

TOP DOWN DESIGN IN CREO PARAMETRIC



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Chapter 4: Skeletons

An Excerpt

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4. Skeletons

Key Points

- Skeletons are special part models created in assemblies that contain the geometry we want to share between components.
- Skeletons build Design Intent into assemblies. Instead of modifying multiple individual components for a design change, we simply update the Skeleton. When we regenerate the assembly, changes to the Skeleton propagate to all the models that reference it.
- Skeletons contain the same kinds of features as standard parts. A Skeleton is as simple or as complex as you need.
- A subassembly Skeleton can reference either all or part of a higher-level Skeleton. Subassembly Skeletons will contain geometry relevant to controlling components in that assembly.
- A Skeleton for cables or pipes may have a **Shrinkwrap** feature to access all the necessary surfaces for routing.
- An assembly can contain multiple Skeletons.
- Skeletons are created in an assembly with the Component **Create** command.
- Mechanisms – assemblies with moving components – can be captured in Skeletons with either multiple Skeletons assembled with mechanism connections, or a Motion Skeleton.
- A standard part model can be turned into a Skeleton using a couple of techniques. See section 4.8 for more information.

4.1. Skeletons Overview

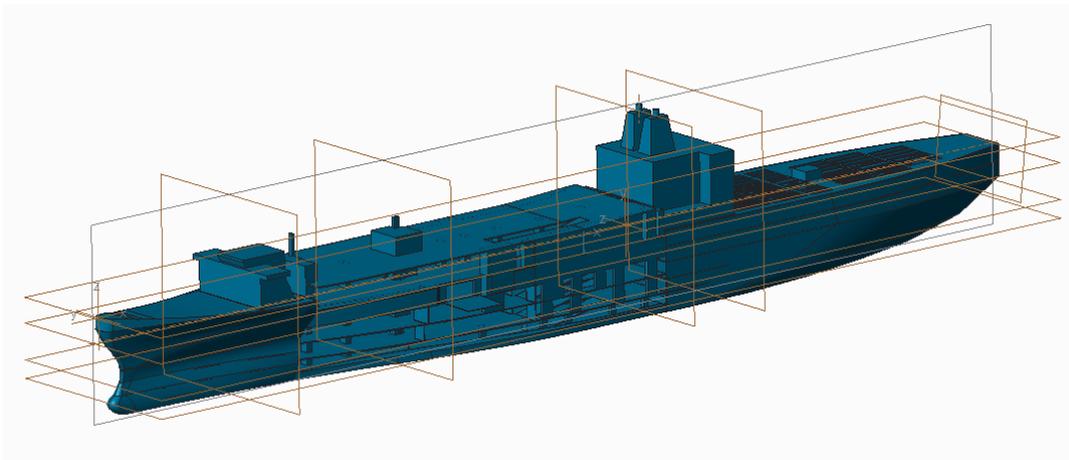
The topic of Skeletons can cause quite a bit of confusion, especially amongst users who are relatively new to Creo Parametric. As mentioned many times before, the Skeleton is a place where we can capture geometry that affects multiple components in our products. They can be quite complex; they can also be relatively simple.

In this chapter, we will discuss the following:

- Why and how we use Skeletons, and their similarities and differences compared to part models.
- Various strategies for employing them in your products.
- How to create Skeletons.
- Design considerations when choosing what features to put into your Skeletons for maximum effectiveness.
- Using Skeletons with Sheetmetal assemblies.
- Two different approaches for controlling mechanisms with Skeletons.
- How to turn a standard part model into a Skeleton.

4.2.Skeletons Demystified

Skeletons are special kinds of part models that are used as the framework of an assembly. Skeletons contain geometry to be referenced by components and child subassemblies. They can also reference geometry from higher-level (parent) and parallel-level assemblies.



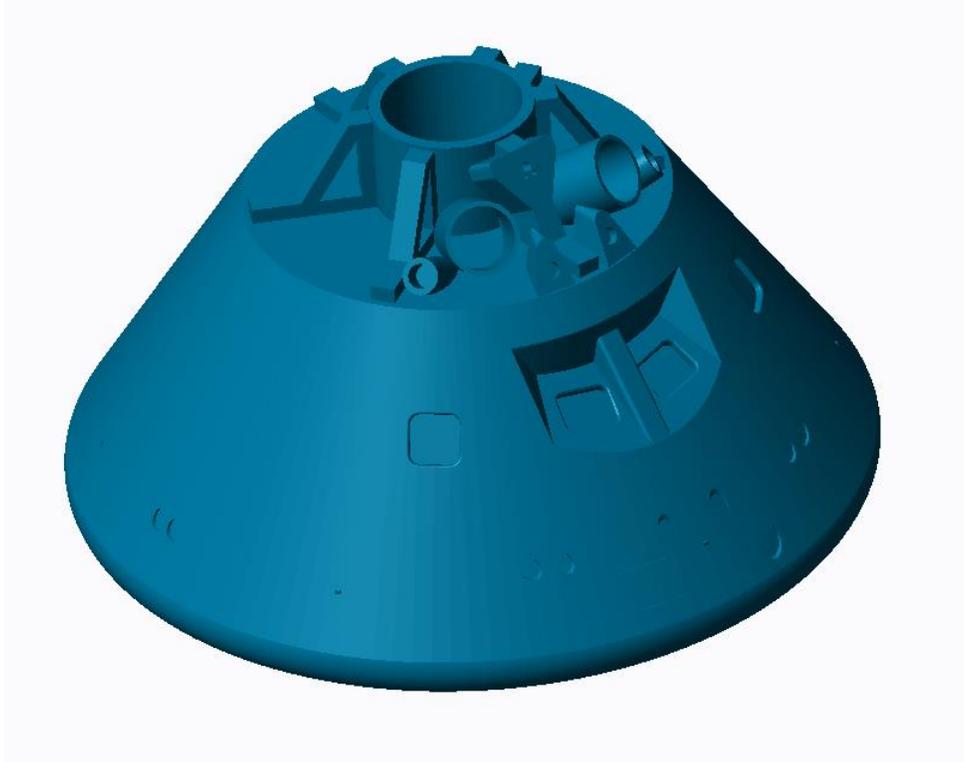
Top-Level Skeleton (detail removed for clarity).

Skeletons serve the following purposes:

- Skeletons allow us to capture Design Intent – what we care about most in our product – in a central location. This can be helpful at the beginning of product development and throughout the entire lifecycle.
- Skeletons consolidate geometry common to multiple components in order to simplify the modification process.

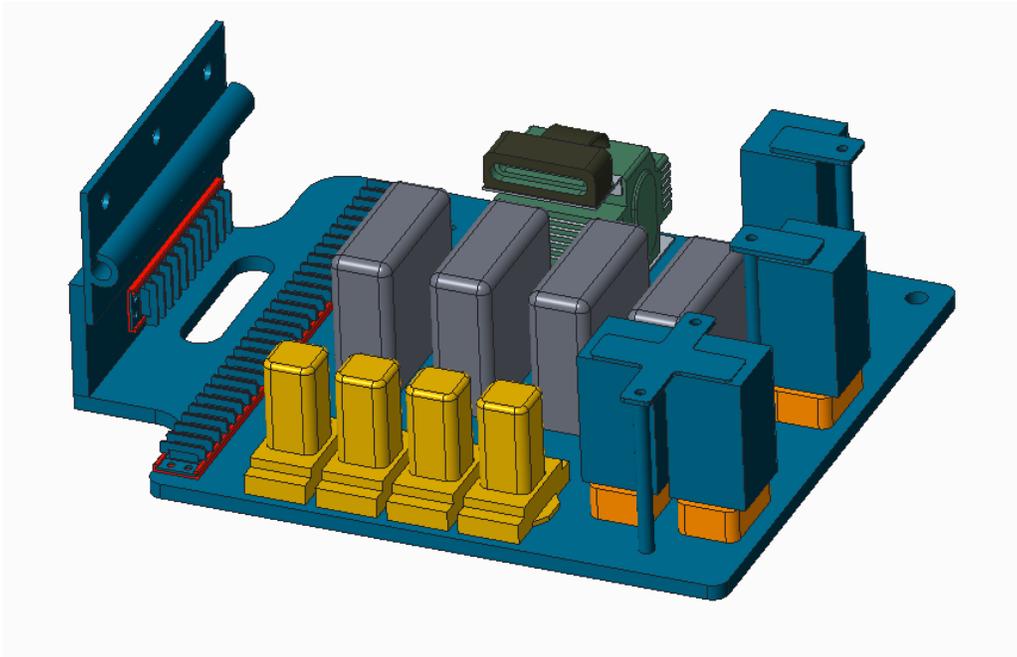
Skeletons can be used for the following purposes:

- Skeletons capture the overall shape of a product. This can be referred to as the master model, industrial design, OML (outer mold line), or various other terms specific to your industry or company.



Imported OML Skeleton.

- Skeletons capture important interfaces between subsystems and components. This can include details such as:
 - Mating surfaces.
 - Hole / shaft locations.
 - Lips, tabs, snaps, and other means of securing components to one another.
- They can capture reference geometry from higher-level or parallel- subassemblies for purposes including designing components or routing wires and cables. (In the past, this was accomplished with standard part models known as “map parts.”)



Skeleton of shrinkwrapped components for cable routing.

- Mechanism modeling to define how components move with respect to one another.



Mechanism captured via multiple skeletons assembled with connections.

- Space claims to facilitate concurrent engineering.

Skeletons differ from standard design parts in the following ways:

- Skeletons can only be created in the context of an assembly. With an assembly open, the **Component Create**  **Create** command. There is no File > New > Part > Skeleton

command. However, if you have an existing part model that you want to be a Skeleton, I'll show you how to do that in section 4.8.

- Skeletons are automatically filtered from Bills of Material (BOMs).
- Solid geometry in Skeletons is not included in mass properties calculations for the assembly.
- Skeletons have different icons in the Model Tree  .

Skeletons are similar to parts in the following ways:

- They have the same .prt file extension.
- They contain the same features, including but not limited to:
 - Sketches and sketch-based features (e.g., Extrudes, Revolves, Sweeps, and Blends).
 - Surfaces and Quilts, which can be created either parametrically, freeform (using the Style feature in ISDX), or by Subdivisional modeling (via the Freestyle feature).
 - Datums, including planes, axes, points, and coordinate systems.
 - Non-sketch based features, such as Holes, Rounds, Chamfers, and Draft. However, finishing features that are particular to individual components should not be included in Skeletons.
 - Imported geometry (e.g., STEP, IGES, Neutral, DXF).
 - Data Sharing Features (e.g., Copy Geometry, Shrinkwraps, Merges, and Publish Geometry).
- They share the same ribbon interface.
- They can both be opened in their own window and edited.

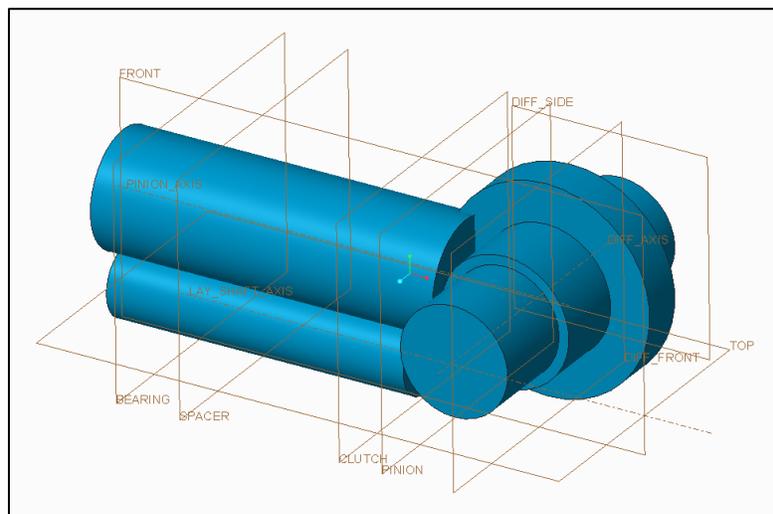
4.3.Skeleton Strategies

Skeletons can vary from a few (less than a dozen) features, to literally thousands of features that control all part geometry. As with other aspects of Top Down Design, you want to find the balance between Control, Creativity, and Speed. Here we will discuss some of the different strategies to consider in your approach to Skeleton modeling depending on various scenarios. Some of the things to consider include:

- The Level of Detail in the Top-Level Skeleton.

- The amount of referenced geometry and new geometry in subassembly Skeletons.
- Techniques for references geometry for Routed Systems like cabling and piping.
- The use of more than one Skeleton in an assembly.
- Using the same Skeleton at multiple levels of hierarchy in the Skeleton.

One common misconception regarding Skeletons is that they have to be complicated. Often, they end up being quite simple. They only need as much geometry and datums as necessary to control multiple components.



Example of a simple skeleton.

4.3.1. Top-Level Skeleton

Generally, the Skeleton at the top level of the assembly contains the overall shape, as well as interfaces between the subassemblies at the top level. Depending on your company and industry, you might hear the top-level Skeletons referred to as:

- Master Model: Some companies do not use Skeleton functionality. Instead, they might use a standard design part model to perform the same function as a Skeleton. They may keep the Master Model external to the assembly, perhaps to avoid BOM issues.
- Outer Mold Line or OML: This is very common especially in the aerospace industry to refer to the exterior envelope of the vehicle.
- Industrial Design or ID: This term is very common with consumer products or anything that has a shape that tends to be more aesthetically pleasing, ergonomic, or organic in shape. These shapes typically involve surface modeling techniques, such as parametric surface modeling, freeform surface modeling (using the Style tool in ISDX), subdivisional

modeling (using the Freestyle tool), imported geometry (such as Rhino data), point cloud data from scanning, or a combination of these methods.

4.3.2. Lower-Level Assembly Skeletons

Each lower-level assembly typically needs only a subset of the geometry from the Top-Level Assembly or any other assemblies above it. To reduce model retrieval and regeneration time, as well as end user confusion, a lower-level assembly would typically use:

- Copy Geometry features to capture specific surfaces or datums from the higher-level assembly; or,
- A Merge feature to grab the entire Skeleton of the other assembly.

Lower-level assemblies typically also contain original geometry, relevant to controlling the interdependent components that reside within them, the interfaces between its subassemblies, and assembly constraints of its components.

4.3.3. Routed Systems Skeletons

Skeletons are often used to facilitate the design of cable harnesses and pipes, which are also known as *Routed Systems*. Typically, the starts and ends of the routed systems are placed first. In the case of cable harnesses, these are typically electrical, electronic, and power generating components. For pipes, these can be equipment, inlets, and outlets. Then the harnesses or pipes are routed between them.

The cable harnesses and piping assemblies may be their own assemblies. How the routed systems are organized in the context of the product structure depends on a number of factors, including:

- How the BOM is organized. We may intend to separate electrical or fluid components from structural components.
- How the routed systems are designed, sourced, or manufactured.
 - If the cables or pipes are designed concurrently with the design of other components, our harnesses and piping will be integrated into the same assemblies as other components.

- If the users performing the routing are different or separated from those designing structural or other functional systems, for Concurrent Engineering it may be practical to have separate assemblies.
- If outside vendors or separate departments fabricate the harnesses or pipes, then they may be kept separate from the primary subassemblies.
- How and when the routed systems are installed into the product.
 - If cabling and piping are physically assembled into the product as the other components are placed, then they might be integrated into the same assembly structures.
 - If the routed systems are installed (“plugged in”) towards the end of fabrication and assembly, then it might be more efficient to keep them as separate assembly structures.

If the routed systems are kept separate from the rest of the assembly structure, then Skeletons can be particularly advantageous towards capturing the necessary references. Typically, you would want to grab all the relevant surfaces from higher-level assemblies or parallel-level assemblies. In these situations, the **Shrinkwrap** feature can be the most convenient method of capturing all those necessary references quickly and easily.

4.3.4. Multiple Skeletons

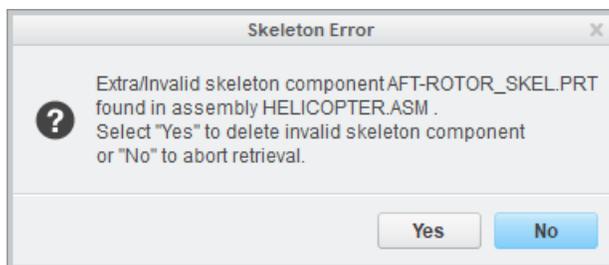
To organize your reference geometry and simplify the organization of your Skeleton, you may want to employ multiple Skeletons in an assembly. You could divide up the geometry by function (e.g., structural vs. electrical), location/region, or some other factor.

In order to utilize multiple Skeletons in an assembly, the configuration option *multiple_skeletons_allowed* must be set to yes. The default value is no.

The first Skeleton in an assembly is always placed as the first object in the assembly, even ahead of the default datum planes.

When you create additional Skeletons, the Component Placement dialog box opens. This allows you to specify the constraints to locate the new Skeleton. You could use the Default constraint to locate it at the origin of the assembly, and it could remain there throughout the design process. Since the Skeleton will likely contain no geometry when it is first created, later after adding geometry, you could edit the definition of its placement to define the correct constraints.

Note: if another user attempts to open an assembly with multiple Skeletons at any level, and he or she does not have *multiple_skeletons_allowed* set to yes, the following warning will open in the Graphics Area:



Aborting retrieval and changing the configuration option is often the best choice, as removing the additional Skeletons will likely result in a regeneration error.

4.3.5. Same Skeleton Placed at Multiple Levels

If a Skeleton is assembled into an assembly that doesn't contain a Skeleton, it is automatically placed as the first component, even before the Default Datum Planes.

Because of this fact, designers can place a Skeleton, such as the top-level Skeleton, and place it at many (or all) levels within the assembly. This technique is very convenient for assembling components at all levels relative to the same geometry, such as datum planes, coordinate systems, or surfaces.

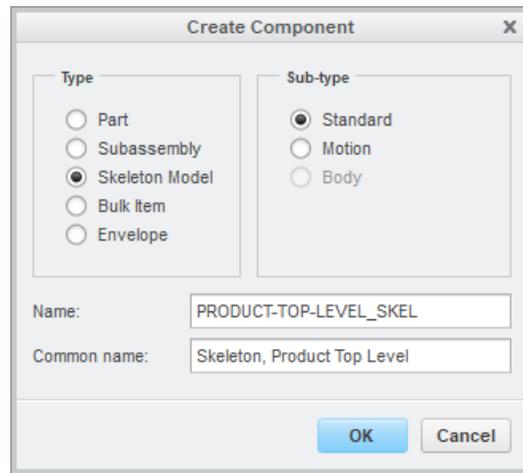
The same Skeleton could be used across multiple individual products, for example, if they are a family of similar products or a product line. However, you want to make sure that the skeleton does not have any dependencies.

4.4. Skeleton Creation Process

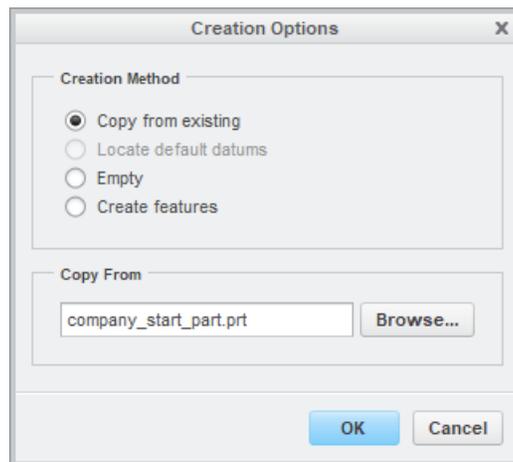
The process described in this section is probably the most nebulous one in this book. Asking how to build a skeleton is akin to asking how to write a book, compose a song, or paint a picture. How many features should be in a skeleton? As many as are needed. How complex does it need to be? It needs to convey as much information as necessary. You will see that the creation process below is outlined in broad strokes.

The Skeleton creation process is as follows:

1. In the assembly, from the Component group choose **Create**  .
2. In the Create Component dialog box, select the radio button for **Skeleton**.



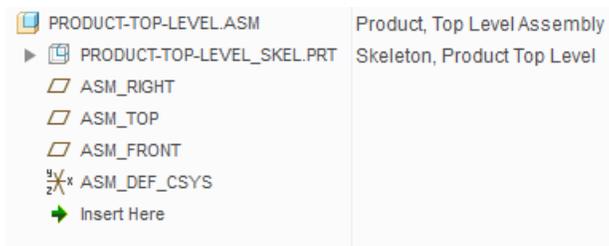
3. Enter the Name and Common Name for the Skeleton using your company's naming convention.
4. In the Creation Options dialog box, the radio button for **Copy from existing** will be selected by default. Your company's standard default start part should be listed in the Copy From field. If desired, click the **Browse** button to navigate to and select a different start part. Click **OK**.



5. If this is the first Skeleton in the assembly, it will automatically be located as the first object, even ahead of the Default Datum Planes.
6. If the configuration option *multiple_skeletons_allowed* is set to yes and one or more Skeletons already exist in the assembly, the Component Placement dashboard will open. Define constraints (such as Default) or mechanism connections to locate the

Skeleton where you want it. Click the middle mouse button or green checkmark on the dashboard when finished.

- The Skeleton will appear in the Model Tree.



- Either open the Skeleton in its own separate window or right mouse click on it and select **Activate**.
- If the Skeleton is not the top-level Skeleton, and it needs to reference geometry from a higher- or parallel- level assembly, create Data Sharing features that leverage the necessary geometry. **Error! Reference source not found.** are covered in detail in chapter **Error! Reference source not found.**
- Create native features in the Skeleton as necessary. See section 4.4.2 for the kinds of features that typically appear in Skeletons.

4.4.1. Naming Convention

By default, Creo Parametric will suggest a name the Skeleton model that is the name of the assembly in which it is created with `_skel` appended to it.

If the configuration option *multiple_skeletons_allowed* (discussed in section 4.3.4) is set to yes, then the suggested name will be the name of the assembly with `_skelxxxx` appended, where `xxxx` is a four digit number, starting with 0001 and incremented with each subsequent Skeleton created in the assembly.

You can name your Skeleton however you want, of course. But it is highly recommended that your company have a naming convention that makes it obvious to users that a part is a Skeleton and not a design model.

If you are using Windchill, your administrators may have set Object Initialization Rules (OIRs) with your company's part numbering scheme.

4.4.2. Skeleton Features

In this section, we will briefly discuss some of the kinds of features often found in Skeletons.

Datums

Datum features are imaginary references we create in our models in order to define and control surface and solid features. The most common types of Datum features are:

- Datum Planes.
- Datum Axes.
- Datum Coordinate Systems.
- Datum Points.
- Datum Curves.

Sketches

Sketches are two-dimensional curves. They can be used to represent a variety of different purposes, such as:

- Outlines and space claims.
- Cam profiles.
- Lines of action for belts and pulleys.
- Industrial Design (ID) inputs.

Sketches can be inputs for features in the following ways:

- Extrudes and Revolves.
- Fill (flat) surfaces.
- Trajectories and boundaries for Sweeps, Blends, and Swept Blends.
- Creating three-dimensional curves via Intersect, Project, and Wrap.
- Creating embossed features using the Offset command.

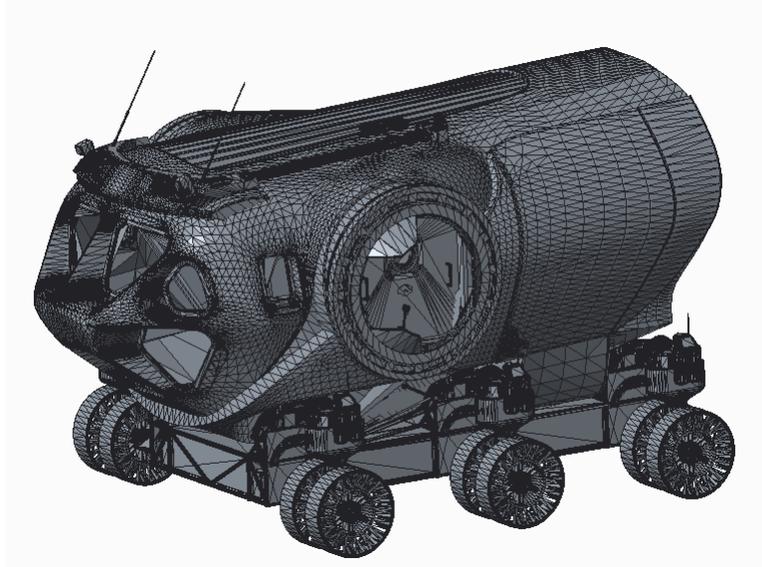
Surfaces

Surface features are three-dimensional features with a shape that you define. They are infinitely thin / have no thickness, and therefore have no mass. (They can enclose a volume depending on their definition.) Although Surfaces can be created with commands like Extrude and Revolve, they are typically created with different features in order to create geometry that is more organic, aesthetic, ergonomic, or complex.

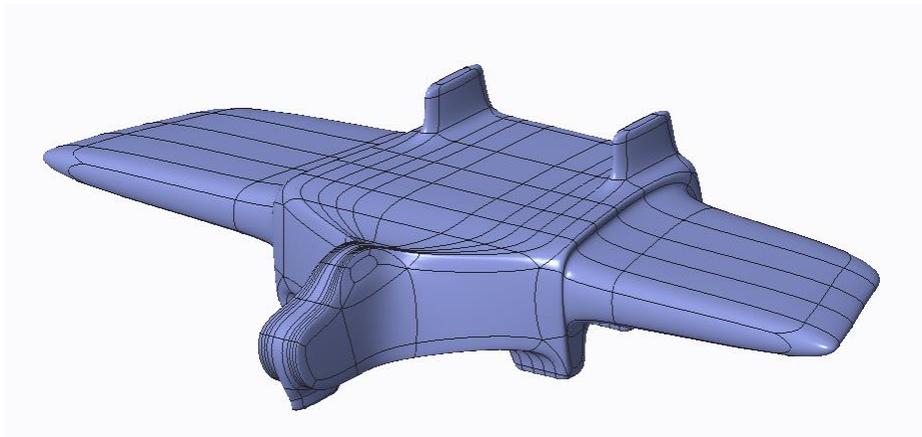
The term Quilt is very common in surface modeling. While technically it refers to the result of multiple surfaces that are stitched together, Quilt can also refer to a single surface, or even a datum plane.

Surfaces can be created in a variety of ways, including:

- Sketch-based features, such as Extrudes, Revolves, Sweeps, and Blends.
- Boundary Blends, Swept Blends, and Rotational Blends.
- Style features using ISDX (Interactive Surface Design Extension).
- Imported geometry from other CAD systems or modeling applications.



- Subdivisional modeling using the Freestyle feature.



- Reverse engineered point cloud data, which can be featurized using the Restyle feature from REX (Reverse Engineering Extension).

Surface features can be edited using a number of commands, including:

- Merge: to stitch two or more quilts together using either Intersect or Join.
- Trim: to divide a surface using a curve, plane, or other quilt.
- Extend: to lengthen or shorten a surface by translating the edges.
- Offset: to create additional surfaces a specified distance from the input surface.
- Warp: change the shape via stretch, bend, twist, sculpt or other operations.

4.5. Considerations

Here are just a few of the many various scenarios you need to consider when planning and developing your Skeleton:

Enclosures or Components First? Imagine that you are designing a product with several interior electronic components, such as a computer, tablet, or phone. The product contains various components such as circuit boards, memory modules, display components, audio components, cooling fans, wireless antennas and transmitters, external input interfaces, and so on. All these components are contained within an enclosure. Your approaches include:

- Place the electronic components first, and then design the enclosure around it. (This is known as “Middle Out Design” in Autodesk Inventor.) If you are using a lot of Commercial Off The Shelf (COTS) components of known sizes, this approach would be prudent. Since established COTS components are unlikely to change size, there would be no need to use External References to drive the dimensions of the enclosure.
- Design the enclosure and then place the electronics inside it. If the Industrial Design is locked down first, this would make sense.

The approach you choose depends on your situation.

Holes or Axes? The Skeleton contains geometry that affects multiple components. Those often include holes or hole/shaft combinations. In the Skeleton, you would choose between the following approaches:

- Create hole features in the Skeleton: If you need to ensure all the holes in all the components have the same diameter and would change together, then this approach is recommended.

- Create axes in the Skeleton: This approach might make more sense if any of the following are true:
 - You are controlling both holes and shafts.
 - The holes in the different components have different diameters due to dimensional tolerances.
 - The component designer needs to choose options such as counterbore, countersink, drill depth, or taper.

The components that need those references can then use a Copy Geometry or Merge feature as necessary.

Dimensioning Scheme for Mounting Locations? Everything in Creo Parametric relates to Design Intent: how we expect our model to change and how we want it to update to react to those changes. Take the case of a mounting pattern. We will use an example of a rectangular plate with four holes located at each corner:



Consider the following when choosing your dimensioning scheme:

- Dimension the holes from the sides of the part. That way, if the part changes size, then the hole locations update.

- If you want to control the distance between the holes, then you may need to use Relations to control the dimension for the distance between the holes as a function of the plate dimensions.

We want to build Design Intent into the model to ensure that the hole locations update properly so that they don't end up too far to the interior of the plate, or in empty space beyond the part borders.

Recreate Component Geometry in the Skeleton. Sometimes you create geometry in a part. Later, you realize that other components could use that geometry, and it should have been captured in and controlled by the Skeleton.

- Recreate the geometry in the Skeleton.
- Create Data Sharing features from other components to the part in which the geometry was originally created.

Is it worth taking the time to go with the first option, or will the second option suffice? Only you can answer that question.

Patterns in Skeletons. You can capture all kinds of geometry in the Skeleton using patterns. However, you need to consider how components will reference that pattern later on. Data Sharing features are covered in the next chapter. When referencing the pattern, you have the following choices:

- Create a Copy Geometry feature that references the pattern members explicitly.
- Use a special technique to reference pattern the Skeleton's pattern in the target part.

Which method you choose depends on your Design Intent: the changes you anticipate and want to plan for.

These are just a few examples of the kinds of problems and decisions you will have to consider when designing both your Skeleton and your individual components.

4.6. Sheetmetal Assemblies

Sheetmetal mode contains feature creation commands specific to relatively thin, constant thickness components created from ductile materials that undergo bending, forming, punching, and other related manufacturing operations. However, there is no Skeleton subtype specific to creating Sheetmetal geometry.

A Skeleton for a Sheetmetal assembly will typically contain the following:

- Datum planes.
- Surfaces to represent enclosures or panels.
- Axes for holes, mounting locations, and the like.

Just like with Standard components, geometry will be shared from the Skeleton to the target components with Data Sharing features. Then Sheetmetal commands like Planar, Extrude, Flat Wall, and Flange Wall can be used to generate geometry with thickness.

If the Skeleton contains surfaces representing enclosures or panels, then the Offset command in Sheetmetal mode (not to be confused with the Offset command for creating surfaces in Standard mode) can be used to generate thickness. Use of this technique will be covered in more detail in Chapter 7 on Populating the Assembly.

4.7. Mechanisms

Top Down Design can be used to define products with both static and kinematic components. However, a Skeleton is a special kind of part model, and parts consist of features with explicitly defined locations and sizes. In other to leverage Top Down Design with mechanisms, we can employ two techniques:

- Multiple Skeleton models assembled with mechanism connections.
- Motion Skeletons.

4.7.1. Multiple Skeletons

With the configuration option *multiple_skeletons_allowed* set to *yes*, we can place more than one skeleton in an assembly. As mentioned earlier, the first Skeleton is automatically placed as the first object in the assembly and does not require any constraints. Adding more Skeletons to the assembly will open the Component Placement dialog box.

In order to convey motion, we would assemble additional Skeletons using mechanism connections – such as Pin, Slider, Cylinder, Planar, Ball, Bearing, and so forth – as opposed to typical assembly constraints (e.g., Coincident, Distance, Default, etc.). You could also leave the additional component underconstrained so it has translational and/or rotational Degrees of Freedom. However, mechanism connections are preferred, because then you could:

- Apply limits to the range of motion.
- Apply Motors to the connection to define various motion profiles.
- Calculate various measures at the connections in a Kinematic or Dynamic analysis.

Note: Kinematic analyses can be performed with any Creo license since all packages come with MDX, the Mechanism Design Extension. However, Dynamic analyses require MDO, the Mechanism Dynamics Option, which is not available in the base Creo license.

When creating Data Sharing features from additional Skeletons to target parts, be sure to Externalize the Data Sharing feature. Externalizing the Data Sharing feature uses a Coordinate System to locate the copied geometry in the target part. Without Externalizing, the copied geometry's location in the target part is determined by wherever the source component is located. Since the additional Skeletons are capable of moving, the location of the Data Sharing feature in the target part can change upon regeneration. Any features that reference those Data Sharing features could change location and possibly result in unwanted geometry. Even worse, drawing views could update incorrectly.

4.7.2. Motion Skeletons

To solve the problem of combining Top Down Design with Mechanisms, PTC added a new kind of Skeleton called a Motion Skeleton in Pro/ENGINEER Wildfire 3.0. Motion Skeletons differ from standard part Skeletons in that they are assemblies, consisting of a Sketch feature and Body Parts.

Honestly, Motion Skeletons can be quite challenging and difficult to get to work as intended. Also, since the motion is controlled by a two-dimensional Sketch feature, you are limited to motion within that plane.

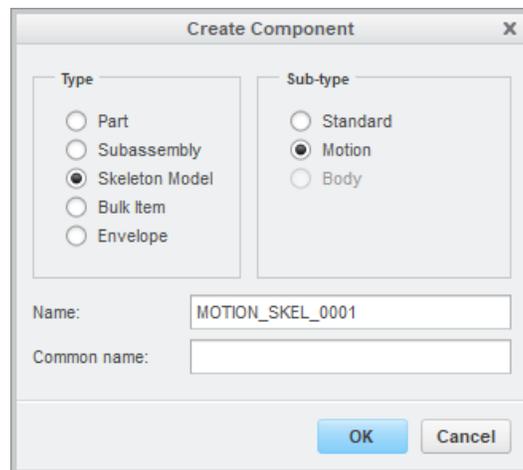
In the event that you want to explore their use, we will present the general methodology here. Note that Creo Parametric does not guide you through the process, and the Help information is somewhat scant.

As mentioned previously, a Motion Skeleton is actually an assembly. It will contain the following entities:

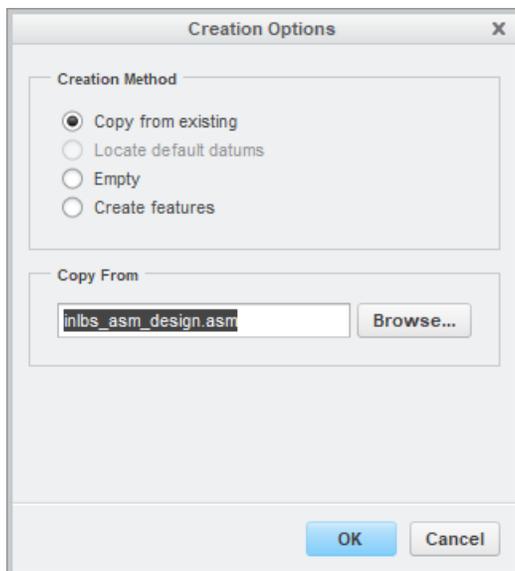
- A Sketch at the assembly level to define the capable motion.
- Body Skeletons, which are part models in the Motion Skeleton assembly.
- Mechanism connections between the first Body Skeleton (the ground) and the other Body Skeletons (the moving components).

To create a Motion Skeleton and the corresponding solid parts, perform this general process:

1. In the assembly, click the **Create**  **Create** command.
2. Select the radio buttons for **Skeleton** and **Motion**.



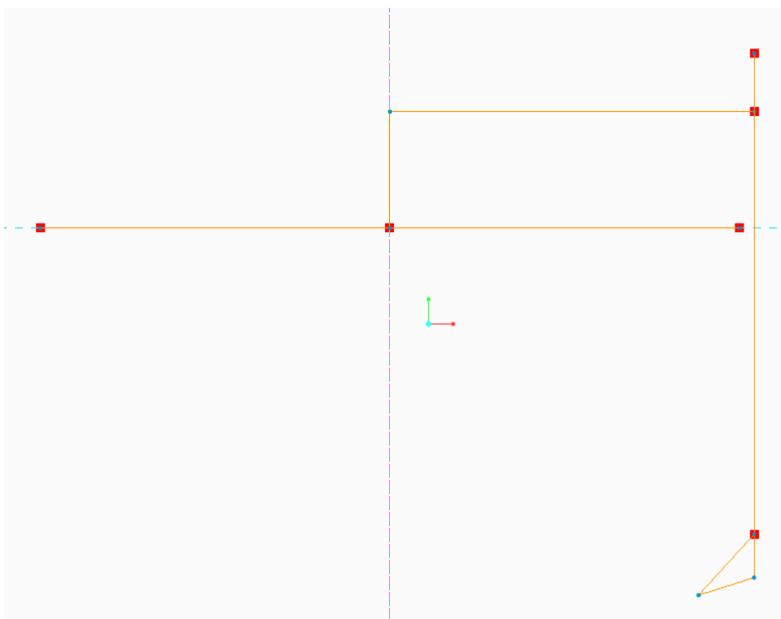
3. Enter the Name and Common Name. Click **OK**.
4. In the Creation Options dialog box, use the **Browse** button to select a different start assembly if desired. Click **OK**.



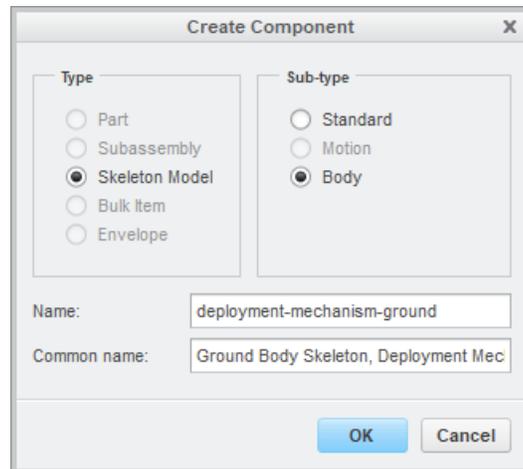
- Right click on the Motion Skeleton and click **Activate**.



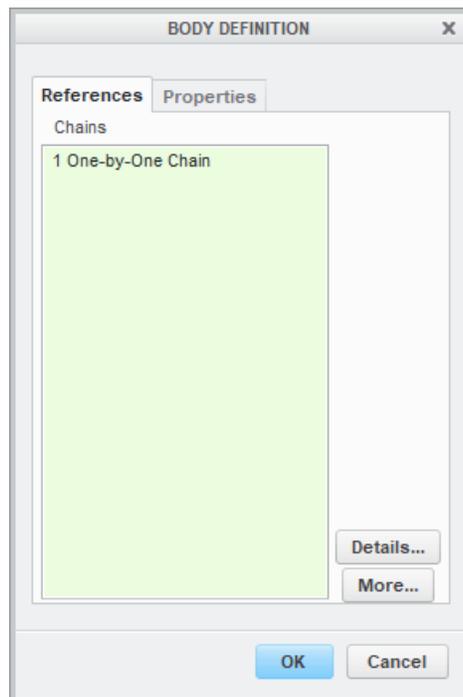
- Click the **Sketch** Sketch tool and pick the Sketch Plane.
- Sketch lines, arcs, circles, and whatever entities you need to represent your motion. Where lines intersect, overlap, or terminate on other entities will define your potential connections at that location. Complete the Sketch.



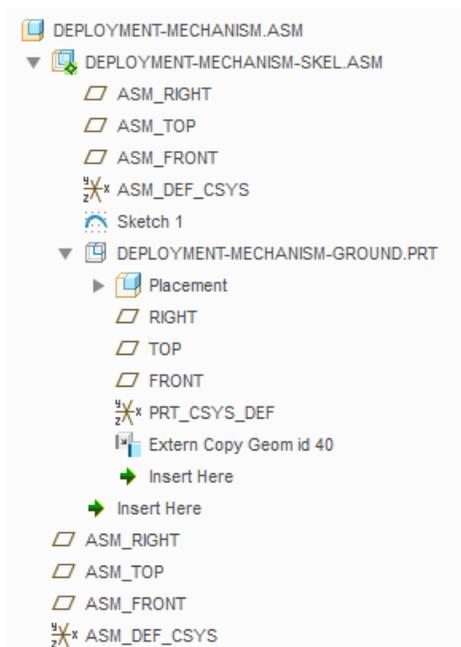
- Click the **Create**  icon. **Skeleton Model** should be the only available Type. Click the radio button for **Body**. Enter the Name and Common Name. Click **OK**.



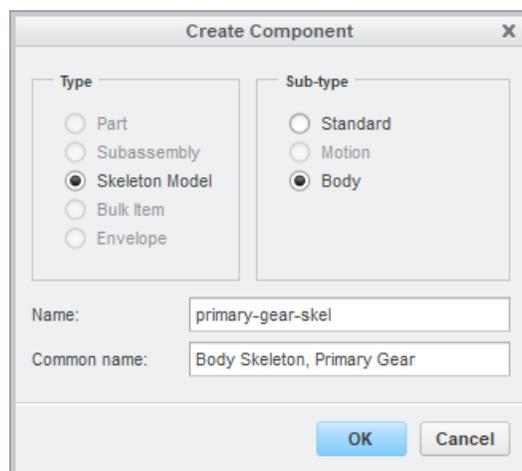
9. In the Creation Options dialog box, ensure that **Copy from existing** is selected. If necessary, click the **Browse** button and navigate to the desired start part. Click **OK**.
10. The Body Definition dialog box will open. In the Graphics Area, select the curve(s) that will represent the Ground for the mechanism. (You may have to use Query Select to pick an entity from the sketch.) Click **OK**.



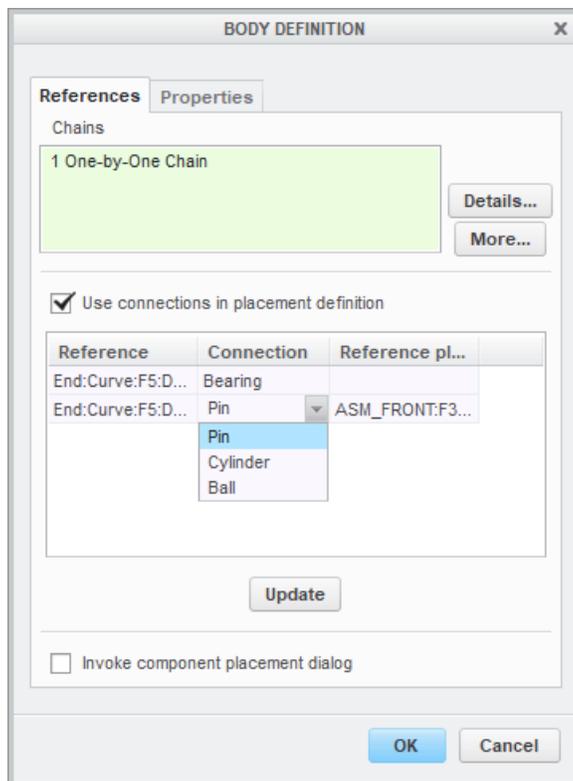
11. The Ground Body Skeleton will be located in the Model Tree. Now we will create the moving parts of the Skeleton.



12. Click the **Create**  icon. Click the radio button for **Body**. Enter the Name and Common Name. Click **OK**.



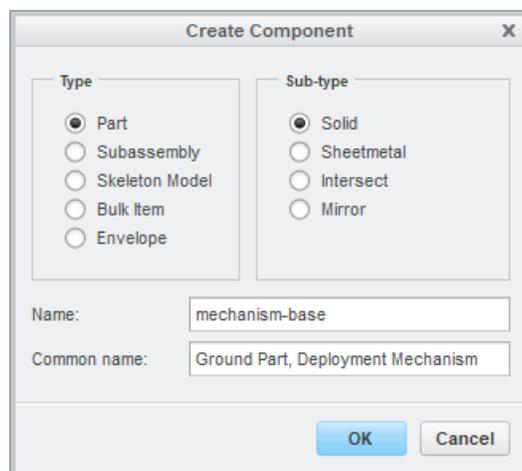
13. In the Creation Options dialog box, ensure that **Copy from existing** is selected. If necessary, click the **Browse** button and navigate to the desired start part. Click **OK**.
14. In the Body Definition dialog box, select the curve(s) for a moving body.
15. Click **Update**. The possible connections will display. If multiple possibilities exist, click on each to examine where it appears in the Graphics Area. Right click on any you do not want and click Remove. You can also click on the type of connection (e.g., Pin) to see if other options are available (e.g., Cylinder or Ball). When you are happy with the remaining connections, click **OK**.



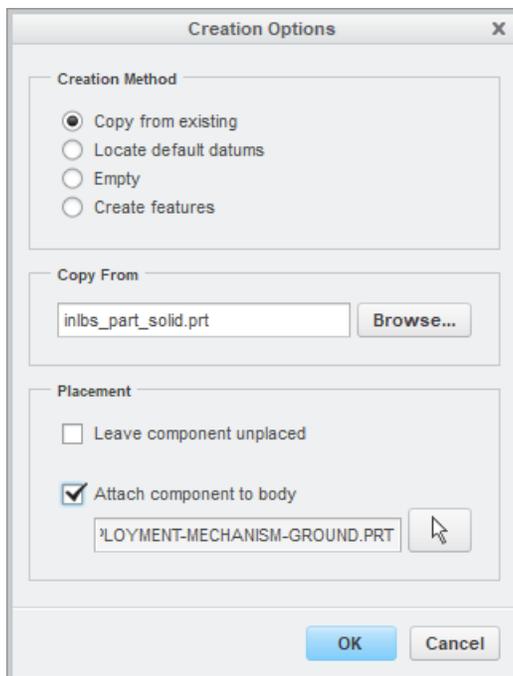
16. Repeat steps 11 through 14 for the remaining moving entities in the Sketch.

17. Use CTRL-A to activate the mechanism assembly.

18. Click the **Create**  icon. Select the **Part** radio button. Enter the Name and Common Name. Click **OK**.

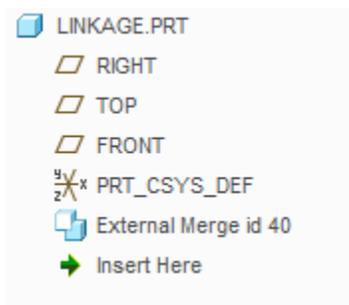


19. The Creation Options dialog box now has an additional option. Check the box for **Attach component to body** and select one of the Body Skeletons from the Graphics Area or Model Tree. Click **OK**.



20. Repeat steps 17 and 18 for the remaining Body Skeletons.

21. Now when you open any of the created parts, you will see it has an External Merge feature. Use the curves (and datum axes) from the Merge to create the necessary part geometry.



22. If desired, apply Motors to your mechanism and perform a Kinematic or Dynamic analysis.

If you decide to attempt a Motion Skeleton, I recommend starting with something relatively simple like a four-bar linkage. Once you develop understanding and skill with that, move onto more complicated mechanisms.

When you start to develop something more complicated, I recommend working it piecemeal. Do not attempt to sketch and build the entire mechanism at once. Create the entities for one

set of interconnected components, build the Body Skeletons, individual parts, and component geometry, then do the following:

1. Use the **Drag**  tool to test the motion.
2. Click **Applications > Mechanism** .
3. Apply a **Motor**  at a connection.
4. Use the **Mechanism Analysis**  tool to run a kinematic or dynamic analysis.

When this works to your liking, develop another moving component in your Motion Skeleton and assembly, and retest. It is much easier to debug a mechanism one step at a time, as opposed to building the entire Motion Skeleton and trying to figure out why it doesn't move the way you want.

4.8. Technique: Turning a Standard Part into a Skeleton

Sometimes a user will create a standard part model and later realize he or she wants it to be a Skeleton. There are two techniques in which a standard part can be turned into a skeleton:

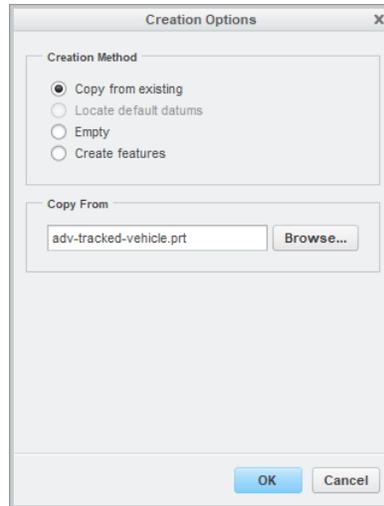
- The **Component Create** command.
- Using **Replace > By Copy**.

4.8.1. Component Create

In this first method, we create a Skeleton using the normal method. However, instead of using a default template or start part, we select the existing part model. The process is as follows:

1. In the assembly, from the Component group choose **Create** .
2. In the Create Component dialog box, select the radio button for **Skeleton**.
3. Enter the Name and Common Name for the Skeleton using your company's naming convention.

- In the Creation Options dialog box, the radio button for **Copy from existing** will be selected by default. Click the **Browse** button to navigate to the previously created part model. Select it and click **OK**.



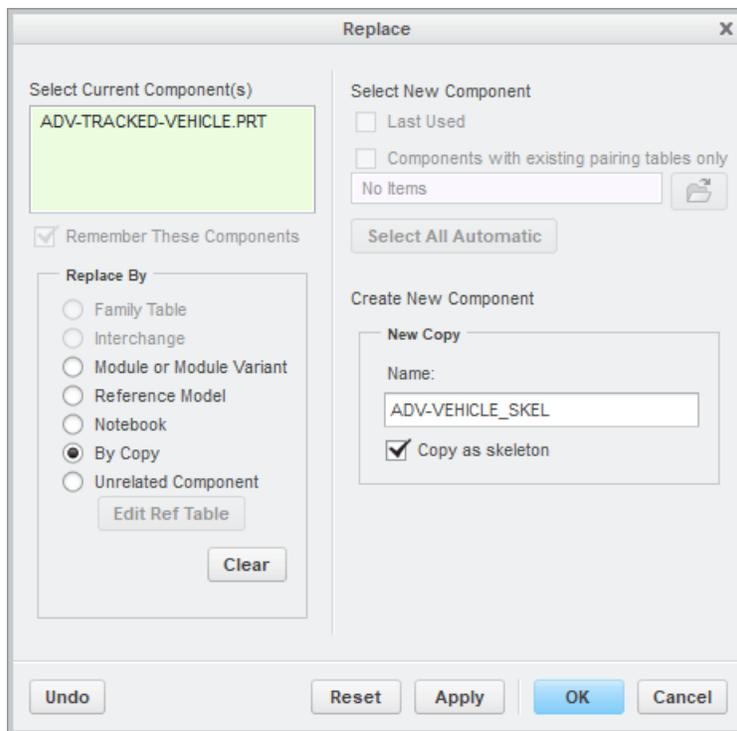
- The rest of the Skeleton creation process is the same as in section 4.4.

All the features and geometry from the part will now be located in the Skeleton.

4.8.2. Replace By Copy

In this technique, we use the Replace By Copy command with the option Copy By Skeleton. To convert a standard part model into a Skeleton, perform the following steps:

- Add the part to the assembly with the Default constraint using either the **Assemble** command from the Component group.
- Drag the part to the top of the Model Tree above the default datum planes. Note 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.
- Right click on the part and select **Replace** .
- In the Replace dialog box, select the radio button for **By Copy**.
- Check the box for **Copy as skeleton**.



6. Edit the name for the Skeleton (following your company's naming conventions) in the Name field.
7. Click **OK**.

Your standard part is now a Skeleton.

4.9.Next Steps

Now that we have a Skeleton, we can reference it for making features in target components. We do this via Data Sharing Features, which is covered in the next chapter.

The Skeleton can also reference critical dimensions and parameters from a Notebook. This is covered in Chapter 6 **Error! Reference source not found.**

I cannot stress enough that Top Down Design is not a linear process. As you are developing your product, you will revisit the Skeleton many times to add, modify, and delete features. You might also get far along the design of individual components and realize that to make your model easier to modify later, or to ensure that your components fit together properly, it could benefit from a Skeleton. As the design of the product continues, so does your work on the Skeleton.

Thanks for reading. If you would like to read more of “Top Down Design in Creo Parametric,” you can find us on Amazon.com. If you like what you read, please write a review. If you didn’t like it or disagree, please feel free to reach out to me.

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