



Advanced Assembly Extension (AAX)

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Advanced Assembly Extension Overview

The Advanced Assembly Extension (AAX) is an additional module available for Creo Parametric. When most people think of AAX, they think of Top Down Design (TDD). However, AAX provides additional functionality beyond TDD.

This presentation will review the functionality available from Top Down Design, as well as additional capabilities provided for general assembly modeling and Options & Variants.

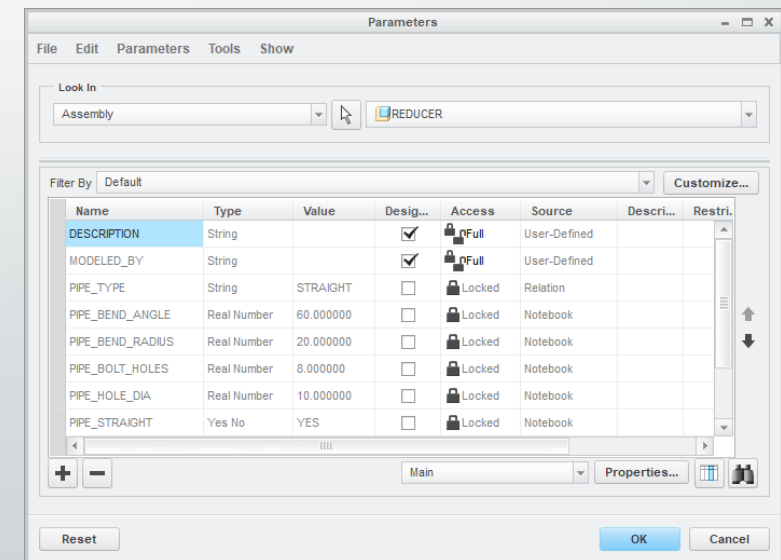
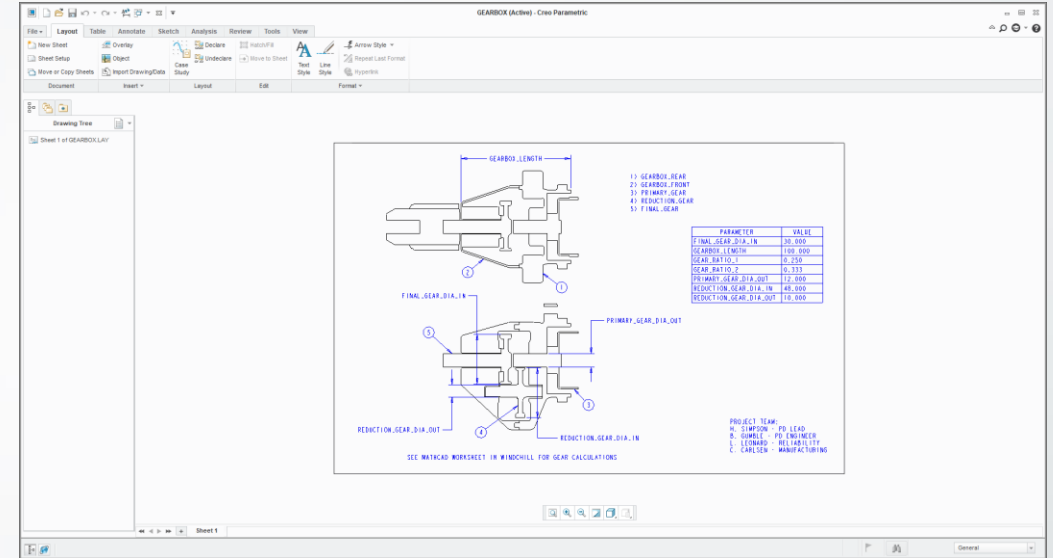


Top Down Design

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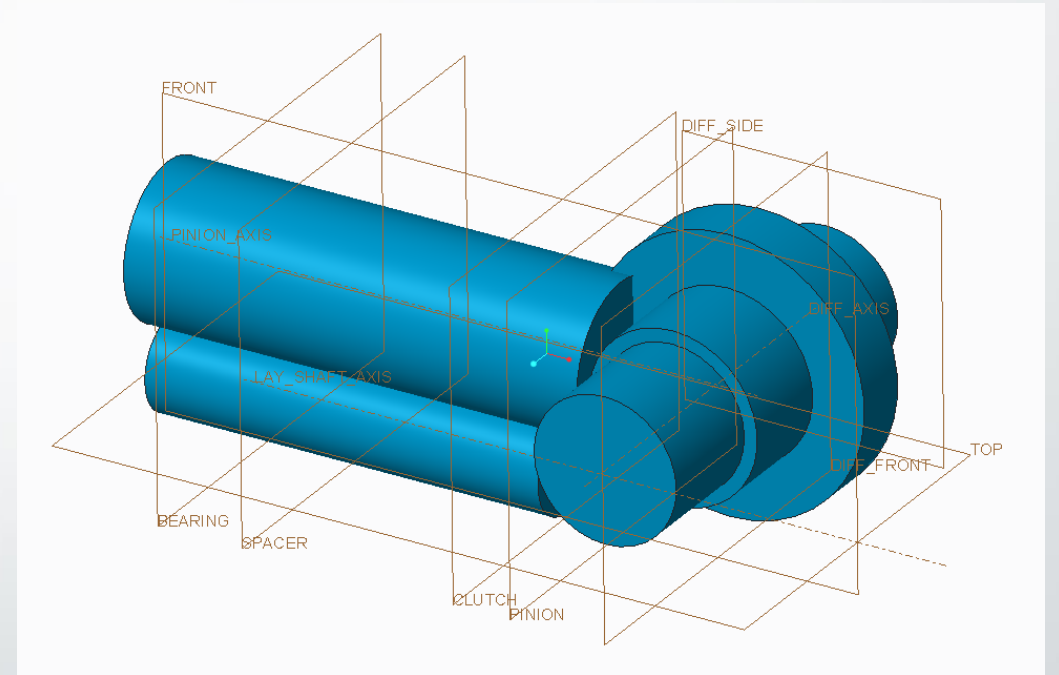
Notebooks

- Notebooks document a product or assembly's Design Intent.
- Notebooks contain important dimensions and parameters that affect multiple components in an assembly.
- Notebooks can also contain images, notes, tables, symbols, and balloons.
- Models are connected to Notebooks using the Declare command.
- Then you can write Relations that define local dimensions as a function of Notebook parameters.
- Notebooks are retrieved into session whenever a model declared to them is retrieved, ensuring the model always has the latest values.



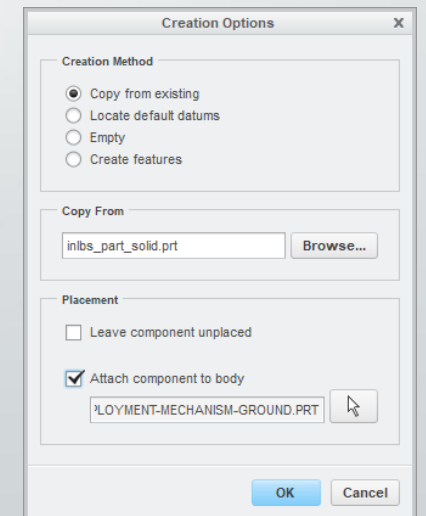
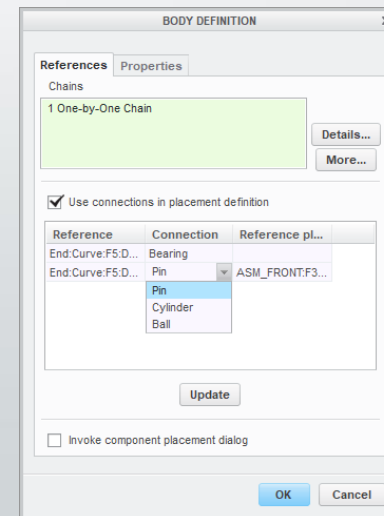
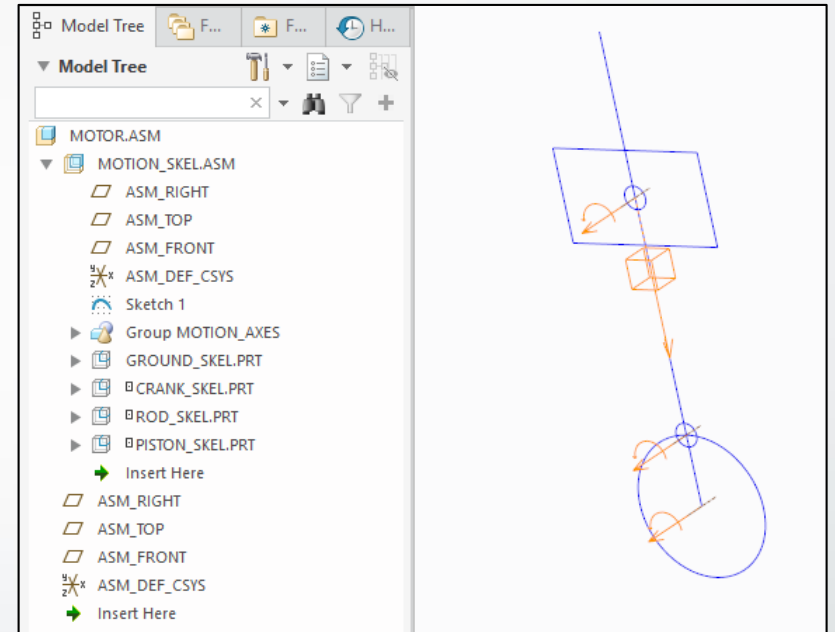
Skeletons

- Skeletons are special kinds of part models that contain geometry that drives multiple components in an assembly.
- Skeletons can contain surfaces, curves, planes, axes, points, and coordinate systems.
- Geometry is communicated from a Skeleton to target parts using Data Sharing features such as Copy Geometry, Shrinkwrap, and Merge/Inheritance.
- By sharing geometry from the Skeleton to target parts, updating the Skeleton can propagate changes to multiple components.

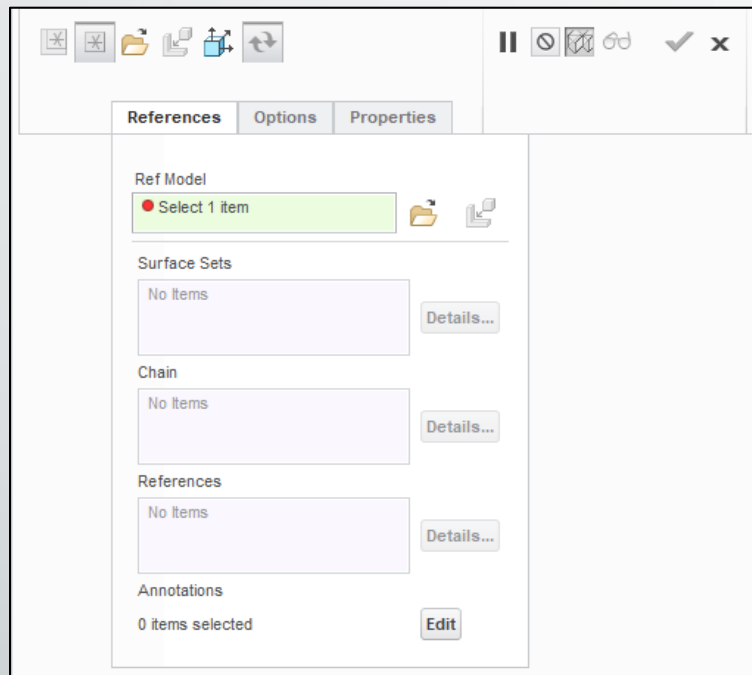


Motion Skeletons

- Motion Skeletons are used to apply Top Down Design techniques to mechanisms – assemblies in which components can move relative to each other.
- A Motion Skeleton is an assembly, whereas a standard design Skeleton is a part.
- Motion Skeletons contain:
 - A 2D sketch
 - A Ground component
 - Body parts with connections for the defined motion



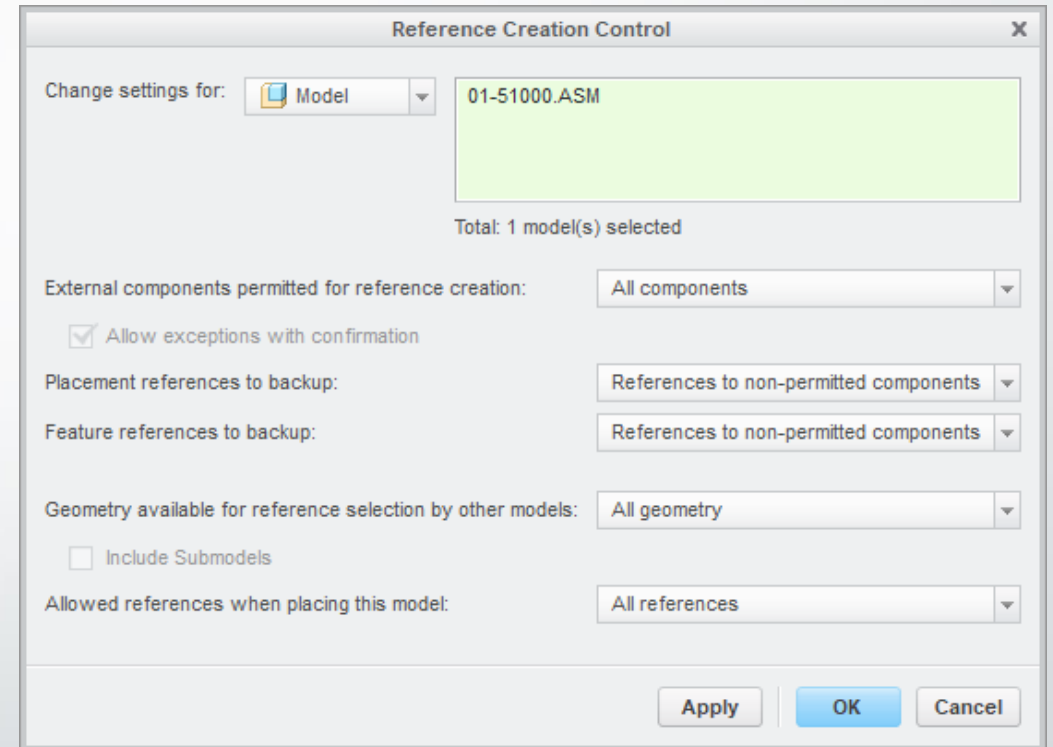
Data Sharing Features



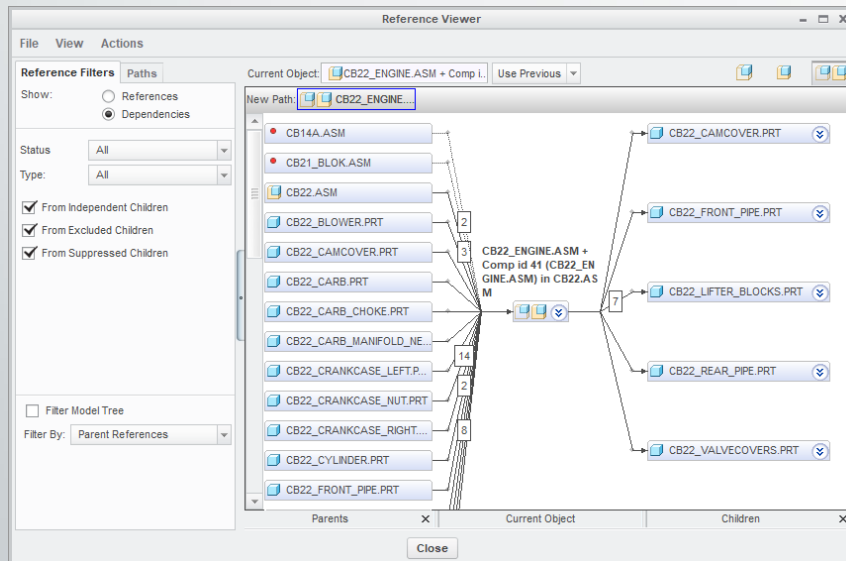
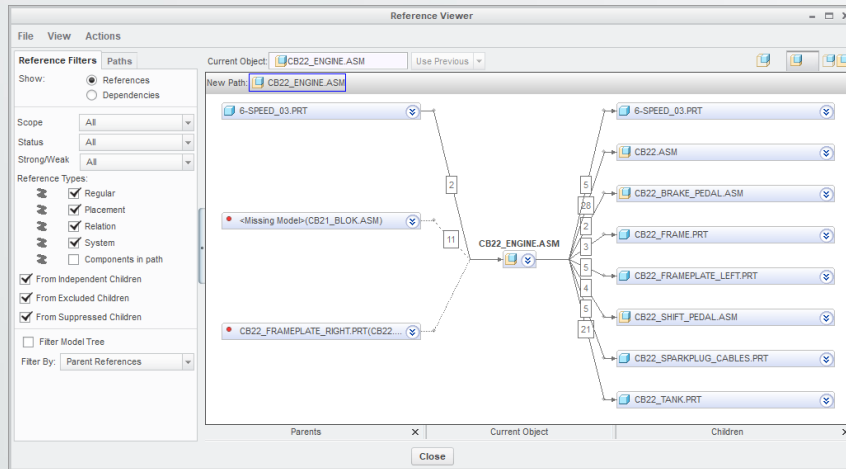
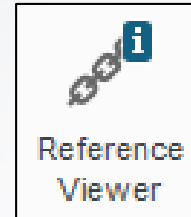
- Data Sharing Features allow you to reference geometry from one component to another.
- Data Sharing Features include:
 - Copy Geometry
 - Publish Geometry
 - Shrinkwrap
 - Merge / Inheritance
- Data Sharing Features are preferable to direct External References because they provide control over dependencies and updating.

Reference Scope Control

- Reference Scope Control allows you to limit the External References that a model can create, and the External References that can be made from another model to it.
- The Reference Creation Control dialog box is available from **File > Prepare > Model Properties** and clicking the blue **change** hyperlink for Reference and Backup.
- Reference Scope Control can also be defined via configuration options (**File > Options > Configuration Editor**) and session settings (**File > Options > Assembly**).



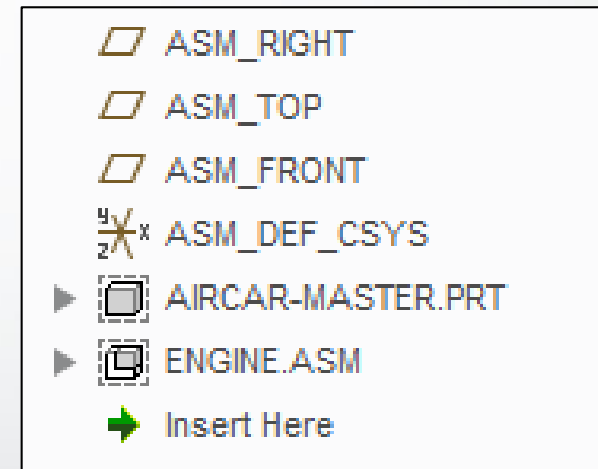
Reference Viewer



- Parent-Child Relationships are a critical method of building Design Intent into models in parametric modeling.
- The Reference Viewer allows you to examine Local References, External References, and Dependencies.
- The Reference Viewer also helps you find and understand Circular References – situations in which one model is both the parent and child of another.
- Dependencies can also be broken using this tool.

Included Components

- The Include command allows you to add a part or subassembly to the assembly, without adding the component's geometry.
- This is useful for laying out the assembly's Product Structure, especially before you are ready to locate components in the assembly.
- Included components appear greyed out with dashed boxes around the glyph (symbol) in the Model Tree.
- Use Edit Definition to define constraints in order to bring the component's geometry into the model.



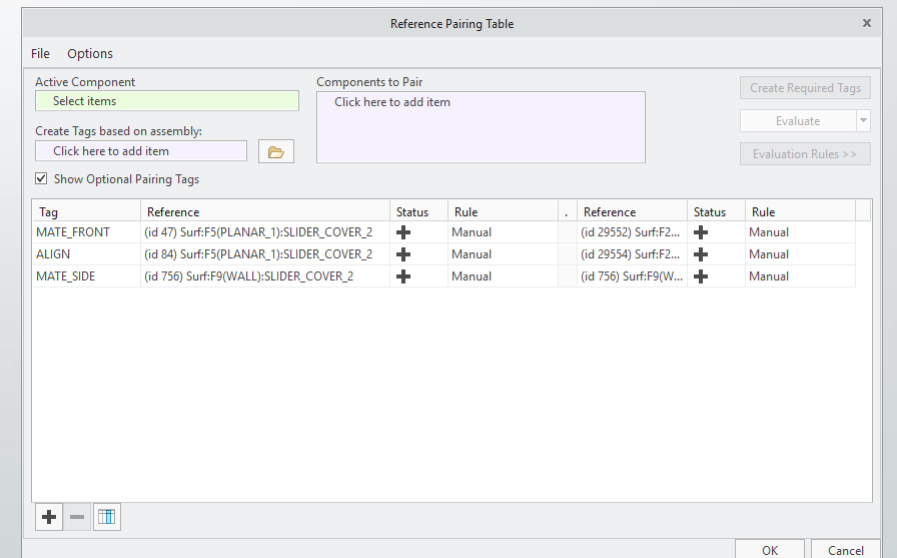
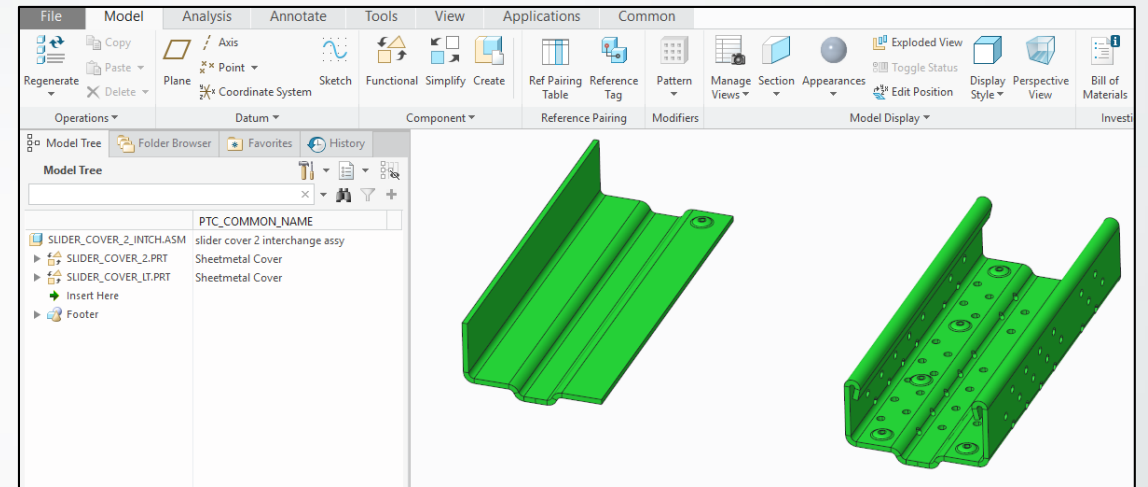


General Assembly Functionality

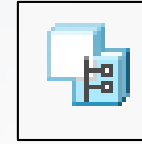
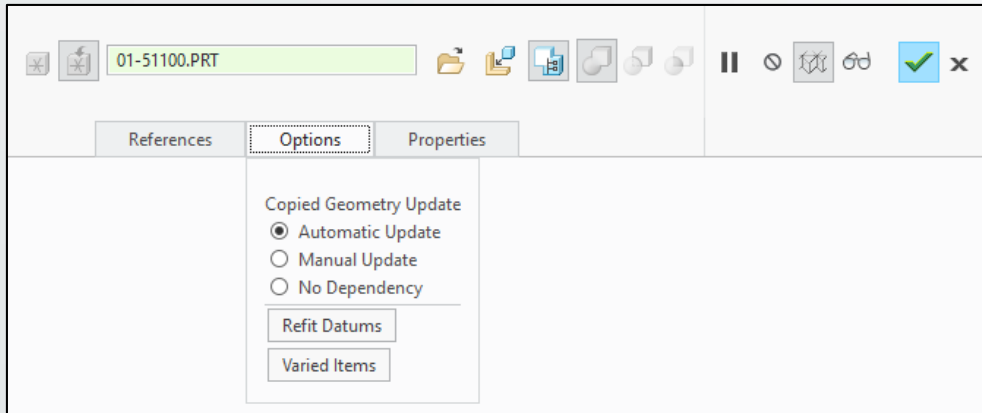
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Interchange Assemblies

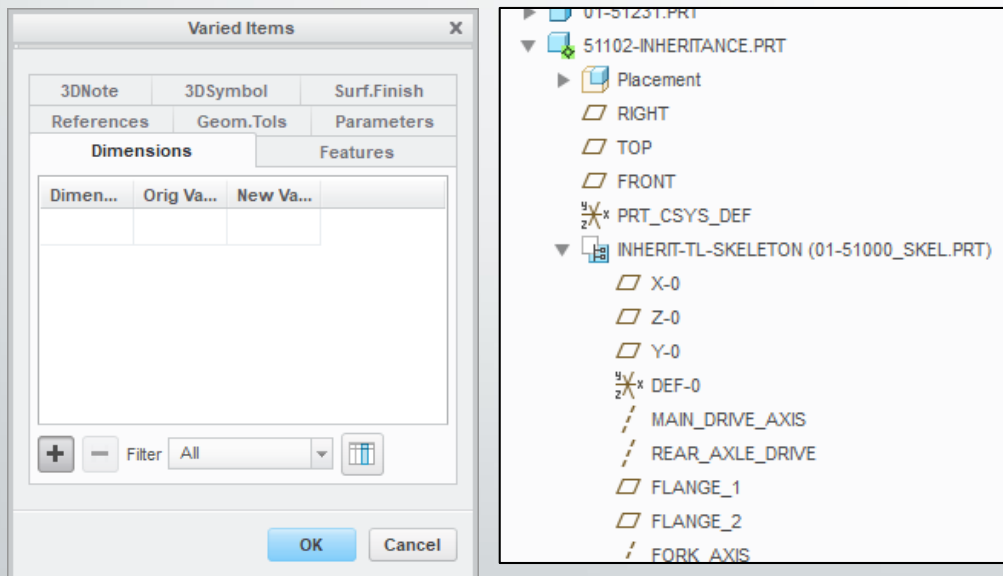
- Interchange Assemblies facilitate replacing components with one another in design assemblies.
- Members of Interchange Assemblies can be both parts and assemblies.
- Reference Tags define what geometry is equivalent to each other for use in assembly constraints.
- Create using **File > New > Assembly > Interchange** (subtype).
- Configurable Modules are very closely related to Interchange Assemblies.



Inheritance Features



- Inheritance Features are a one-way associative merge of all the features from another model, with the ability to vary items from the source model.
- Inheritance Features are great for making one-off variations without using Save As or Family Tables, and documenting different manufacturing steps (e.g., as-cast, as-machined).
- Varied Items include Features, Dimensions, Parameters, and References.



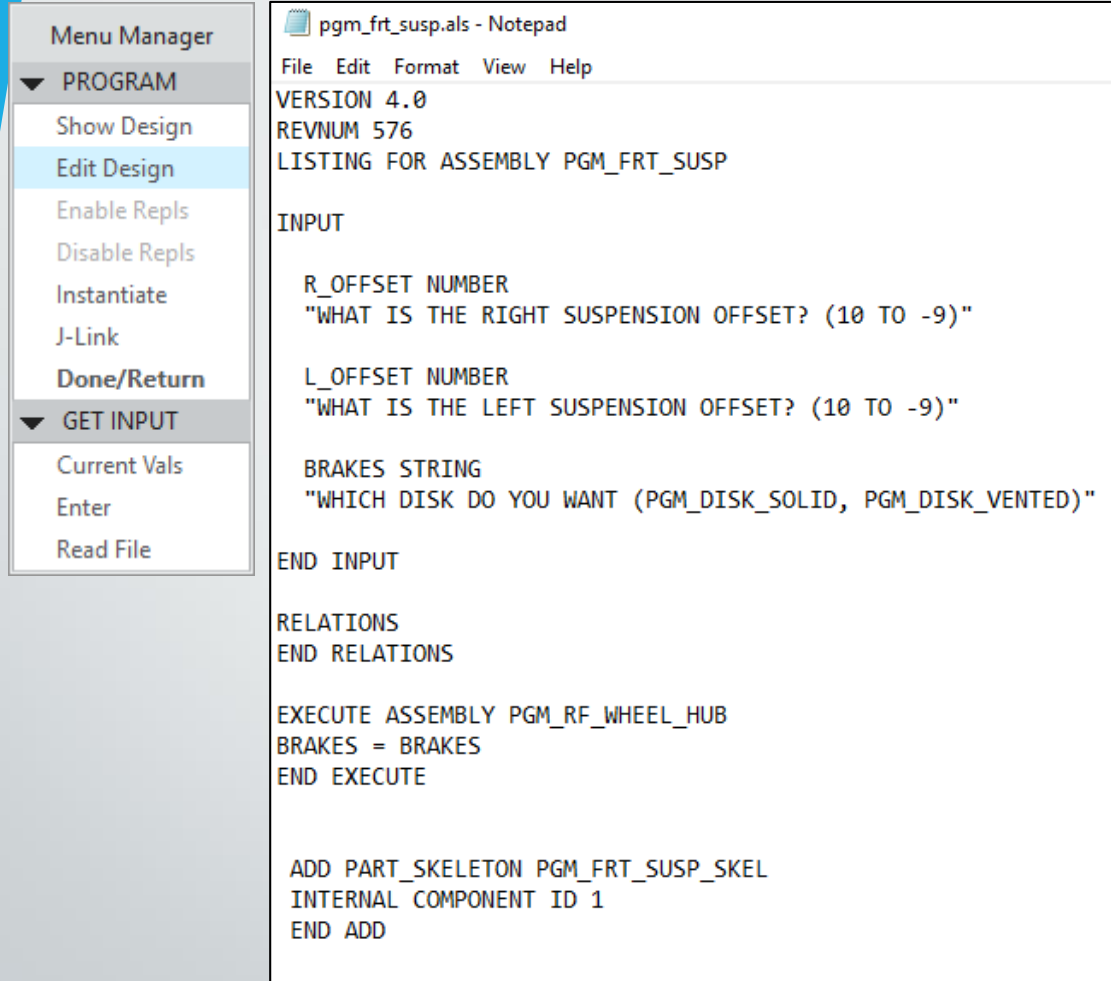
Drawing Representations

- Drawing Representations are the equivalent of a Simplified Rep for a 2D production drawing.
- Drawing Representations allow you to control:
 - Which models, sheets, and views are retrieved and displayed.
 - Updating of tables.
 - Where the drawing opens (sheet, view, or saved view).
- In a drawing, Representations are created and applied from the **Tools > Utilities > Drawing Representation** command.
- Drawing Reps can be accessed upon retrieval in the File > Open dialog box and using the **Open** drop-down list and **Open Representation**.

The screenshot shows the 'Drawing Representation Tool' dialog box with the 'View display' tab selected. The 'Name' field contains 'REP0001'. Under 'Command definition', there are 'Display' and 'Erase' buttons. The 'All' checkbox is checked. There are fields for 'Individual', 'Sheet', and 'Type' (with a dropdown menu showing 'General', 'Projection', and 'Detailed'). The 'Model' field is set to '01-51000.ASM'. There are 'Add', 'Insert', and 'Delete' buttons. A text area contains 'Display all views.' and a checkbox for 'Delay view command(s) until applied'. The 'Navigate sheets' spinner is set to 1. At the bottom are 'Apply', 'OK', and 'Cancel' buttons.

The screenshot shows the 'Drawing Representation Tool' dialog box with the 'Drawing display' tab selected. The 'Name' field contains 'REP0001'. Under 'Drawing location', there are checkboxes for 'Go to sheet:', 'Go to named pan/zoom state:', and 'Go to center of view:'. There are dropdown menus for sheet selection and pan/zoom state. Under 'Table preferences', there are radio buttons for 'As is', 'Frozen', and 'Updating' (which is selected). The 'Navigate sheets' spinner is set to 1. At the bottom are 'Apply', 'OK', and 'Cancel' buttons.

Assembly Pro/PROGRAM



The screenshot displays the Pro/PROGRAM software interface. On the left is a 'Menu Manager' with the following options: PROGRAM (expanded), Show Design, Edit Design (highlighted), Enable Repls, Disable Repls, Instantiate, J-Link, Done/Return, GET INPUT (expanded), Current Vals, Enter, and Read File. The main window is a Notepad titled 'pgm_frt_susp.als - Notepad' containing the following assembly code:

```
File Edit Format View Help
VERSION 4.0
REVNUM 576
LISTING FOR ASSEMBLY PGM_FRT_SUSP

INPUT

  R_OFFSET NUMBER
  "WHAT IS THE RIGHT SUSPENSION OFFSET? (10 TO -9)"

  L_OFFSET NUMBER
  "WHAT IS THE LEFT SUSPENSION OFFSET? (10 TO -9)"

  BRAKES STRING
  "WHICH DISK DO YOU WANT (PGM_DISK_SOLID, PGM_DISK_VENTED)"

END INPUT

RELATIONS
END RELATIONS

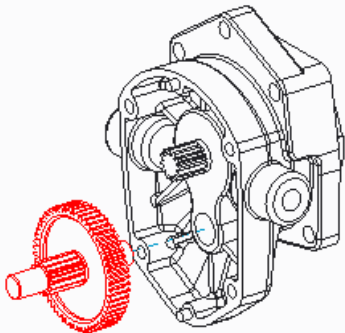
EXECUTE ASSEMBLY PGM_RF_WHEEL_HUB
BRAKES = BRAKES
END EXECUTE

ADD PART_SKELETON PGM_FRT_SUSP_SKEL
INTERNAL COMPONENT ID 1
END ADD
```

- Pro/PROGRAM allows you to create new design variations by turning a model's regeneration cycle into an interactive process where the user can provide new values.
- Prompts are added to the INPUT section of a model's program.
- In the RELATIONS section, dimensions and parameters are driven by the values of those inputs.
- In the Main Body you can:
 - Execute lower-level programs (parts and subassemblies).
 - Add IF THEN statements to the body of the program.
 - Replace components with Family Table instances.

Pro/PROCESS for Assemblies

ASSEMBLY INSTRUCTION SHEET			
MODEL NUM: 09-656821		STA: 3	
DESCR: ASSEMBLY THE REDUCTION GEAR			
		DWG NUM: 09-656821-1	
		REV: 0	
		PAGE: 4 of 10	



SCALE 0.750

FIND	PART NUMBER	DESCRIPTION	QTY
1	REDUCTION.GEAR.SHAFT	REDUCTION GEAR SHAFT	1

EMEP	MFG
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- Process Assemblies document the step-by-step instructions for putting a product together.
- Process Assemblies include both the End Item as well as Fabrication Units (e.g., tooling and fixtures).
- Process Steps can be Assemble, Disassemble, Reassemble, Reposition, and General.
- To define a Process Step, you also specify the components, Simplified Rep, and Explode State.
- Process Assemblies are then documented in 2D production drawings.
- Somewhat similar to MPMLink for Manufacturing Process Planning on the Windchill / PLM side.

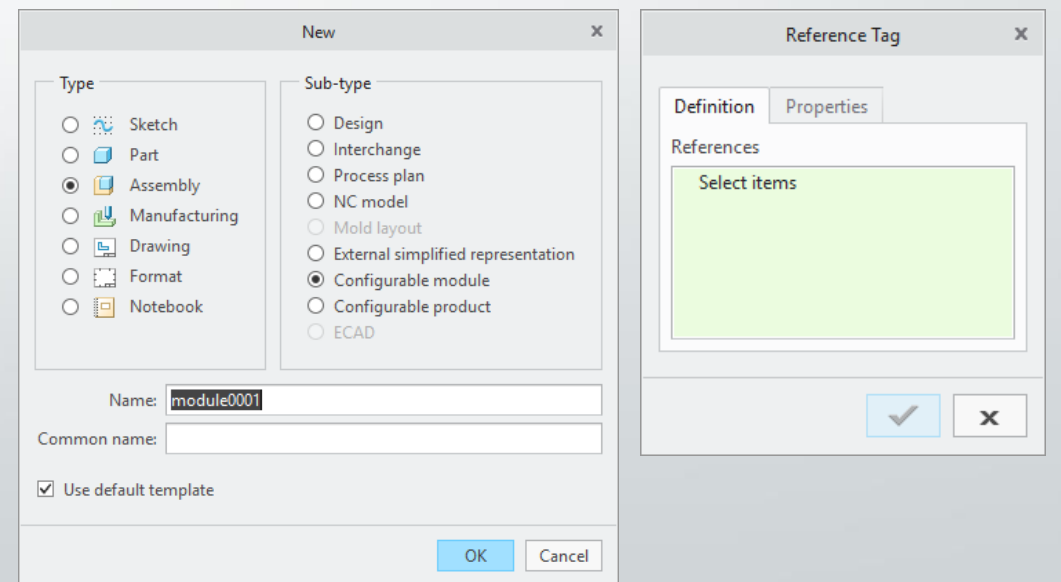
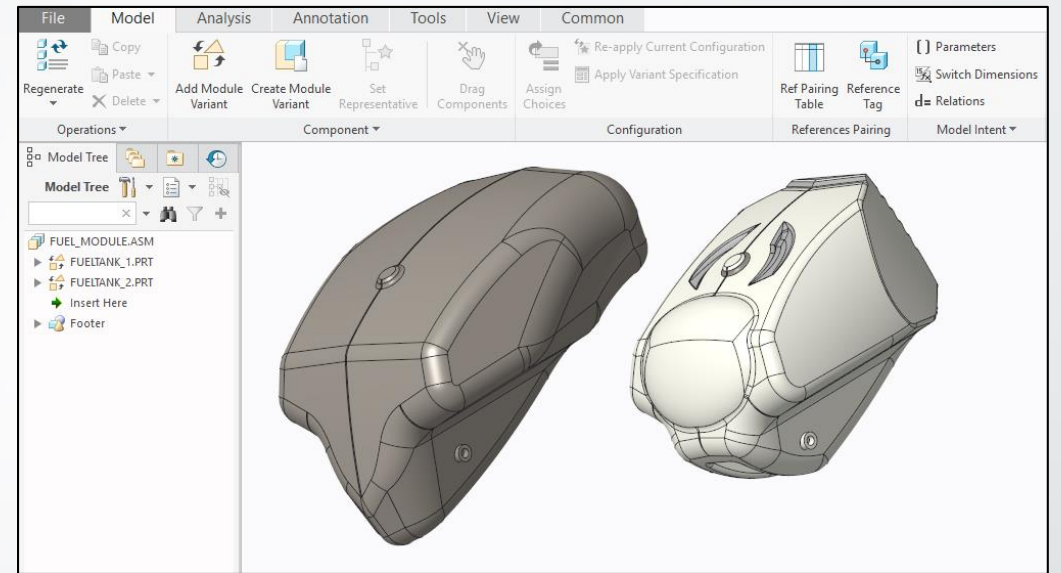


Options and Variants

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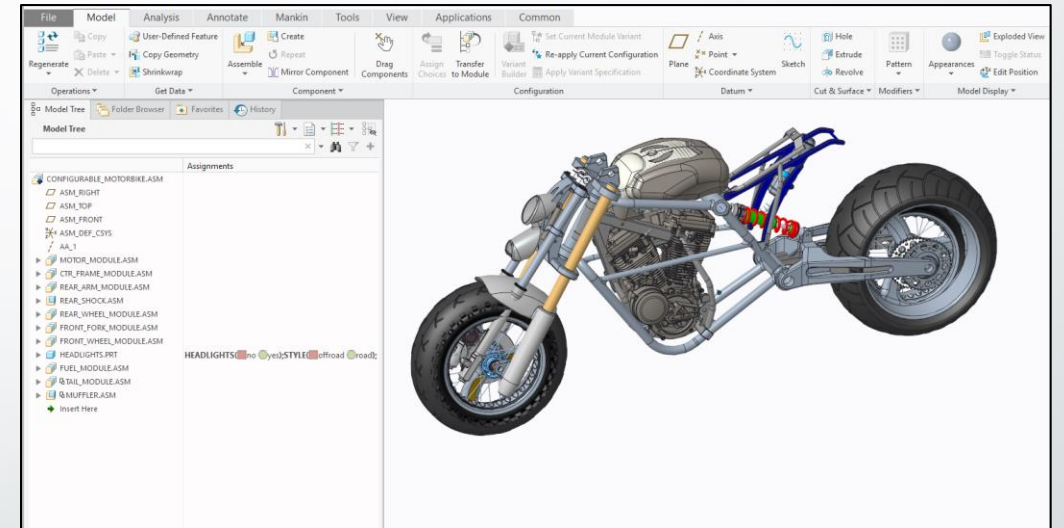
Configurable Modules

- Configurable Modules are assemblies that contain Variants – parts and assemblies – that represent different choices in a Configurable Product.
- Created via **File > New > Assembly > Configurable Module** (subtype).
- Reference Tags define what geometry is equivalent to each other for assembly constraints.
- Configurable Modules are very similar to Interchange Assemblies.



Configurable Products

- Used in the Configure To Order (CTO) manufacturing strategy.
- Configurable Products are special assemblies that contain design subassemblies, design parts, and Configurable Modules.
- Configurable Products are also known as “Overloaded BOMs” since they contain all possible components reflecting all possible customer choices.
- Created via **File > New > Assembly > Configurable Product** (subtype).
- In order to configure Variants – new assemblies generated from user choices – the Options and Variants module is needed.



Thank You!

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