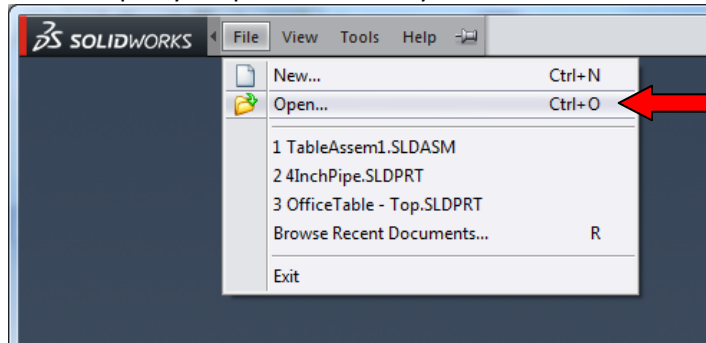


Directions for Creating 2D Drawings Using SolidWorks

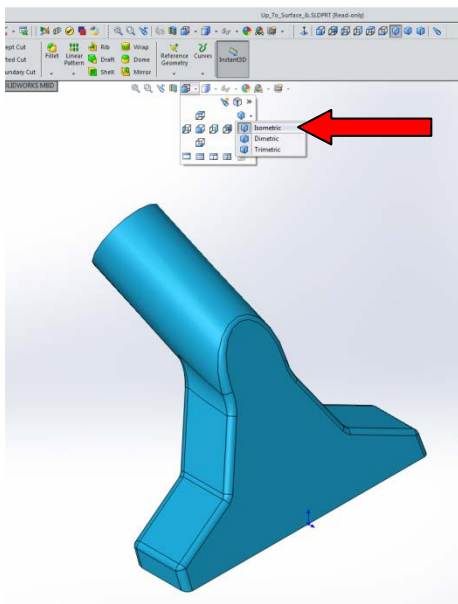
1. Launch SolidWorks



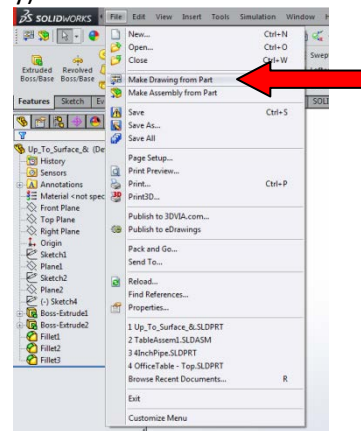
2. Open your part or assembly



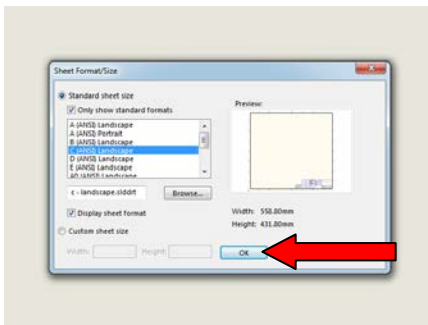
3. Place the 3D data in an isometric orientation



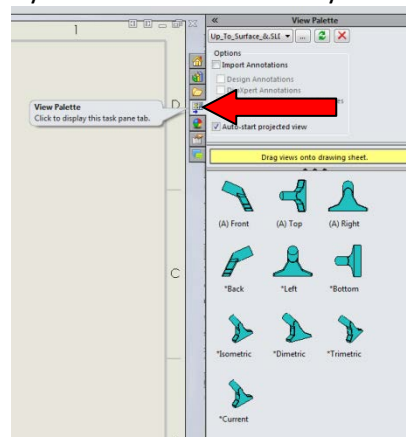
4. Click File → Make Drawing From Part (or Assembly)



5. Select C Size Landscape and click ok

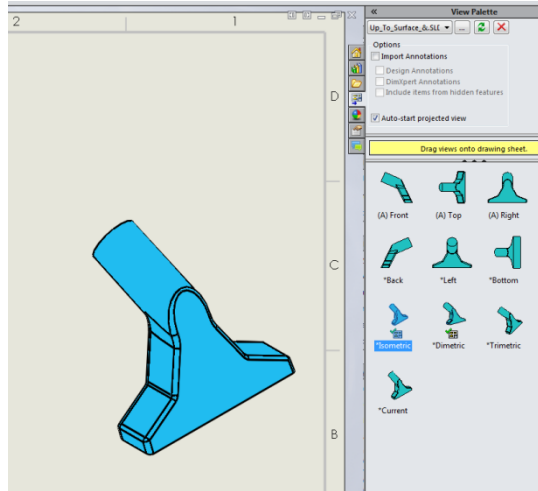


6. Click View Palette at the right side of the screen to display the selectable views of your data

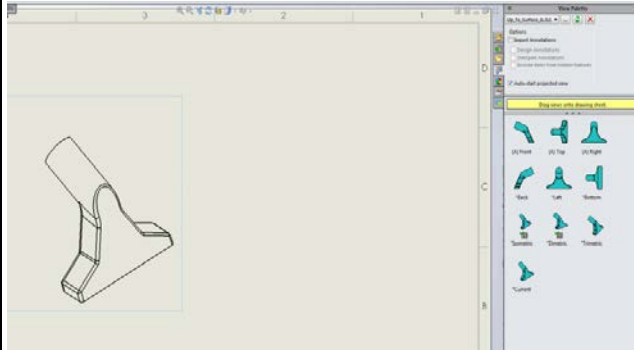


Directions for Creating 2D Drawings Using SolidWorks

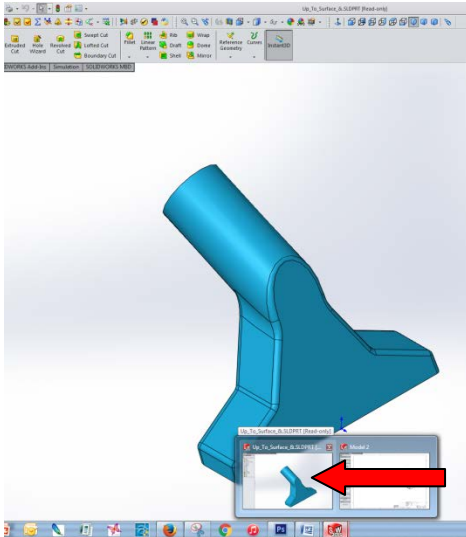
7. Click the isometric view and drag it into the drawing area



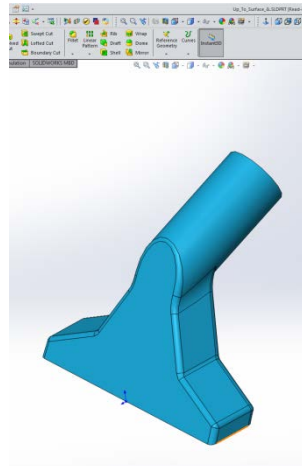
8. Release the left mouse button when you are happy with the location of the view



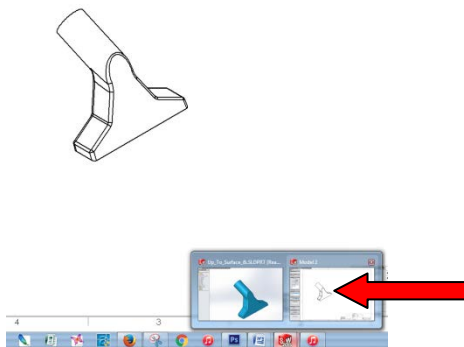
9. Go back to your part file



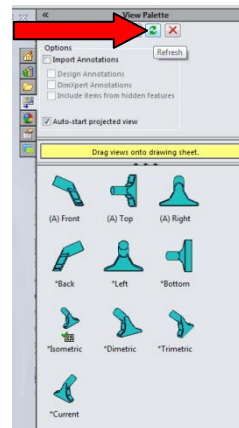
10. Rotate the part data to an opposite isometric orientation



11. Go back to your drawing file

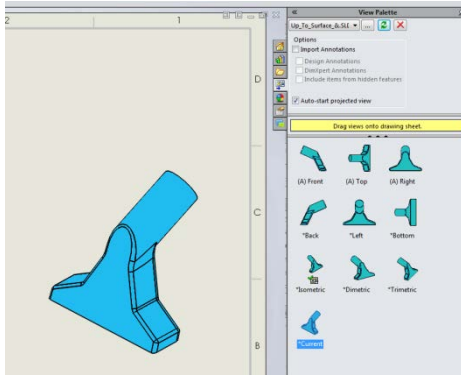


12. Click the View Palette refresh button

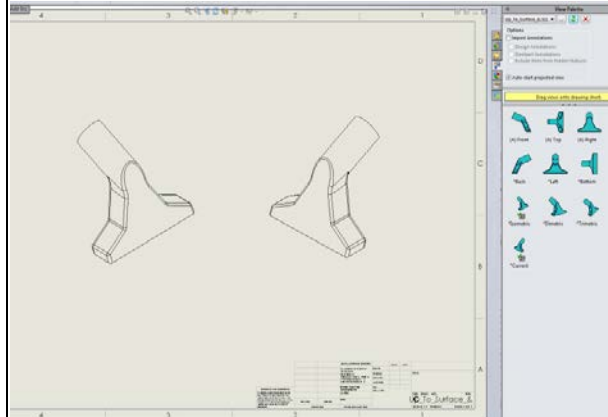


Directions for Creating 2D Drawings Using SolidWorks

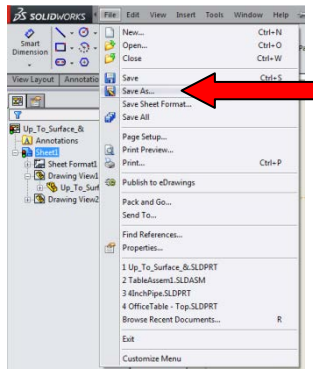
13. Click the current view and drag it into the drawing area



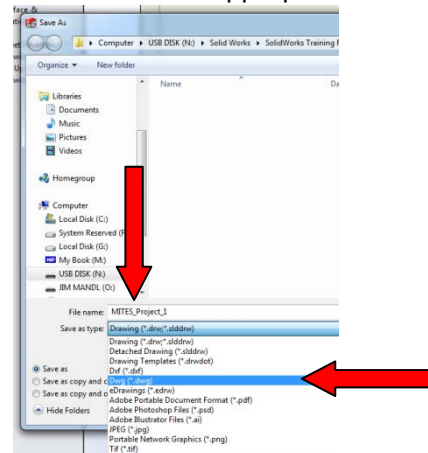
14. Release the left mouse button when you are happy with the location of the view



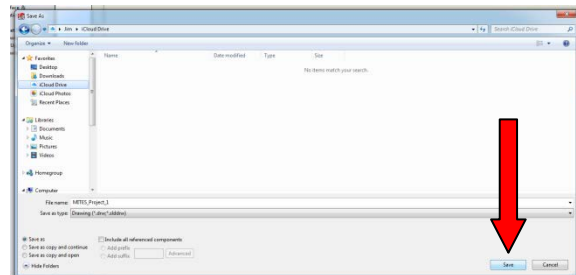
15. Click file → Save As



16. Change the file type to DWG file and change the file name to an appropriate name



17. Click Save



Congratulations!
You have successfully created a drawing from your 3D SolidWorks data!