Rollco

2024-04-25

Introduction to Weldments

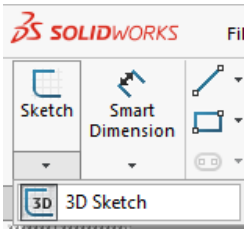
We use the SolidWorks weldment feature when creating our aluminium constructions. The advantage of using weldment is that an automatically created cut list is generated, and the lengths can be easily adjusted by adjusting the length of the sketch lines. To get started, follow these four steps:

1. Create a new part file

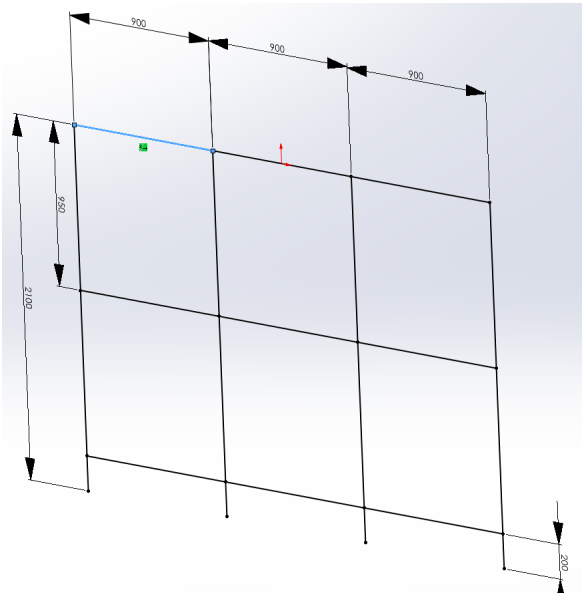
Open a new part file.

2. Create sketch

Create sketch lines that will be applied with profiles from our library. Creating the sketch in a 3D Sketch is recommended. The advantage of this is that you can draw the lines in all three dimensions.



(Tips: In 3D Sketch, to switch between the different planes, press the "TAB" button. To fully define the sketch, click on the line and choose to add a relation to follow either the X, Y, or Z plane.)



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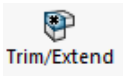
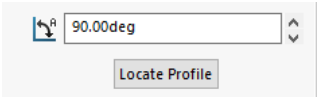
3. Create a structural member

After the sketch is accepted, click on: 

and choose a profile. Thereafter, apply them by clicking on the sketch lines. If you can't click on a line, add a new group from the left panel (because a new group needs to be added every time a new direction of extrusion is made). If you want multiple profile types in the construction, add a new structural member feature. The interconnections depend on how you have drawn the sketch lines. You can also use the trim/extend feature: 

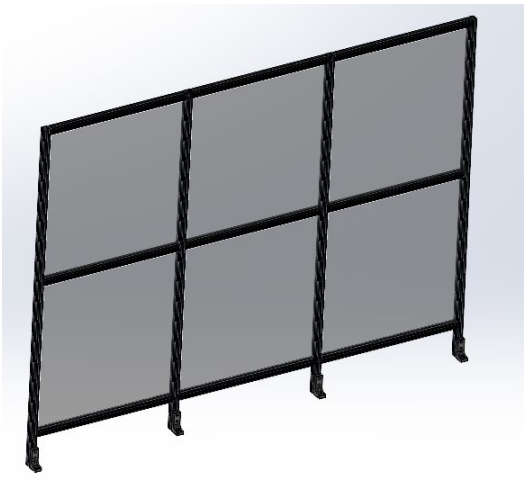
If you use covered-sided profiles and want to rotate the profile, use rotation angle in the left panel. 

4. Saving and export

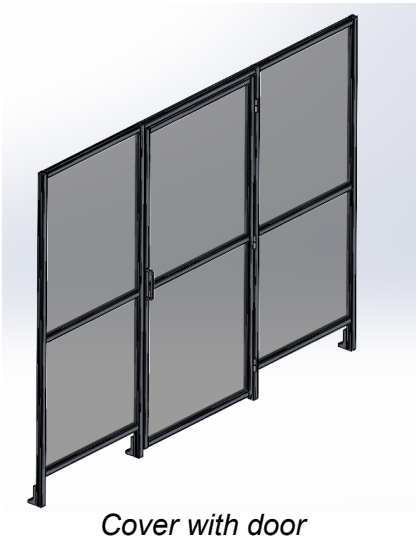
When finished, we prefer you to send the file to us as a part format (.prt) or export the assembly file as a "pack and go" map. This can be done by searching "pack and go" in search by command.

Templates

We have created some examples of how it can look, and which can be used as templates.



Cover Wall



Cover with door