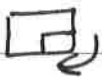


How to create a technical drawing

- Open the part in SolidWorks
- Go to File - Choose 'Make Drawing from Part'
- Choose 'A3' for paper size
- Click & Drag the Front view to place it on the sheet.
- Then click above & to the right, to create the top and side views
- Now there should be three views on the sheet ✓ green tick
- Click on the front view
- Set 'Display Style' to Hidden Lines Visible ✓ green tick
- If centre marks are already in place for studs
 - then add centre marks & centre lines for hidden detail only.
- Click on the Annotation tab
- Click on the 'Centre Mark' button
 - Select the studs (if not already marked)
 - Select the outer hidden detail circle ✓ green tick
- Click on the 'Centre Line' button
 - Select the two inner hidden detail lines on the front view & a centre line should appear
 - Repeat on right view ✓ green tick
- Click on the 'Smart Dimension'
- Need to dimension the drawings:- BEWARE Don't over dimension
- There should be 14 linear dimensions
 - 1 diameter dimension
 - 2 radii dimension
 - 1 angle dimension✓ green tick

- Zoom in on the title block
- Right click on the title block, then choose 'Edit Sheet Format'
Title block should turn blue (and drawing will disappear)
- Click where you want to type and then right click edit text
 - Put your name & date
- Then click anywhere on the sheet
- Click on the icon at top right hand side of page that looks like this 
- Zoom out 
- Save as .pdf file and submit via Moodle