# Creo Parametric Tips and Tricks

Volume IV – Top Down Design

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Sharing Dimensions in Data Sharing Features

- If Data Sharing Features like Copy Geometry, Shrinkwrap, Merge, and Inheritance bring geometry only, then the underlying dimensions cannot be shown on a 2D drawing for the target part.

- If the source model contains 3D Annotations, the Data Sharing feature can copy those too.

- Process:

  - In the source part, create 3D Annotations for the dimensions you want to transfer

  - If creating a Publish Geometry feature, use the **Edit** button to include selected 3D Annotations

  - If creating a Copy Geometry feature manually, click the References tab and use the Edit button to include 3D Annotations

- Then the 3D Annotations will be available to show either in the target part or a 2D production drawing
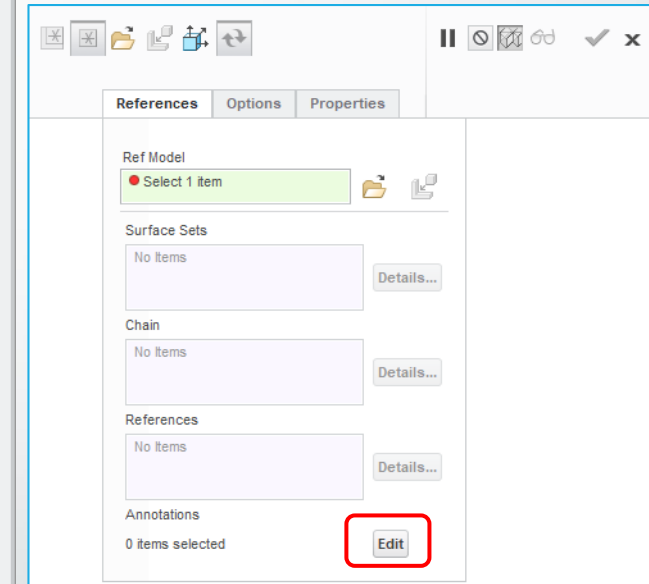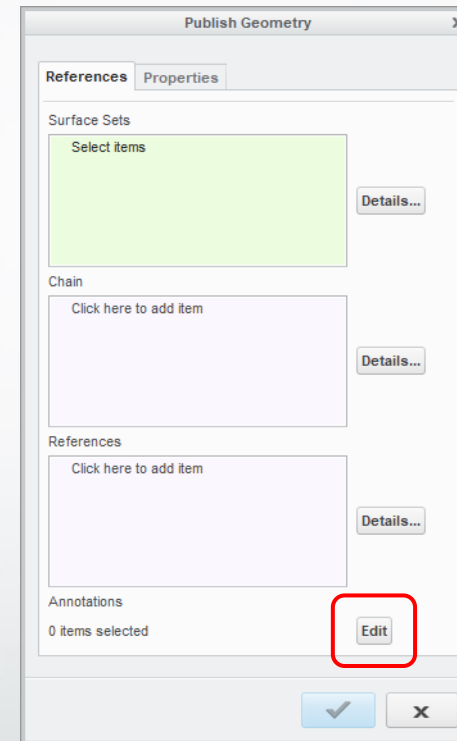
Using Copy Geometry with Patterns

- If the source model for a Copy Geometry feature contains a pattern, there's a trick to getting that geometry to be recognized as a pattern in the target part.

- Process: Create a Copy Geometry feature that contains references (surfaces or datums) that belong to a pattern in the source model.

- In the target part, either:
  - Create a new feature that references the copied geometry and reference pattern the new feature; or,
  - Reference pattern the Copy Geometry feature.

- Also works for Patterns created via Pattern Recognition in FMX – convenient for imported geometry.


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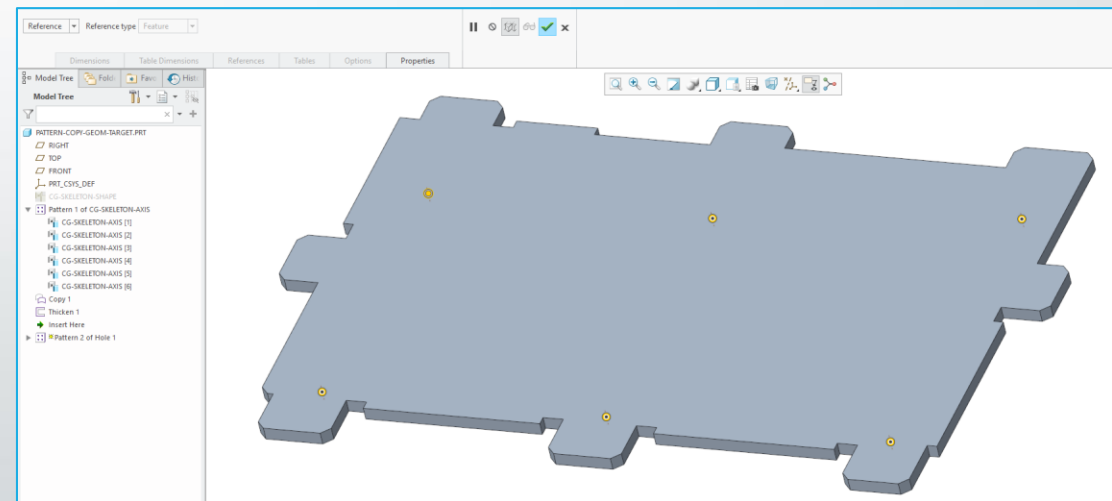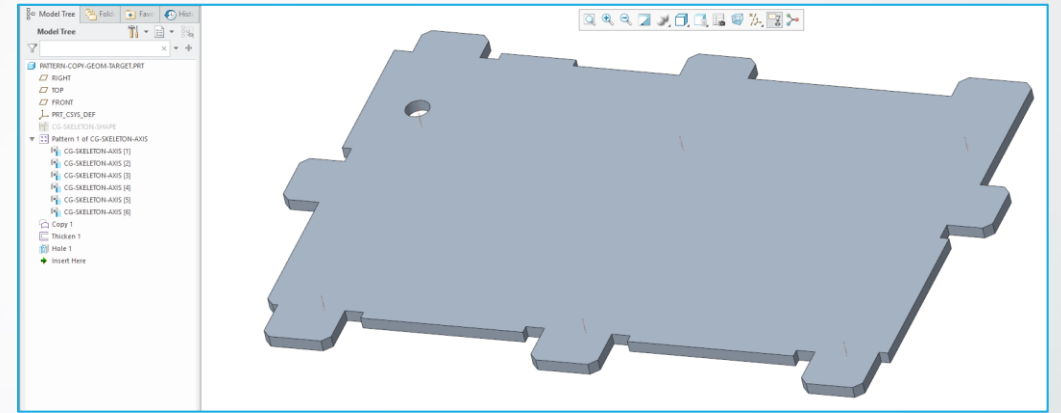
Shrinkwrap with Manual Collection

- Copy Geometry features can reference a single part model only.

- Shrinkwrap features can reference an entire assembly.

- The Manual Collection options allows you to select surfaces and datums from multiple components – which makes it like a Copy Geometry feature for multiple parts.

- This technique is especially useful for Routed Systems like cable harnesses and piping.
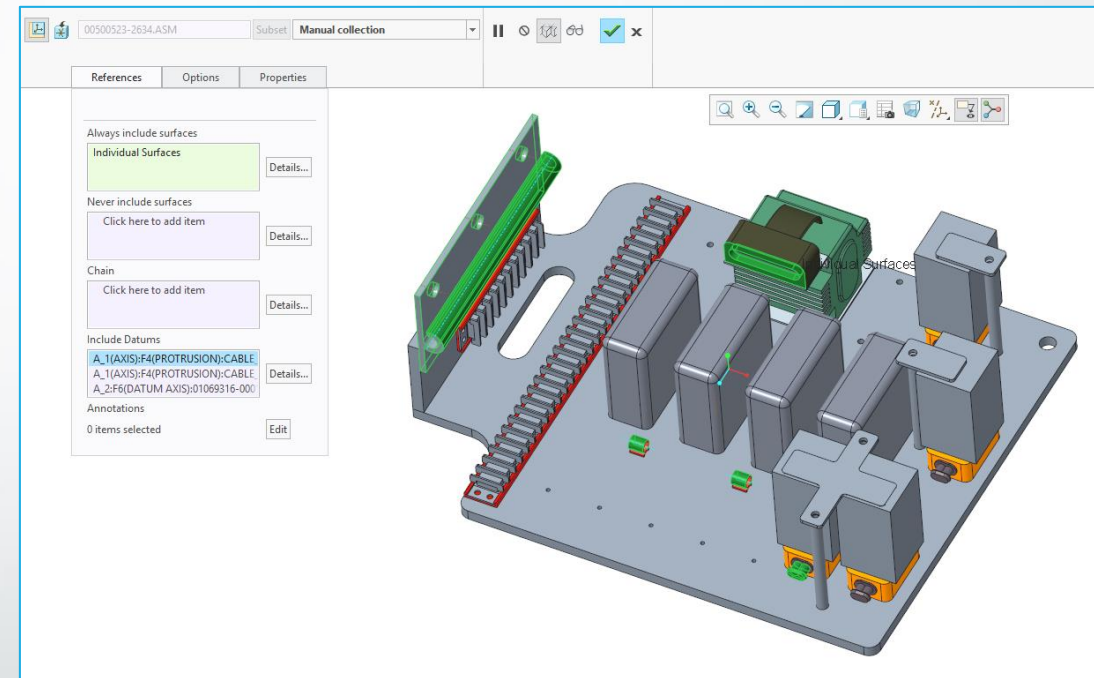
Notebook Table of Dimensions and Parameters

- In a Notebook, you define dimensions and parameters to control multiple components in the assembly

- You can create a table with a Repeat Region that reports the dimensions and parameters automatically

- Process: start with a 2 x 2 table

  - Enter header information in top row

  - Define a simple Repeat Region in bottom row

  - Bottom left: double click and enter lay.param.name

  - Bottom right: double click and enter lay.param.value

- Values can be changed directly in the table
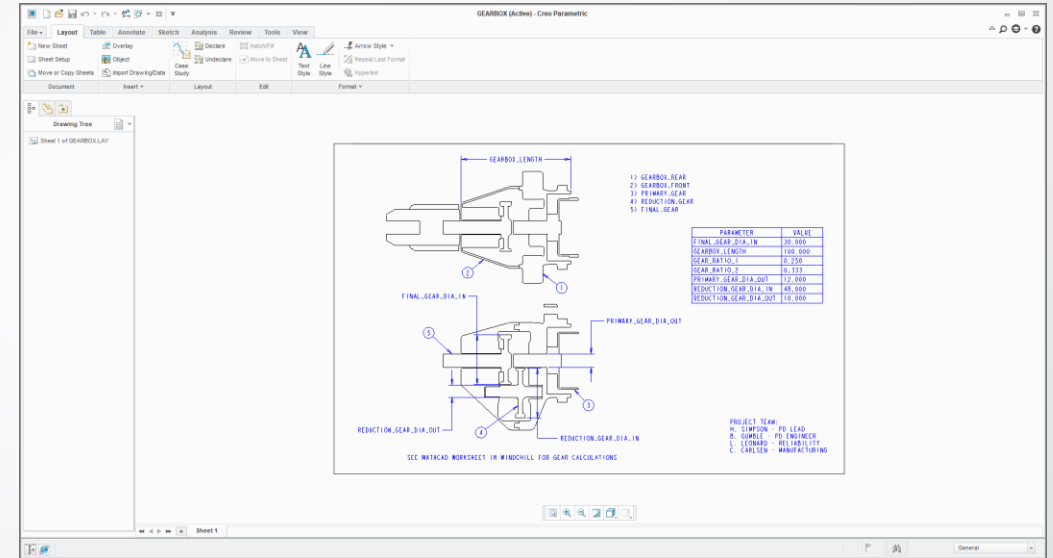
PARAMETER	VALUE
lay.param.name	lay.param.value

| PARAMETER	VALUE
FINAL_GEAR_DIA_IN	30.000
GEARBOX_LENGTH	100.000
GEAR_RATIO_1	0.250
GEAR_RATIO_2	0.333
PRIMARY_GEAR_DIA_OUT	12.000
REDUCTION_GEAR_DIA_IN	48.000
REDUCTION_GEAR_DIA_OUT	10.000

Sheetmetal Offset Command

- Convenient technique when using Top Down Design with Sheetmetal assemblies.

- Create necessary reference geometry in the Skeleton.

- Use a Copy Geometry feature to share it with target Sheetmetal part.

- Use the **Offset** command in Sheetmetal mode (different from Part mode Offset command) to create solid geometry.

  - Specify thickness

  - Use offset distance = 0.0

- Can add bends to sharp edges from **Options** tab.
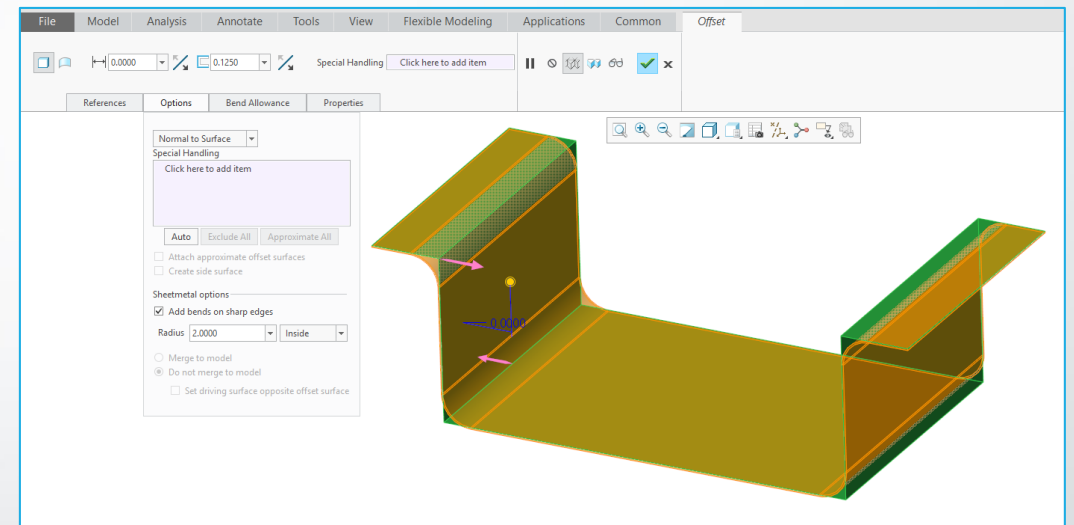
Convert Standard Part into Skeleton

To convert a standard part model into a Skeleton, perform the following steps:

1. Add the part to the assembly with the Default constraint using the **Assemble** command from the Component group.

2. Drag the part to the top of the Model Tree above the default datum planes. You might not be able to drag it above the first default datum plane. If so, simply drag the first default datum plane below the part.

3. Right click on the part and select **Replace**

4. In the Replace dialog box, select the radio button for **By Copy**.

5. Check the box for **Copy as skeleton**.

6. Edit the name for the Skeleton in the Name field.

7. Click **OK**.

Bonus topic; this appeared in Tips and Tricks Volume I
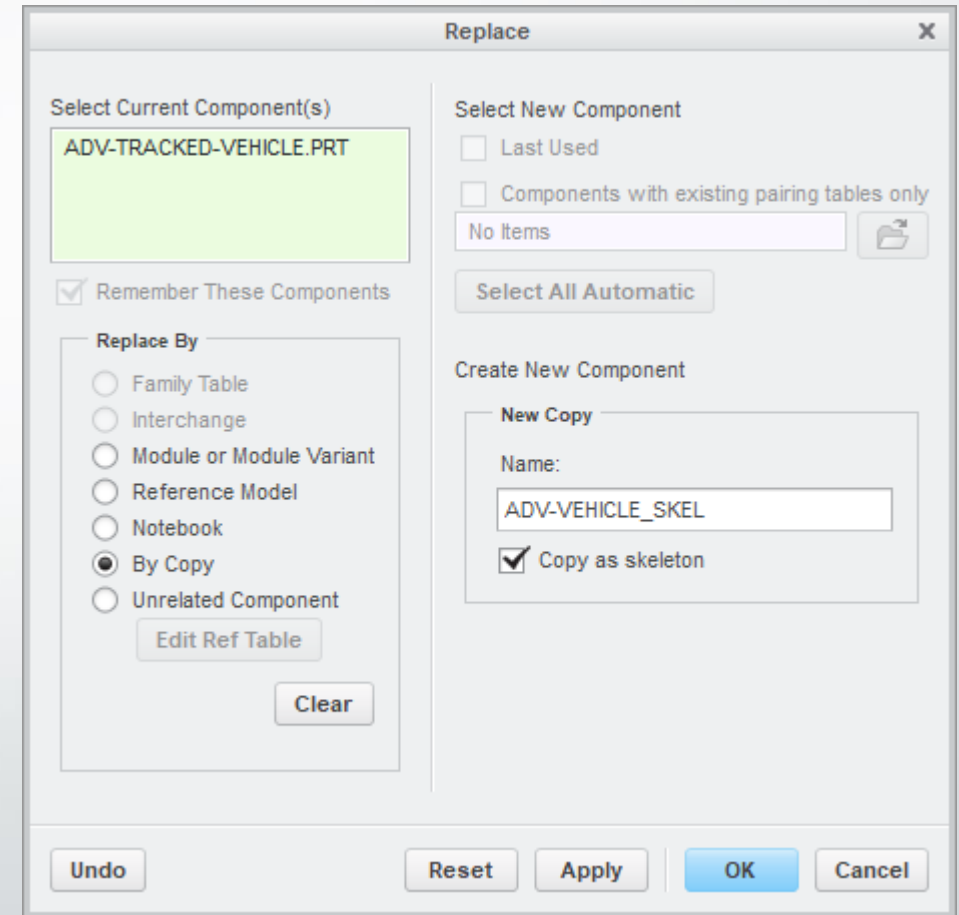
Thank You!

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