



Start a new ANSI Inch part

- Click the Solid Edge application button in the top left hand corner of the window.
- Click on the **"New"** folder then select ANSI Inch Part.
 - If **"ANSI Inch Part"** template is not visible, select the **"Edit List"** button at the top of the list. Then select **"ANSI Inch"** for the templates and click OK.



ANSI Inch Part

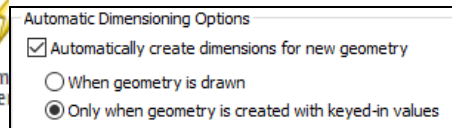
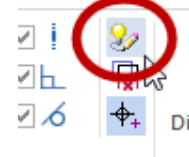
Creates a new Part document using the default template.

Settings

- On the Sketching tab and open the IntelliSketch options.
 - On the Auto-Dimension tab place a check in **"Automatically create dimensions for new geometry"**.
 - Select the the option for **"Only when geometry is created with keyed-in values"**.
 - Click OK to dismiss the IntelliSketch dialog.

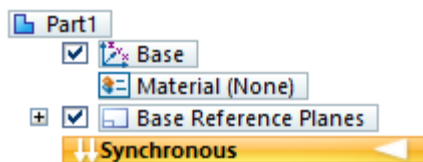
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Add Ins View

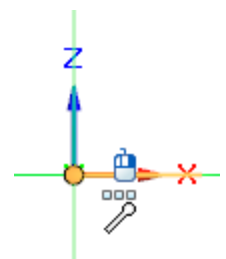
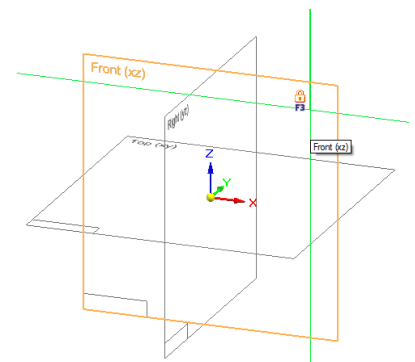
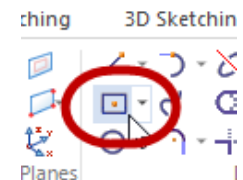


Create Front Panel

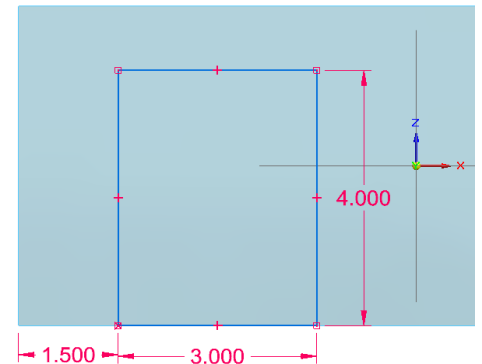
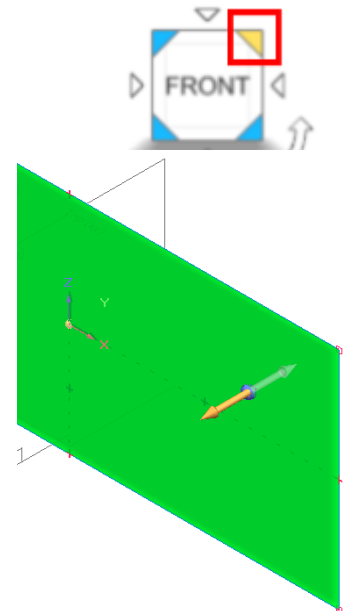
- Under the **"Home"** tab, click on **"Rectangle by Center"**.
- In the PathFinder on the left, check the box next to Base Reference Planes to show the planes on the base coordinate system.



- Hover the mouse over the **"Front"** plane (XZ plane) and click the lock icon or press F3 on the keyboard.
- Click on the Front Face of the view cube in the bottom right corner to change the view.
- Place the cursor over origin point of the base coordinate where the red x axis and the blue z axis meet.
 - The endpoint glyph will appear and click to define the center of the rectangle.

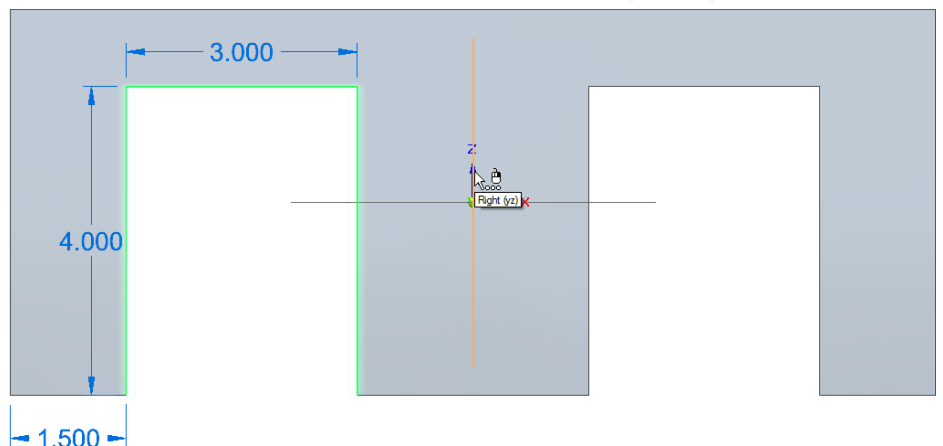
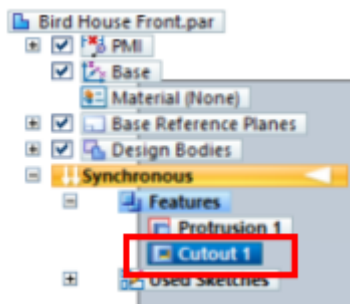


- Drag the mouse to the side to see the rectangle.
 - Type **"12"** for the length.
 - Press tab and key-in **"5"** for the height.
 - Press tab and key-in **"0"** for the angle.
 - Press enter.
- Click on the top right corner of the front face of view cube to switch to an isometric view.
- Click anywhere inside the sketch and select one of the arrows to initiate an extrusion and move the mouse to the left on the screen.
 - Type in **".25"** to create the thickness of the part.
- Save the part as **"Bird House Back"**.
- Select the Save As tab under the Application button menu and select the Save As command to save the file again as **"Bird House Front"**.
- Under the drop down for rectangle by center, select **"Rectangle by 2 Points"** and lock onto the front face of the rectangle.
 - Press Ctrl + H to orient the view normal to the sketch plane.
- Click on the bottom edge of the face to initiate the rectangle.
 - Make the rectangle **3.000** wide, **4.000** high, and an angle of **0.000°**.
- To locate the sketch from the edge of the part, locate **"Smart Dimension"** on the home tab click and select the left line of the sketch and the left edge of the part.
 - Place the dimensions and set to **1.500 in.**
- Press CTL+I to orient to an isometric view and click the region inside the newly created rectangle to display the drag arrows.
 - Click an arrow and drag the region through the part.
 - Click again to finish the cut.



Copying features

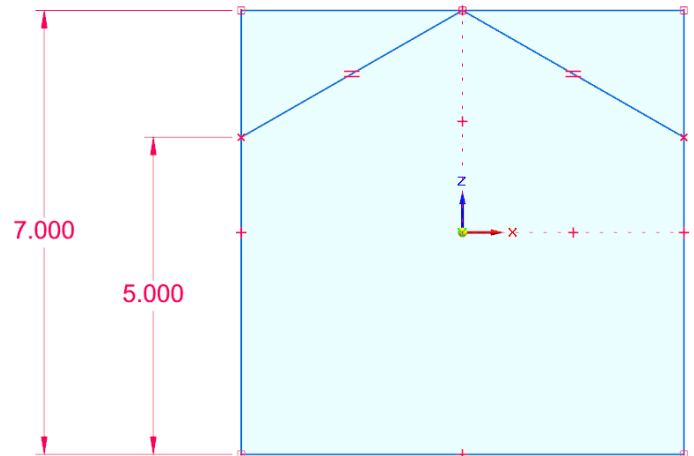
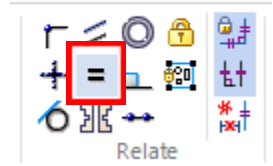
- Change the view to the front view.
- Select the cutout feature in the PathFinder on the left side of the screen.
 - Select the Mirror command and select the Z Axis (Right Plane) to mirror the cutout to the right side of the part.
- Save the part and close the file.



Creating the Side Panels

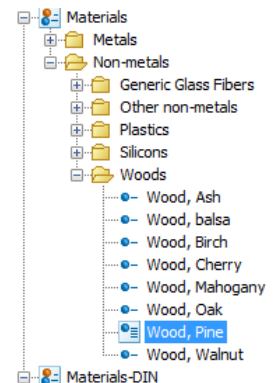
- Start a new ANSI Inch part.

- Using the rectangle by center command, lock onto the front plane and click on the origin of the base coordinate system.
 - Make the rectangle 7.00 in. square.
 - Use the line command to create 2 lines that start from the mid-point of the top line and reaches to each vertical line on the left and right of the rectangle.
 - Add a Smart dimension by selecting on the angled line near the endpoint then select the bottom horizontal line near the corresponding endpoint.
 - Click to place the dimension and key-in 5.00 for the distance.
 - On the Home tab, select the "Equal" Relate command and click on both angled lines.
 - Add a dimension from the top line to the bottom line of the square.
 - Press the ESC key.
- Press CTL+I and click on the center region that is in the shape of a pentagon.
 - Extrude this shape .25 in.
 - Click on the check mark next to sketches in the feature tree to hide the sketch.
 - Click the check box next to PMI in the feature tree to hide the dimensions.



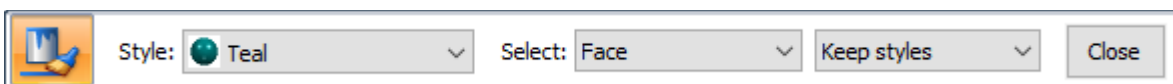
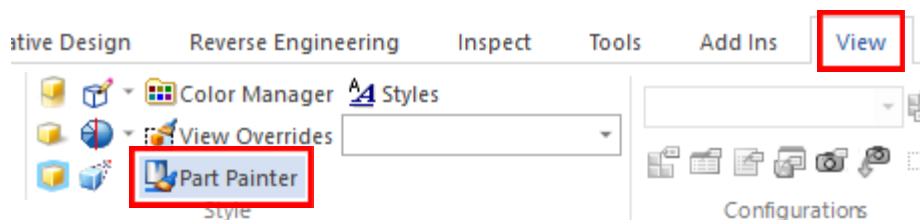
Adding Material Properties

- In the PathFinder, double click on "Material (None)" to open up the materials properties dialog box.
 - Expand "Non-Metals" and then "Woods" to select "Wood, Pine".
 - Click on "apply to model" to add the material properties to the part.



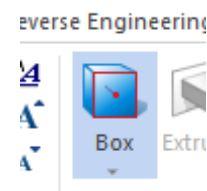
Adding Color to Parts

- Select the "View" tab at the top and select Part Painter in the style command bar.



- Set the "Style" to Teal, set "Select" to face.
- Click on the front face of the part.
- Save the part "Bird House Side".
- Close the Part.

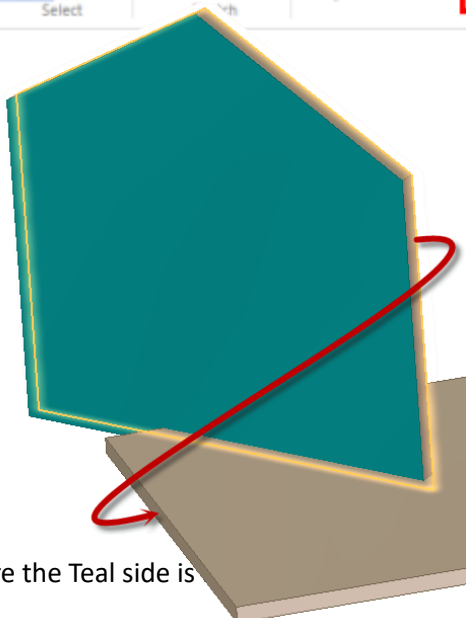
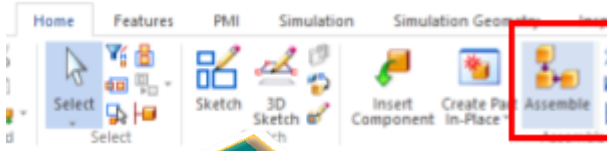
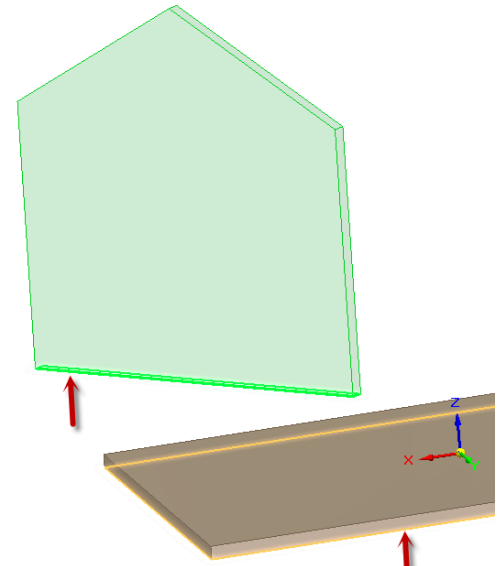
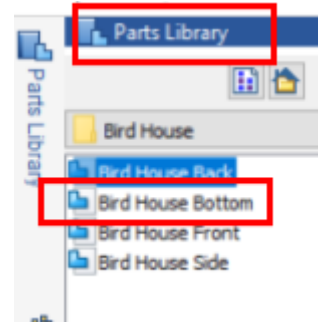
Create Bottom Panel



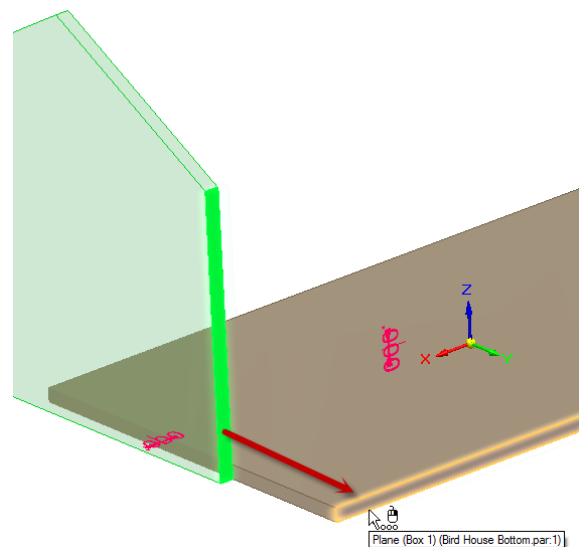
- Start a new ANSI Part.
- This time, let's use the Box command to create a box that is started on the Top plane (x,y Plane), centered on the coordinate system, and is 11.5 in. wide, 7 in. tall and with a thickness of .25 in.
- Add the "Pine" material property to the part, but this time right mouse click on "Material (None)" and under recent materials, select "Wood, Pine" from the top of the list.
- Save the part as "Bird House Bottom".

Assembling the Bird House

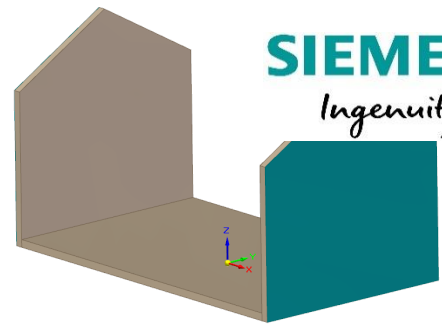
- Start a new ANSI Inch Assembly file.
- On the left side of the window, select the Parts Library tab.
 - Navigate to the location where you have saved the Bird House parts.
- Double click on "Bird House Bottom".
 - Notice the bottom panel is immediately placed at the center of the coordinate system; the part coordinate system is automatically aligned with the assembly coordinate system.
- Now drag and drop the part "Bird House Side".
 - The part is now ready to start relating to the Bottom part.
 - Select the bottom edge face of the side panel and align to the bottom face of the Bottom panel.
 - Right mouse click to exit assembling the parts to see which face was painted Teal.
- Once the painted side has been identified, run the Assemble command.
 - Select the side opposite the teal side and mate to the short edge face of the Bottom panel.
- Next select the side edge face of the Side panel and align to the long edge face of the Bottom panel.



- Be sure the Teal side is

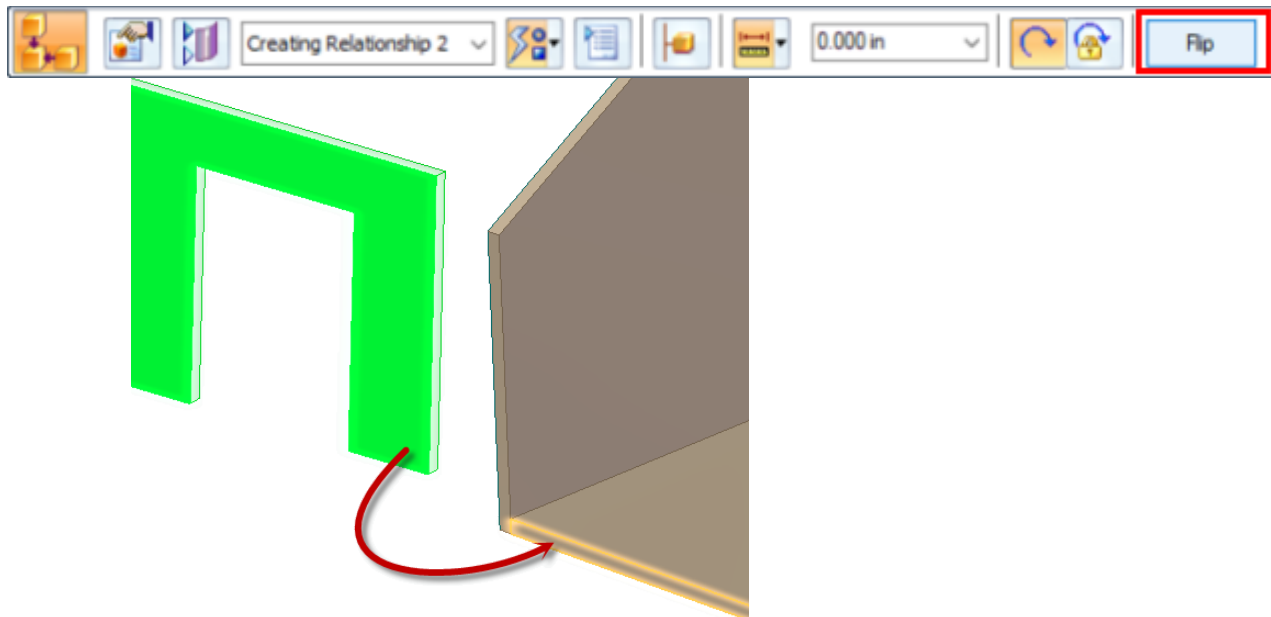


- Drag a copy of the Side panel from the PathFinder into the window and repeat the previous steps for the other side of the bird house.
 - The Teal faces should be pointing outward as shown.
- Save the assembly as “Bird House”

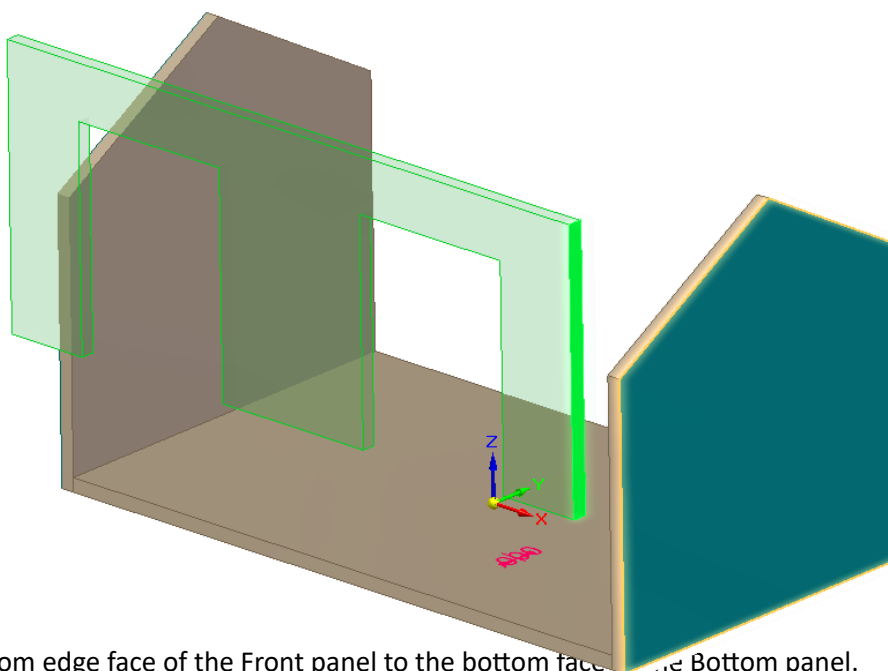


Adding the front and back panels

- Open up the parts library and drag in “Bird House Front”.
 - Select the near face of the Front panel and select the long edge face of the Bottom panel.
 - Initially this will “Align” the faces.
 - Select the Flip button to create a “Mate” between the parts.

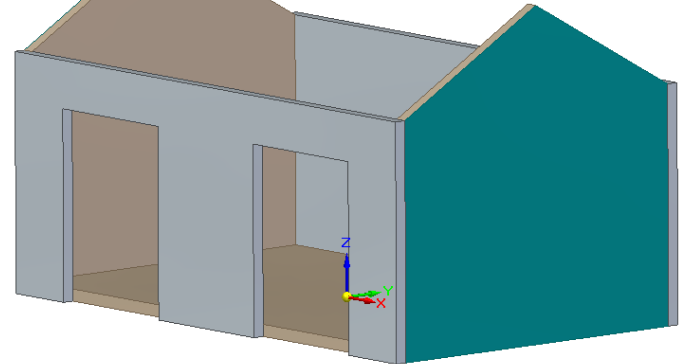
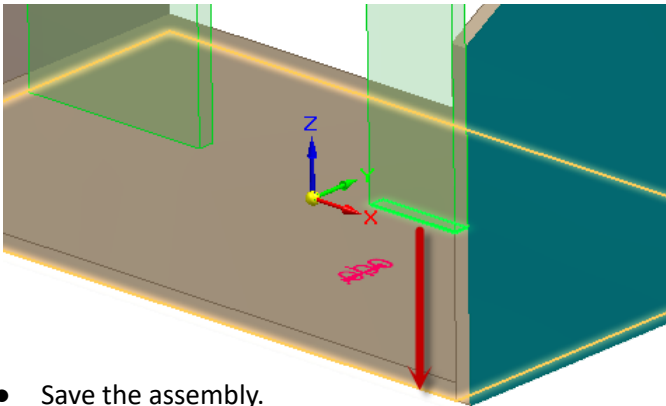


- Align the side edge face of the Front panel to the Teal side of the Side panel.



- Now align the bottom edge face of the Front panel to the bottom face of the Bottom panel.

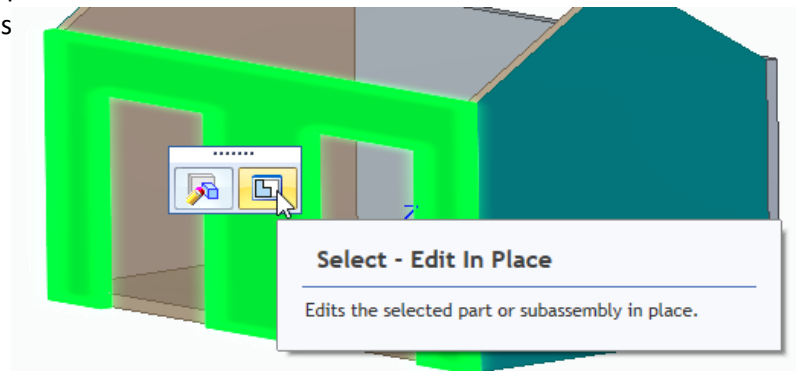
- Repeat the above step to achieve the same results for the back panel.
- The assembly should look like the image on the right below.



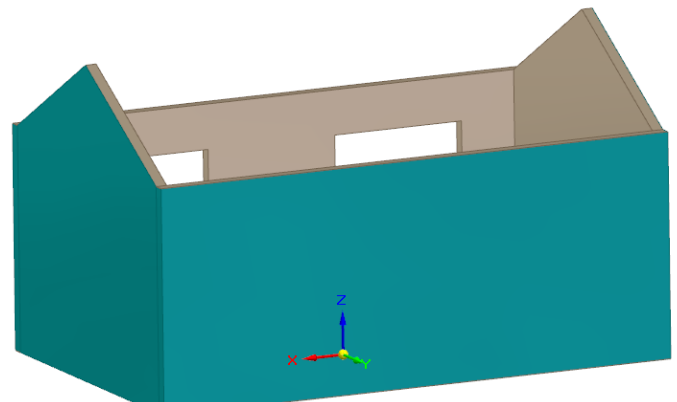
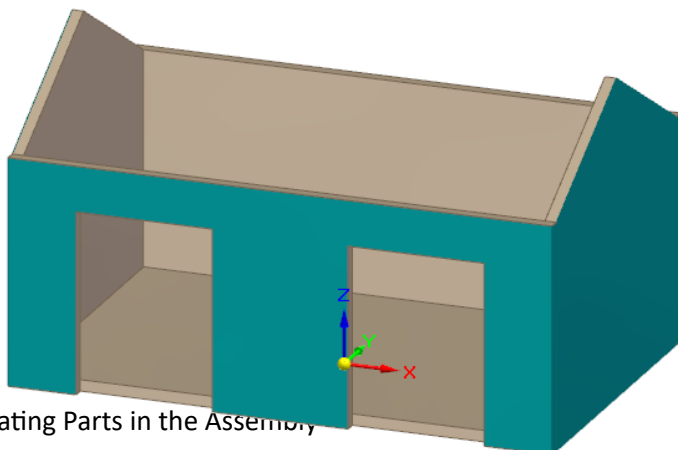
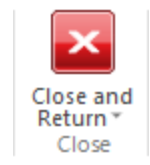
- Save the assembly.

Edit parts in an assembly

- To edit the Front panel, select the panel in the graphics window.
 - A small dialog box will appear with 2 icons
 - Select the icon to “Edit in Place”.



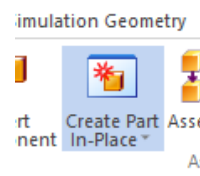
- Assign “Wood, Pine” material property to the part by right mouse clicking on “Material (None)” and selecting it from the top of the list.
- Use Part Painter to apply “Teal” to the front and short sides of the Front panel.
- Click the “Close and Return” button to exit the part editor and return to the assembly.
- Now do the same for the Back panel, making sure to paint the outside faces.
- Save the assembly.



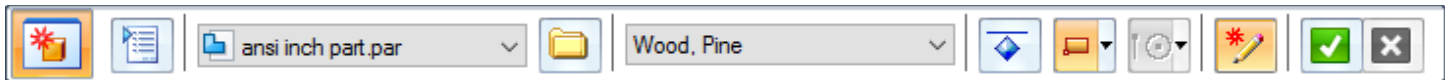
Creating Parts in the Assembly

- On the Home tab select “Create Part In-Place”.

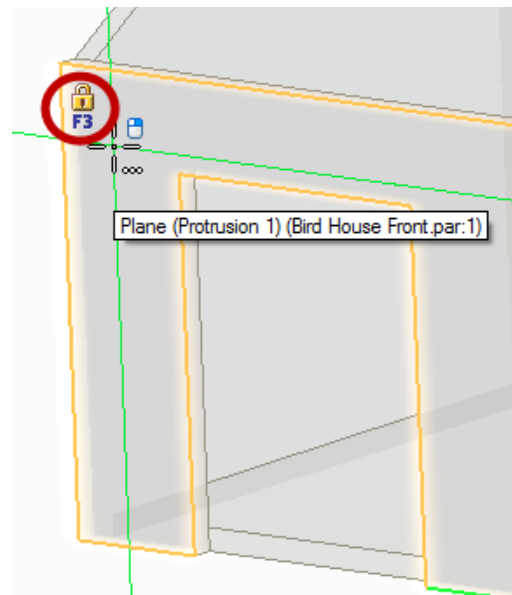
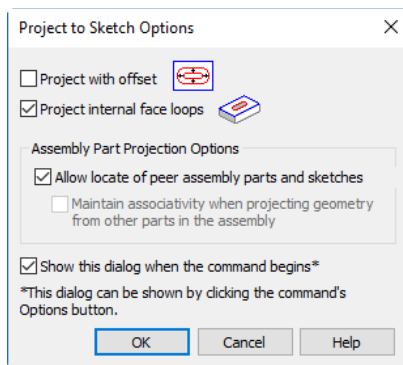
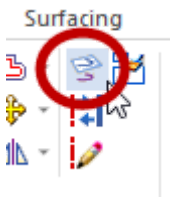
Tutorial: Design a Bird House in Solid Edge



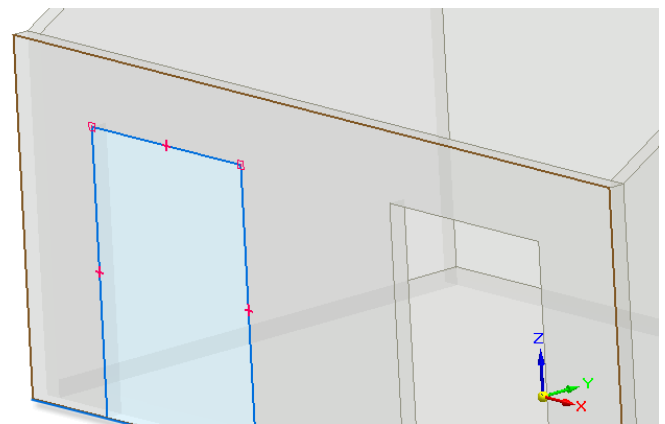
- Make sure the default options are selected:
 - Active assembly location
 - Coincident with assembly origin
 - Create component then edit in-place
- Click OK to dismiss.
- Make sure the template being used is set to “ansi Inch part.par” and set the material to “Wood, Pine” from the drop list.
- Click the green check box and save the part as “Bird House Door”.



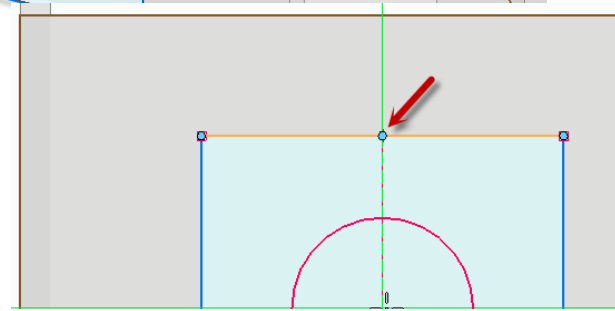
- Notice that Solid Edge immediately places you in the part mode because of the option to “create and edit in-place” was selected.
- Select “Project to Sketch” command on the home tab.
 - Hold down the Shift key to enable selecting faces from background parts.
 - Select the Front face of the Front panel and select the lock icon to lock onto that plane.
 - Accept the default Project to Sketch Options and dismiss the dialog box.



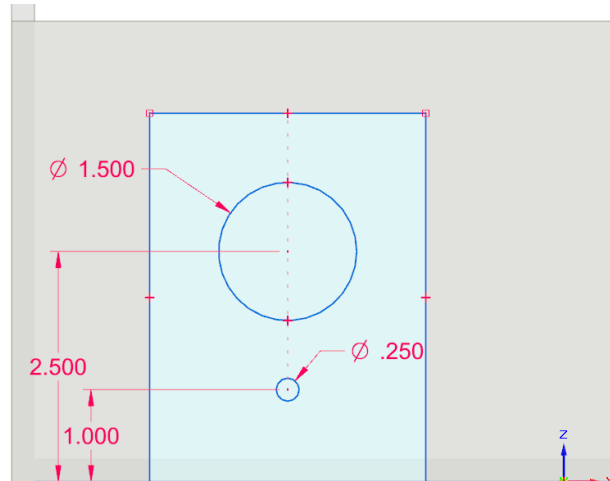
- Select all 4 edges of the left cutout in the Front panel to project onto the plane.
 - Note lower edge is projected from the Bottom panel.



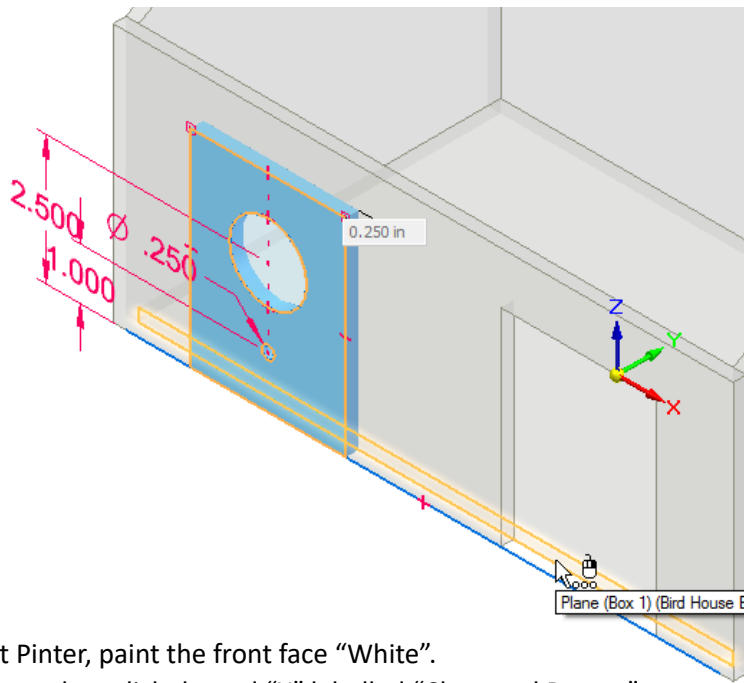
- Now a region has been created that can be extruded to create the door, but first, let's add two circles.
 - Select the “Circle by Center Point” command and key-in 1.5 in. for the diameter.



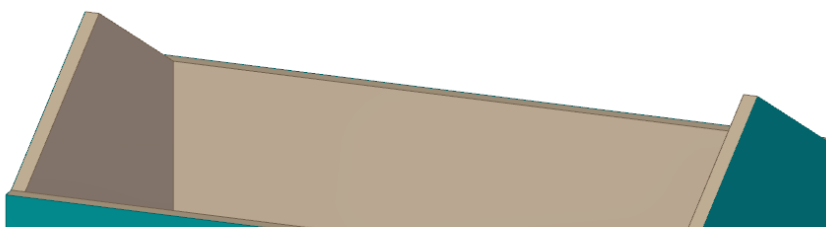
- Touch the top sketch edge with the cursor to highlight the midpoint and drag the circle straight down to establish a vertical relationship to the midpoint.
 - Click to place.
- Add a Smart dimension from the bottom edge to the center of the circle.
 - Click to place the dimension and key-in 2.5 in. to locate the circle vertically.
- Using the same method, add a .25 in. Diameter circle centered horizontally and 1 in. from the bottom edge.



- Press CTL+I.
- Click the region outside of the large circle and in the QuickBar change the extent type to “From-To”.
- Select the long edge face of the Bottom panel (visible in the other cutout) as shown for the “To” face.

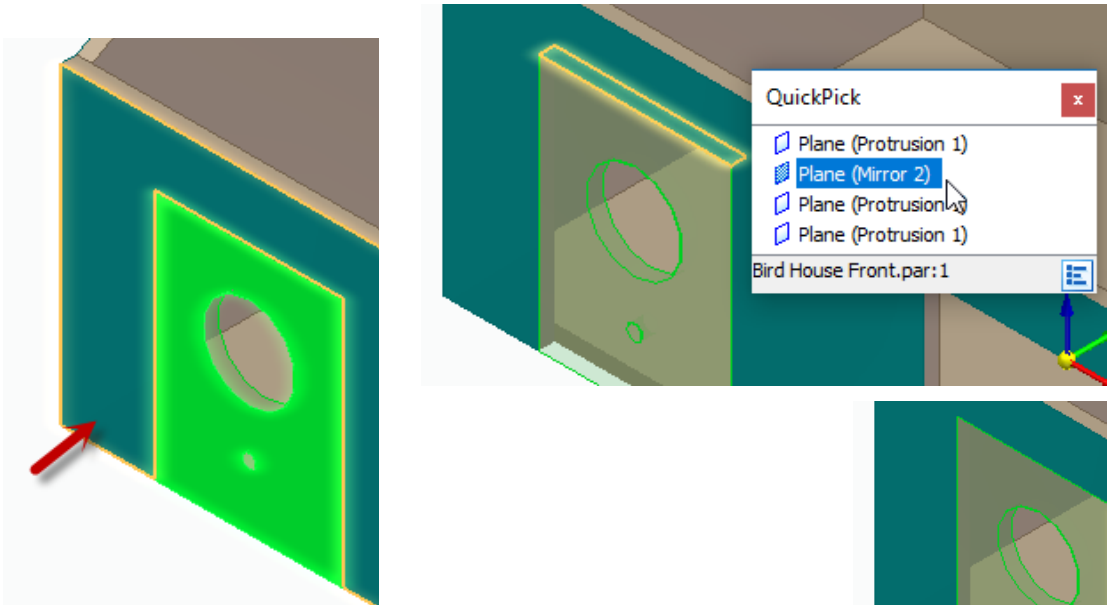


- Using Part Pinter, paint the front face “White”.
- Save the part then click the red “X” labelled “Close and Return”.

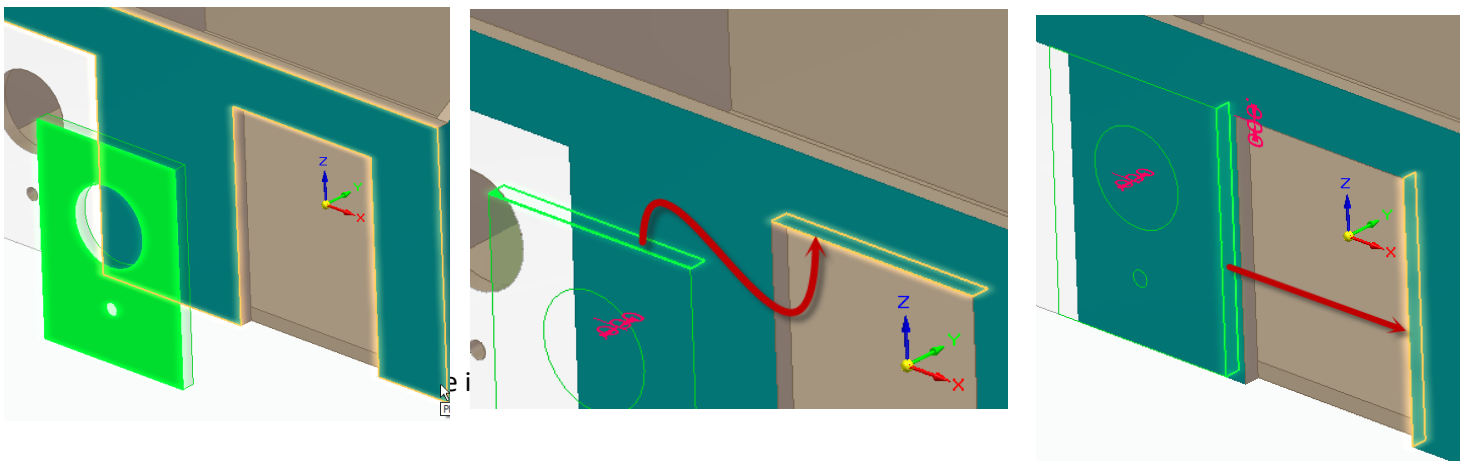
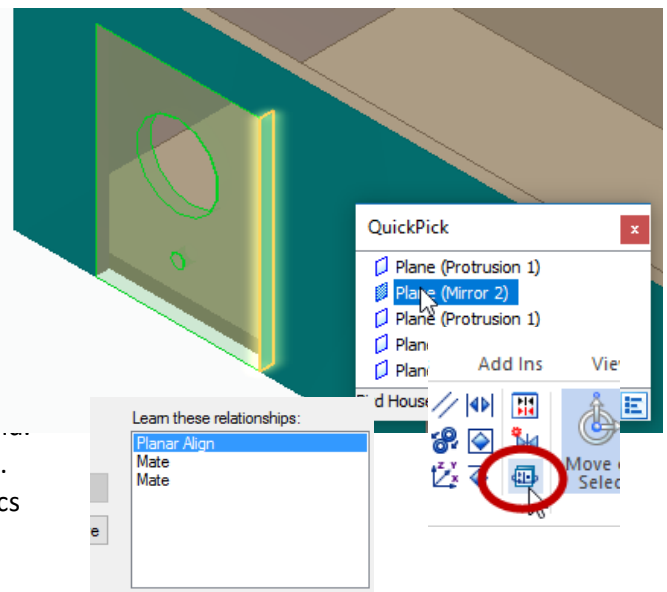


Relate the door to the assembly and add second door

- Even though part was created in-place, the part is not constrained to the assembly.
- Click on the door then select the “Assemble” command.
 - Align the front face of the Door to the near face of the Front panel.
 - Mate the top thickness face of the Door to the top thickness face of the opening in the Front panel.
 - Hover the mouse and right click; Quick Pick will likely be needed to select the correct face.
 - Mate the side thickness face of the Door to the side thickness face of the opening in the Front panel.
 - Hover the mouse and right click; Quick Pick will likely be needed to select the correct face.



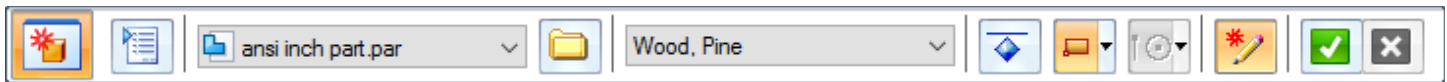
- Now that the first door has been added to the assembly, the second instance of the door is captured to make placing the second instance of the door easier.
 - Select the first door and run the Capture Fit command.
 - Accept the relationships by clicking OK on the dialog.
- From the Pathfinder, drag a copy of the Door into the graphics window.
 - Notice the first face highlights on the new door and all that is required is to select the corresponding faces on the Front panel.



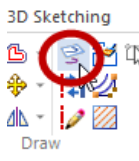
- Save the assembly.

Create wall divider

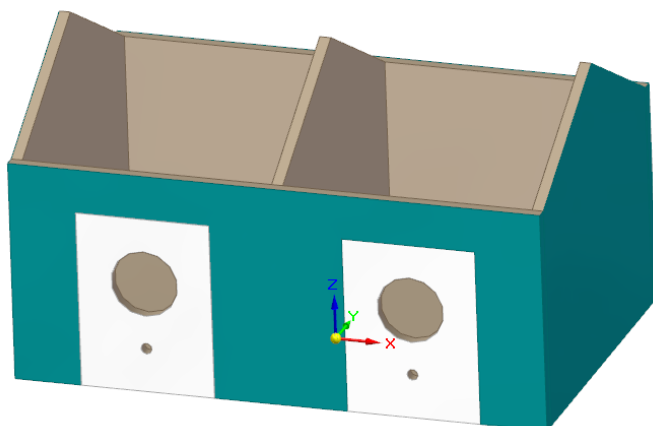
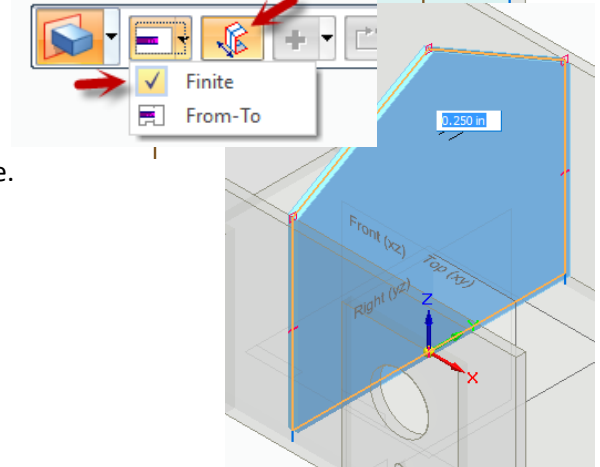
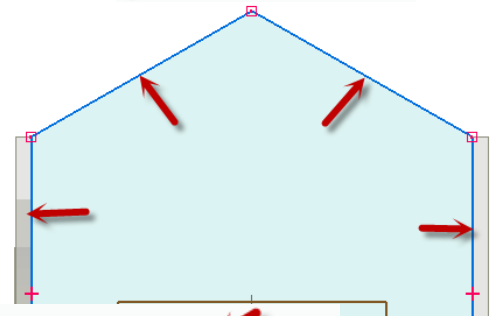
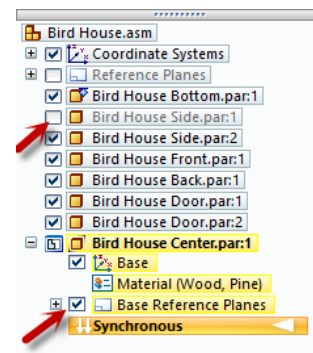
- Use Create Part In-Place to define a new ansi inch part using the default options and assign Wood, Pine. Save the part as “Bird House Center”.



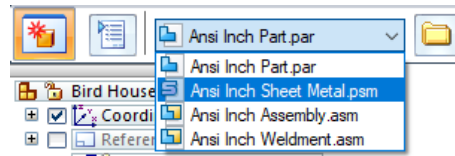
- In the PathFinder, display the part reference planes by clicking on the box next to Base Reference Planes.
- Hide the first instance of the Side panel by unchecking it in the Assembly PathFinder.
- Select the “Project to Sketch” command and select the Right (yz) plane.
 - Accept the default Project to sketch options.
 - Change the view to the right side using the view cube.
 - Select the angled top edges and side edges of the Side panel, and the top edge of the Bottom panel.



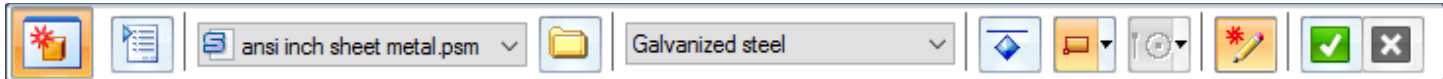
- Press CTL+I to transition to an isometric view
- Click on the region inside the sketch and select the arrow to begin the extrusion.
 - Change the extrusion type from “From-To” to “Finite”.
 - Select the symmetric button to the right of the extend type.
 - Extrude the part .25 in.
- Close and return back to the assembly.
- Show the side panel again.



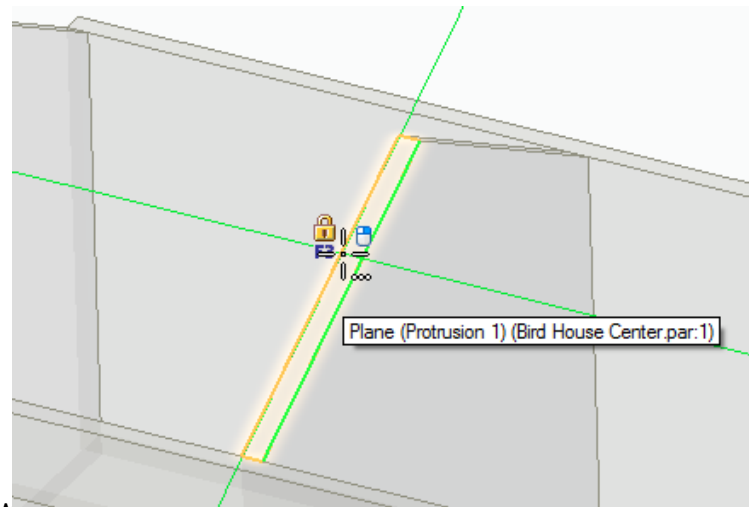
Create metal roof



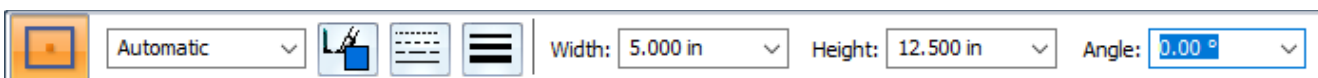
- Run the Create Part In-Place command and accept the default settings.
 - Change the template to “Ansi Inch Sheet Metal.psm”.
 - Set the material to Galvanized Steel



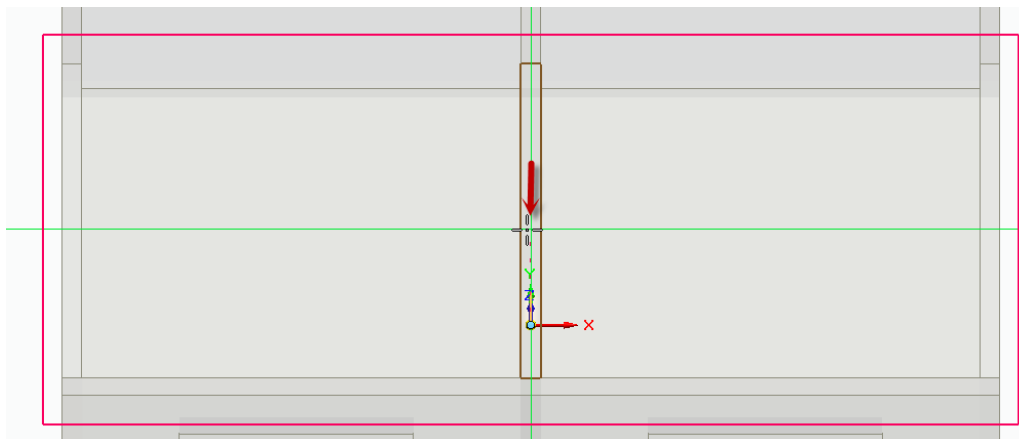
- Accept and save the sheet metal part as “Bird House Roof”.
- Using the Rectangle by Center command, press and hold the shift key down while hovering over the angled edge face of the Center panel..
 - When the F3 lock icon appears, press F3 to lock to that plane.



- Press Ctrl+H on the keyboard to make the sketching view go normal to the locked on plane.
- Make the width of the rectangle 5 in. and the height 12.50 in. on the QuickBar.

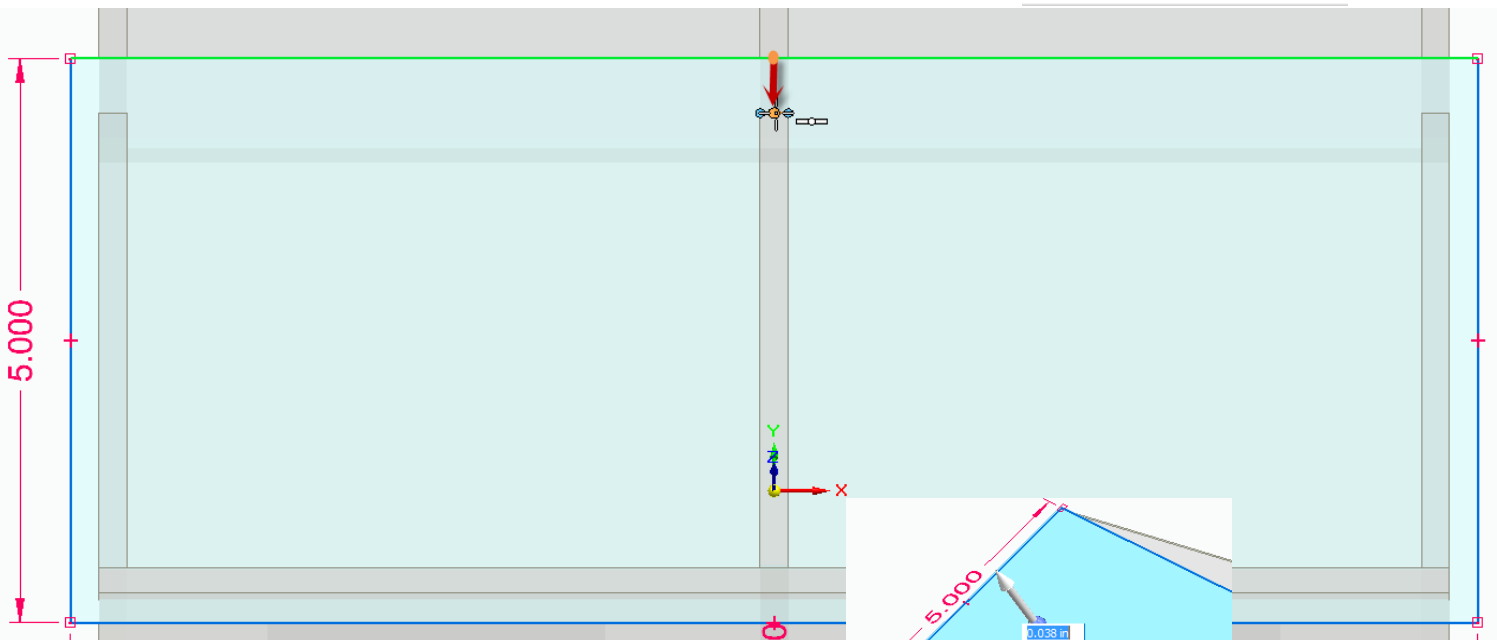
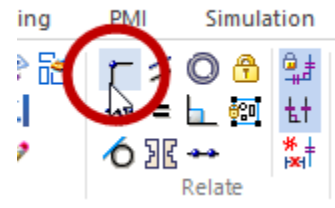


- Move the rectangle to the center of the coordinate system and align to the Y axis.
 - Click to place the rectangle and set the angle to 0.00°

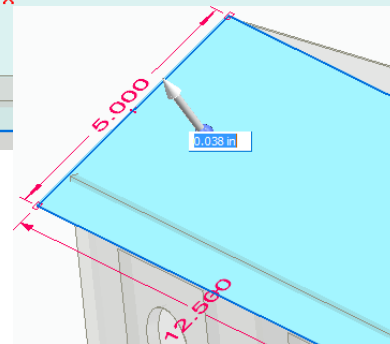


- On the Tools Tab, enable Peers edge locate.

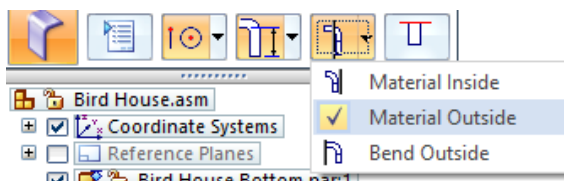
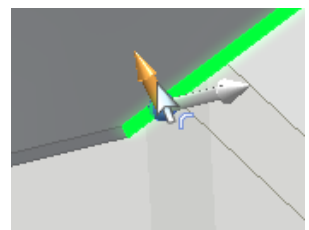
- On the Home tab, using the Connect relationship to connect the midpoint of the upper horizontal sketch line to the midpoint of the upper thickness edge of the Center panel as shown:



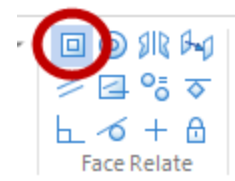
- Select the region and extend the sheet metal material thickness away from the assembly.



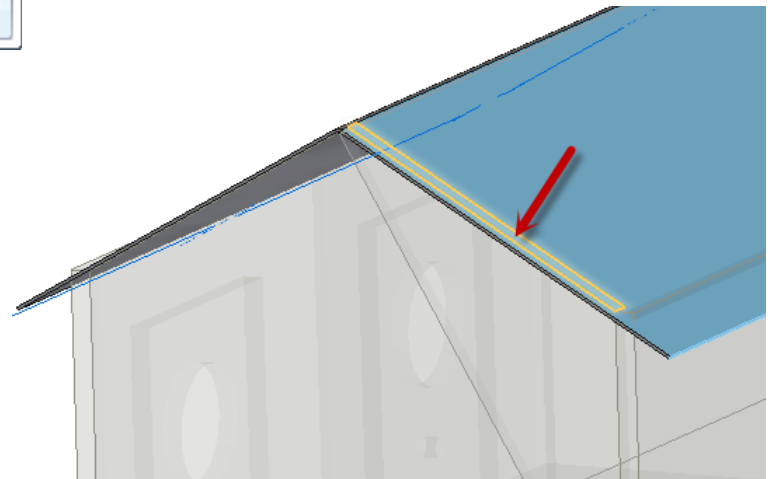
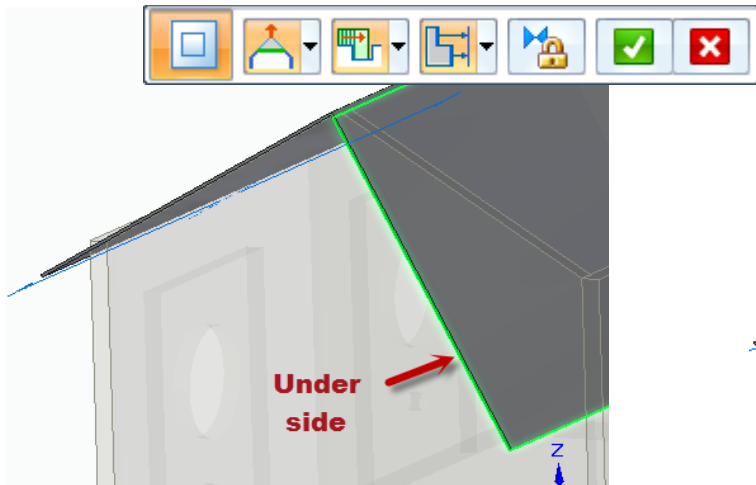
- Click the top edge (at the peak) of the sheet metal part and select the short arrow that is pointing perpendicular to the selected face to add a flange.
 - Change the material side to "Material Outside" in the QuickBar.



- Make the length 5.00 in.
 - NOTE the default 90° angle is not correct.
- From the Face Relate group on the Home tab, select the Coplanar relationship.

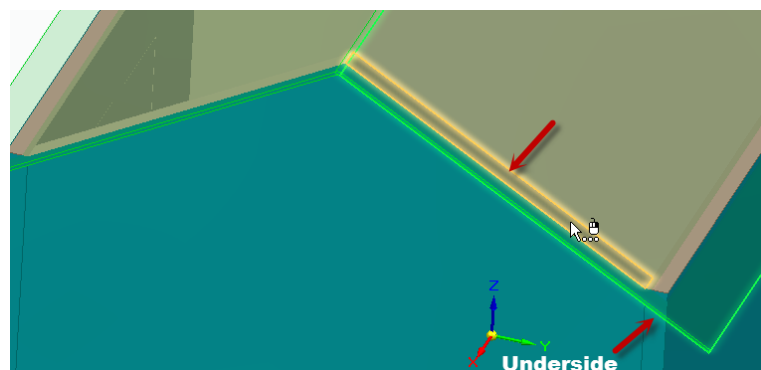
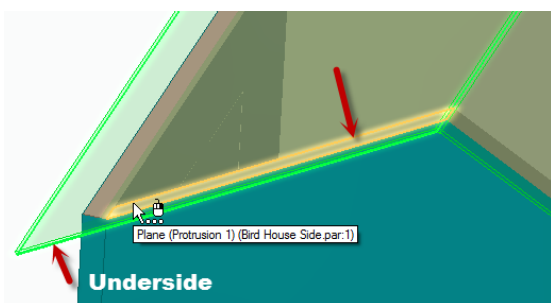
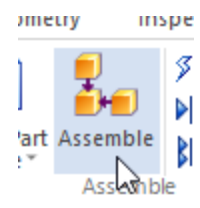


- Select the bottom face of the roof flange and accept with a right click or green checkmark.
- Select the top edge of the side panel to align with.



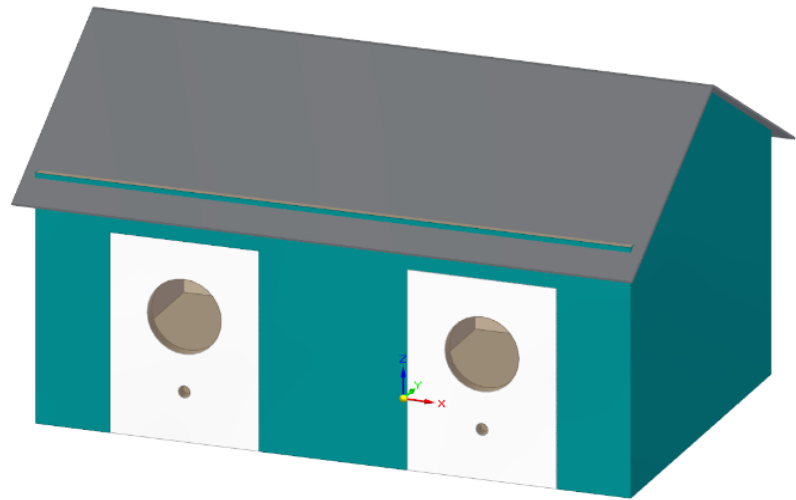
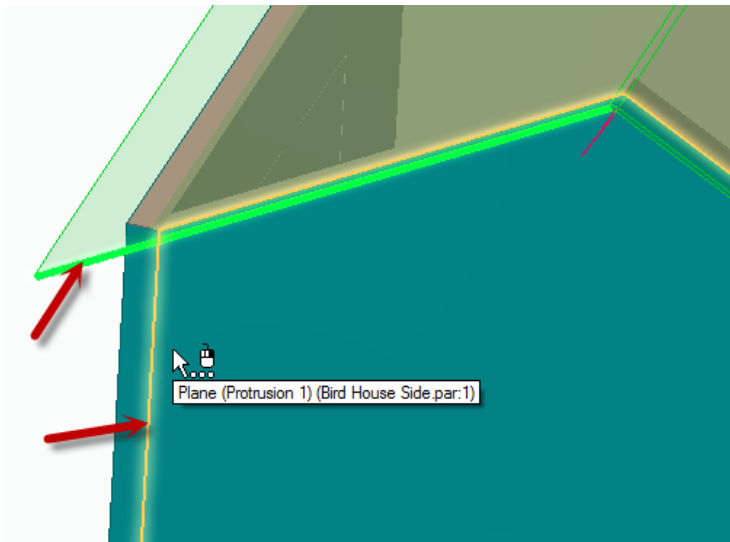
- Save the part and return to the assembly.
 - You will notice interference with the Front and Back panel which we will address later.

- Click on the roof and select “Assemble” on the home tab.
- Select the underside face on the roof tab to mate to top angled edge face of the Side panel.
- Repeat to mate the bottom face of the Roof flange to the other angled face of the Side panel.



- Set the offset distance of the Flashfit alignment relationship to -0.25in.
 - Select the end thickness face of the Roof first to create an offset alignment to outside Teal face of the Side panel.



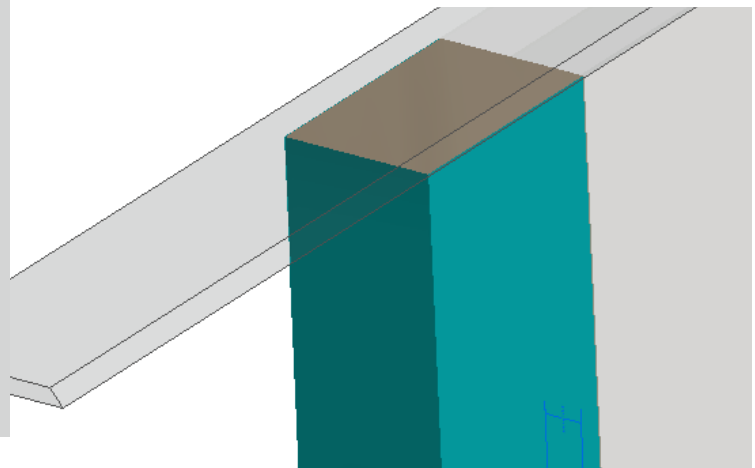
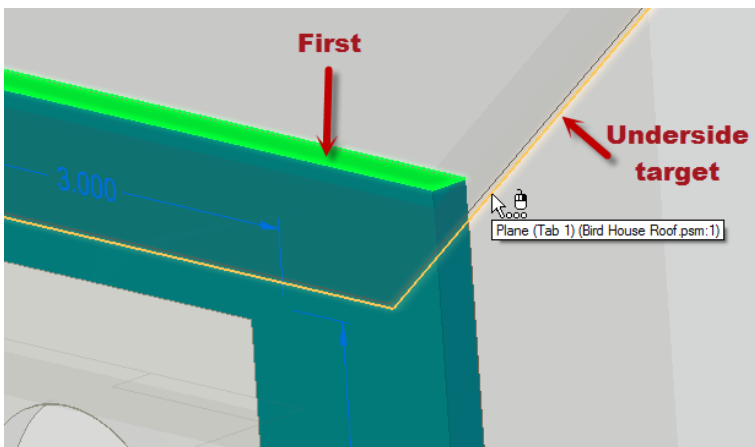
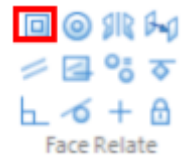


Fixing the Front and Rear Wall with the Roof

- Notice that the edge of the front and rear walls are protruding out of the roof.
- To fix this issue we will use the same Coplanar relate command that we used to change the angle of the Roof flange.
- Click on Bird House Front.
 - Select the icon on the right on the menu to edit the part.



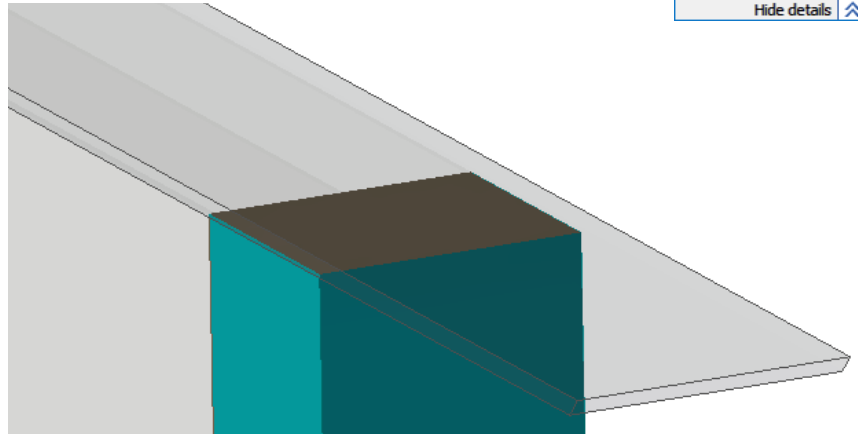
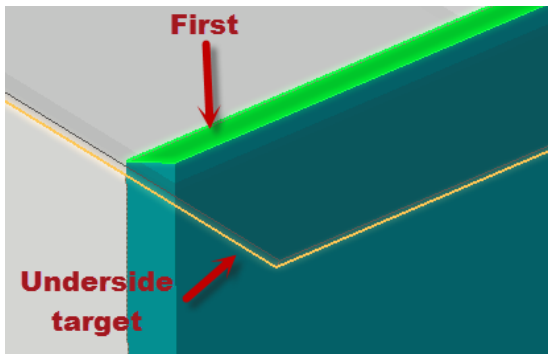
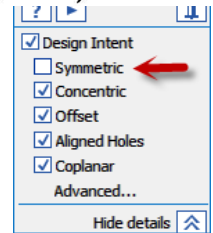
- On the Home tab, run the "Coplanar" command from the face relate section.
 - Select the top edge face of Front panel and right click or press enter to accept.
 - Select the underside face of the roof as the target.
 - Close and return to the assembly.



- Click on Bird House Back.
 - Select the icon on the right on the menu to edit the part.



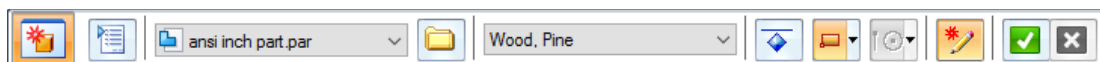
- On the Home tab, run the “Coplanar” command from the face relate section.
 - Disable the symmetric Design Intent by clicking “S” or unchecking Symmetry in the dialog.
 - This will prevent the bottom edge of the panel moving symmetrically when the top edge is angled.
 - Select the top edge face of Back panel and right click or press enter to accept.
 - Select the underside face of the roof as the target.
 - Close and return to the assembly.



- Return to the assembly and save the assembly.

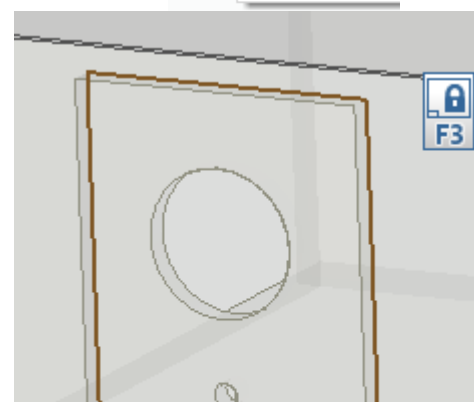
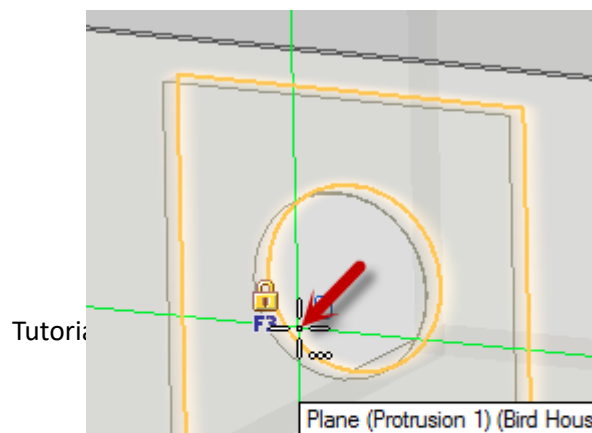
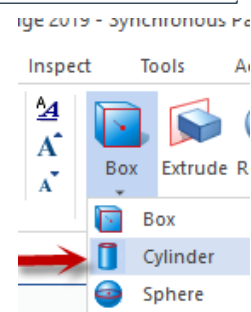
Create Bird Dowel Rod

- Create a new part in-place using the “ansi inch part.par” template and the default options.
 - Set the material to Wood, Pine

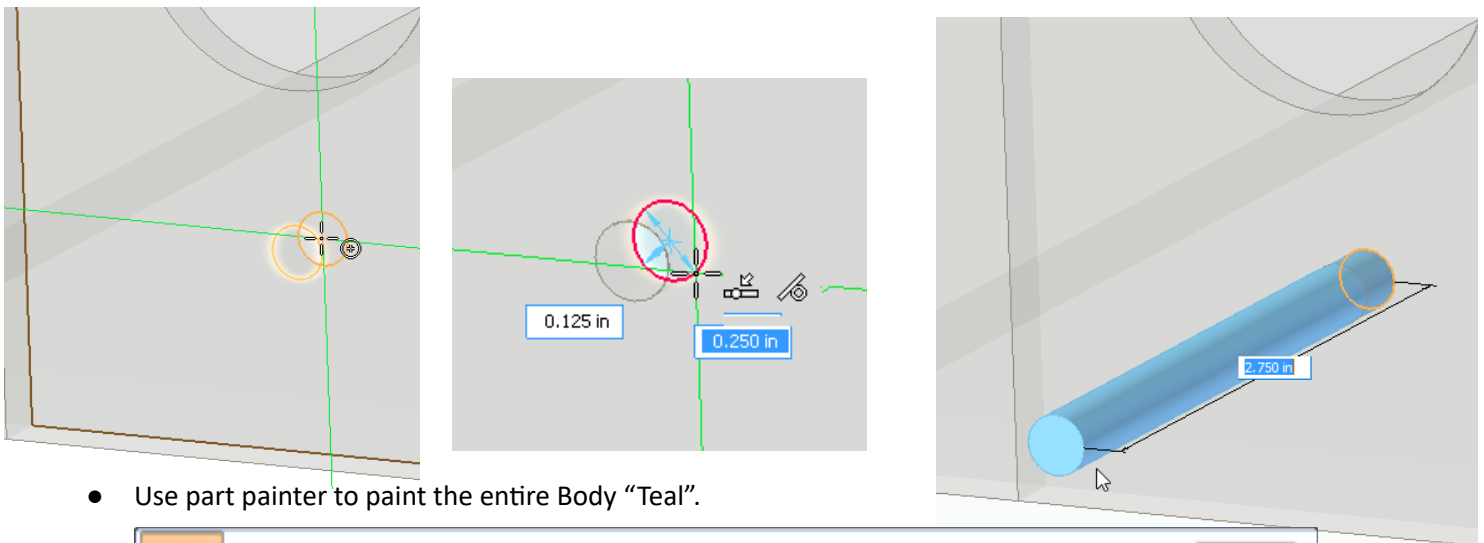


Save the part as “Bird House Dowel Rod”.

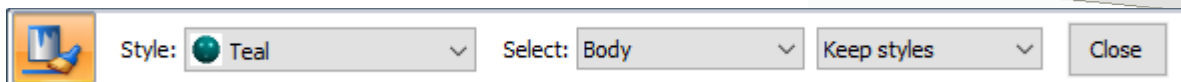
- Select the Cylinder command under the Box command pull-down menu on the Home tab.
 - While pressing and holding the Shift key, move your cursor to the inside edge of the Hole in the Door panel to select the INSIDE face to lock to.



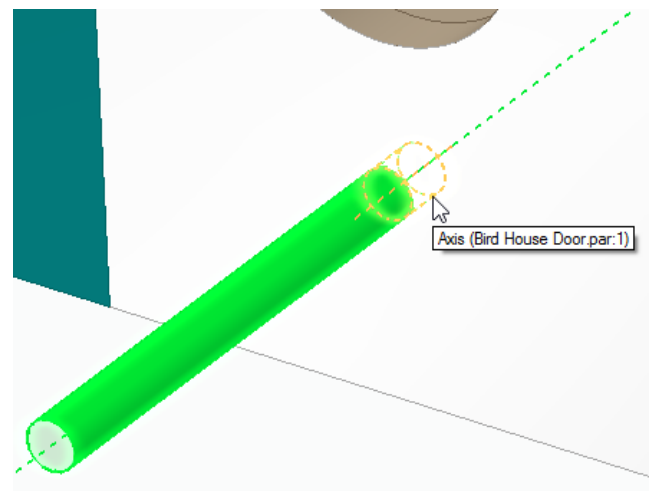
- Find the center of the small circle in the Door to locate the center of the cylinder and click.
- Set the diameter by locating the edge of the hole in the Door or by keying in .250 in. diameter.
- Extend the cylinder 2.75 in. to the outside of the assembly.
 - Note: If the extrusion is extending symmetrically, press the “shift” key to symmetry off.



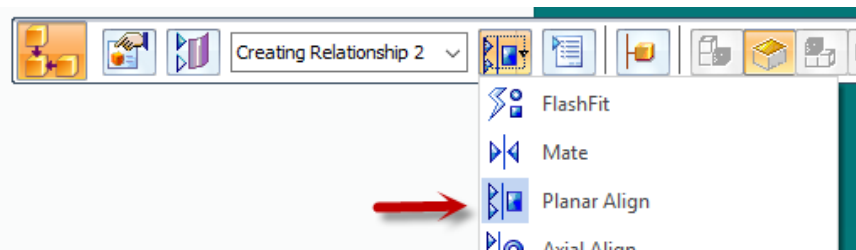
- Use part painter to paint the entire Body “Teal”.



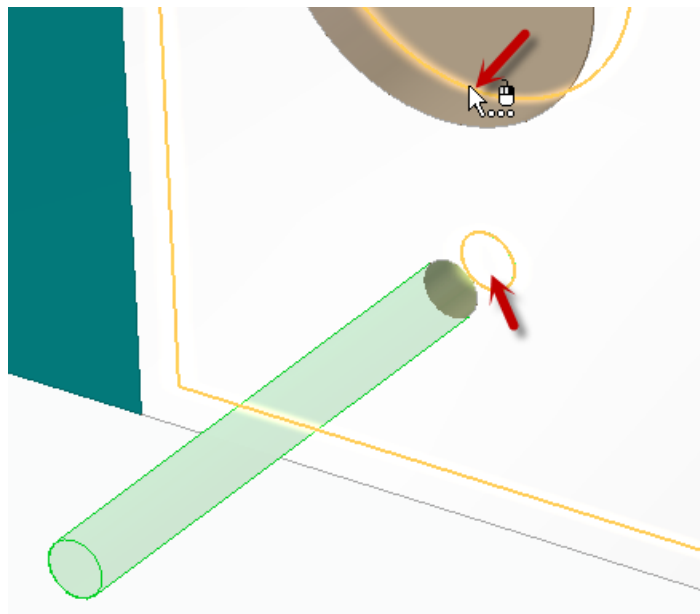
- Close and return to the assembly.
- Select the Dowel part and pick “Assemble” on the home tab.
 - Click the cylindrical shaft of the dowel to align with the bottom hole on the Door panel.



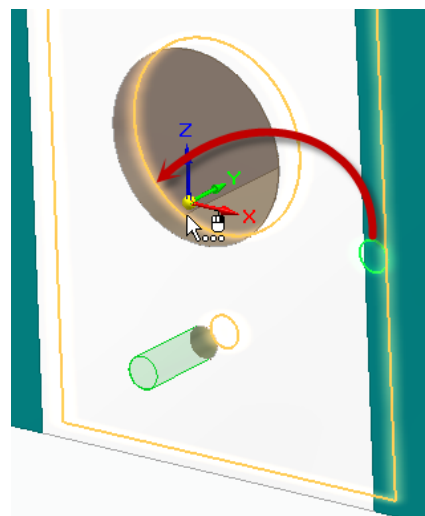
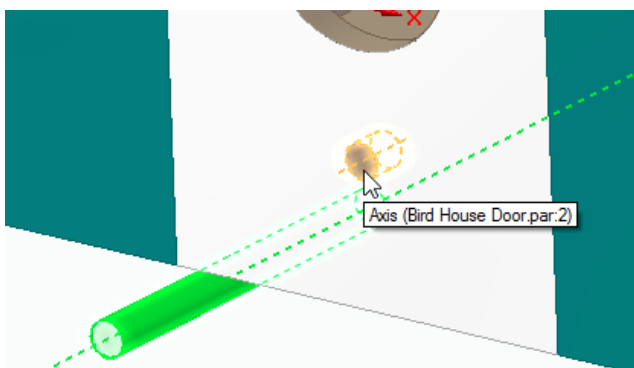
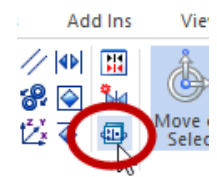
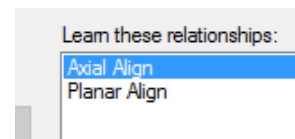
- Change the relationship type to Planar Align in the Assemble QuickBar.



- Select the end face of the Dowel shown and align to the inside face of the Door panel.
 - HINT: Hovering the cursor over the inside circular edge of the large hole in the Door makes it easier to select the inside face.



- Now that the first Dowel has been added to the assembly, the assembly relationships can be captured to make placing the second instance of the Dowel easier.
 - Select the first Dowel and run the Capture Fit command.
 - Accept the relationships by clicking OK on the dialog.
- From the Pathfinder, drag a copy of the Dowel from PathFinder into the graphics window.
 - Notice the cylindrical face highlights on the new Dowel and all that is required is to select the corresponding hole in the other Door panel.
 - Select the inside face of the other Door as the target for the planar alignment.



- Save the assembly.

