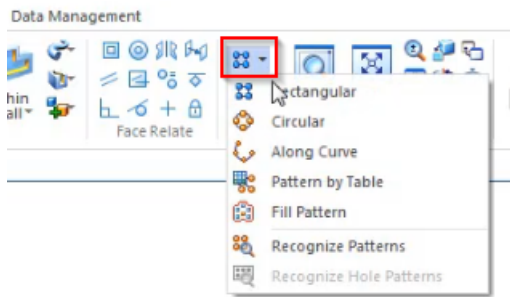
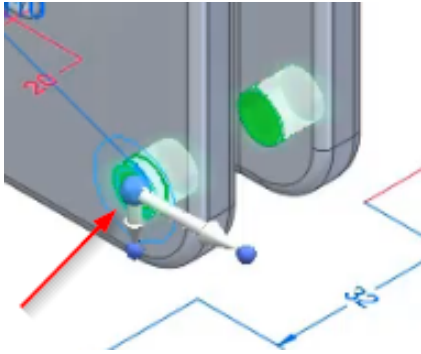


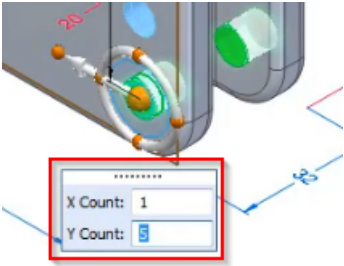
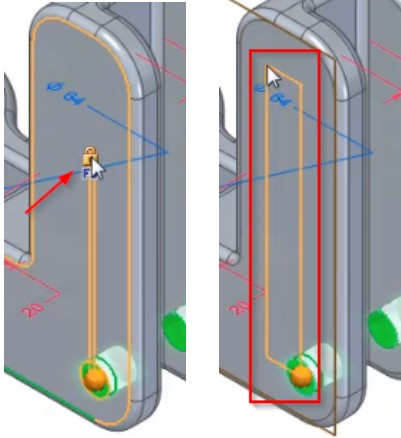
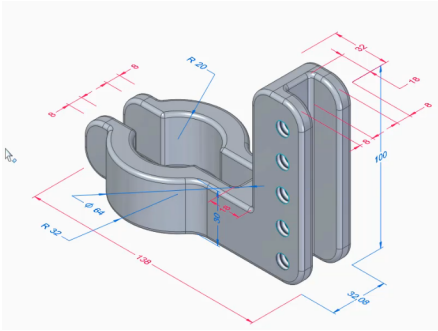

28 Start the Hole command and open the Hole options dialog from the command bar.



29 Change the hole type to threaded and the standard to ISO. Select M10 as the size and click okay to save. Then click on the side face to place the hole.



<p>30</p>	<p>Select concentric face relationship from the ribbon bar and select the hole.</p> 	
<p>31</p>	<p>Right click and select the cylindrical face on the bottom. Right click again to finish the command.</p> 	
<p>32</p>	<p>Select the hole and choose to create a rectangular pattern.</p> 	

<p>33</p>	<p>Click on the lock symbol or hit F3 to lock to the side face. Specify 1 as X Count and 5 as Y Count.</p> 	
<p>34</p>	<p>This finishes the tutorial for creating a coupling arm using synchronous technology.</p>	
<p>35</p>	<p>In this tutorial you learned...</p> <ol style="list-style-type: none"> 1. How to lock to planes and create sketches. 2. How to form regions and create features using the steering wheel. 3. How to apply smart dimension on the model faces and edges and control model size and direction of modification. 4. How to create multiple features in a single operations. 5. How to apply rounds and fillets on multiple edges of the model in a single step. 6. How to create standard holes and rectangular patterns. 	 <ul style="list-style-type: none"> Lock To Plane and Create Sketches Create Features Using the Steering Wheel Apply Smart Dimensions to Model Faces Create Multiple Features in a Single Operation Create Rounds and Fillets Automatically Create Holes and Rectangular Pattern

