

# Complex Assemblies Made Simple—By Design

Pro/ENGINEER® Tools Deliver Faster Design, Improved Quality

By using a top down design method, extended design teams and individual designers alike have a centralized vault of information for working on tasks concurrently.

Top down design makes it easy to design large, complex product assemblies by giving you a geometry data structure that can be easily controlled, analyzed, and changed.

Top down design in Pro/ENGINEER delivers a series of unique capabilities—largely unavailable in other CAD tools, that address the problems and challenges engineers face when designing large assemblies: you can easily control the assembly at the top level; you can make wide-scale changes from a single location and know that the changes will propagate to all levels of the design; and you can empower large teams to share development tasks and communicate critical design criteria easily and quickly, with full confidence that all components will fit seamlessly into the final product.

To make top down design easy, Pro/ENGINEER offers functionality such as layouts, skeleton models, Copy Geometry, Publish Geometry, and Inheritance features. Using these tools, individuals or design teams can enjoy reduced design time, improved quality, better collaboration, and top-level change control.

## Start by Considering Layouts

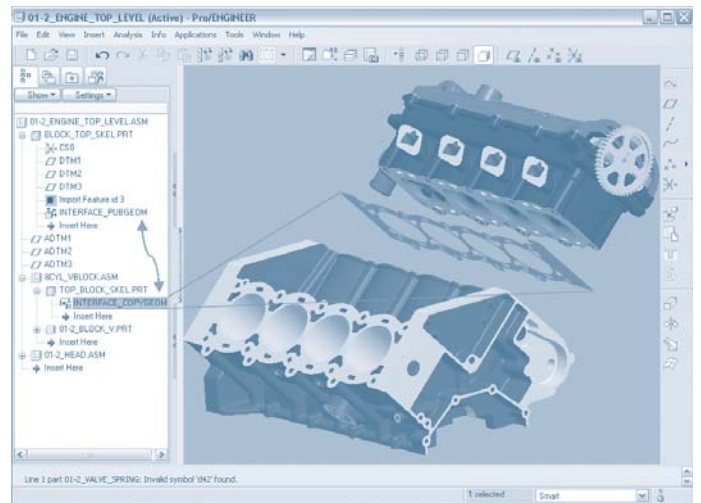
The layout functionality in Pro/ENGINEER features a unique electronic engineering notebook. By using 2D layouts, designs can be conceptualized quickly and easily. The 2D layout can then be used to drive the design, so that changes to the layout will update the design model as the concept evolves.

When used in a top down method, layouts keep the conceptual design information in one central location, and help establish design intent before time and effort is invested in the creation of solid models.

- Tip: Always retrieve the layout into session when retrieving a model that references it. This way, all of the relations required by the model will be available, even without the assembly being in session. This will improve performance and ensure that relations are updated properly.

## Communicating Design Data

A critical aspect of top down design methodology is communicating design data from the assembly to its individual components through a centralized location.



With Pro/ENGINEER data sharing tools, you can publish interface geometry between parts and assemblies. Features can be edited in one place, and the changes are automatically driven into the locations and geometry of lower-level sub-assemblies and parts.

Communication benefits include task distribution, concurrent modeling, and managing external references. By declaring models to 2D layouts, you can distribute the global parameters and datums. And, using geometry features, you can copy references from one model to another.

Steps to Declaring Models to 2D layouts:

1. Under File -> Declare, choose to declare a layout, making all of the parameters and global datums that have been created in the layout visible in the model.
2. Pick a feature in the model and edit it. Pick the dimension and corresponding name from the layout to drive this dimension.
3. Choose DeclareName and select a datum feature from the model, and select a corresponding global name from the layout.

- Tip: Many of these steps will need to be repeated to fulfill design intent.

## Defining the Assembly Structure

By creating an assembly structure, you can essentially create a virtual Bill of Material (BOM). This method defines the major tasks to be completed either by individual designers or the entire design team. This structure can act like a bookmark or placeholder, defining which components will make up the assembly. Having these placeholders not only makes it easy to determine which areas of the assembly still need to be worked on, but also enables other members of the design team to see what else is planned for the design.

Virtual BOMs help you distribute tasks to team members. Engineers can then focus on their individual tasks while keeping their work in the context of the entire assembly. Virtual BOMs also allow the association of library parts upfront, promoting design reuse. They also make it possible to submit models to Windchill® PDMLink™ early in the design cycle to ensure models are assigned to the appropriate vault or folder.

- **Tip:** You can create the virtual BOM from within Pro/INTRALINK® or Windchill PDMLink and then pull the assembly into Pro/ENGINEER.

Steps to Creating a Virtual BOM:

1. Create the top-level assembly. You can enter a name and use the default template, or select another file to copy from.
2. Add empty components or sub-assemblies as required.
3. Add any bulk components, such as grease, to represent items in the BOM that will not be modeled.

### Building Skeletons

A skeleton model is the framework of a design, and acts as the 3D layout of the assembly. Like 2D layouts, skeletons serve as a central location for storing design criteria relating to the assembly, specifically surface geometry, curves, and datums. When the skeleton changes, the solid models connected to it also change.

Skeleton models simplify design creation and visualization, helping you manage parent/child relationships and providing control over external references. In addition, components can be added to an assembly in any order.

- **Tip:** For identification purposes, consider using a naming convention for skeleton models such as “assy-name\_skel.prt”. Also, rename your skeleton datums with a “sk\_” prefix.
- **Tip:** Simple visualization techniques will help communication between team members. Consider using simplified reps and transparency pervasively to make viewing easier. Also, use “Display States” and surfaces to clarify the meaning of centerlines and axes.

Steps to Creating a Skeleton Model:

1. Create a skeleton model.
  2. Choose to work on the skeleton from within the assembly by selecting “Activate”, or work on it from the component level by selecting “Open”.
  3. Use non-solid geometry features such as datum features or surfaces to put “placeholders” in your assembly to represent components; they can also be used to create the interface between components in an assembly or define motion between components.
- **Tip:** Avoid constructing assembly-level skeleton features since the system requires that you perform all edits of these features in assembly mode.

### Publishing and Copying Geometry References

The Publish Geometry feature allows you to mark references that can and should be copied into other components. This method makes it easy for engineers to identify which features in other models are stable and appropriate to use as a reference in the model they are working on.

Steps to Creating a Publish Geometry Feature:

1. In the top-level skeleton model, select Insert -> Shared Data -> Publish Geometry.
  2. Define each of the elements, providing names and selecting any references that need to be available for other skeleton models.
- **Tip:** Name the Publish Geometry feature with respect to the sub-assembly it is to be used in. This will make it easy to determine which subassembly or lower level skeleton that a Publish Geometry feature is intended for.

The Copy Geometry feature is an extremely powerful and flexible feature that is used to share design criteria between parts, assemblies, and sub-assemblies. The design criteria can be any type of datum feature, curves, common interface surfaces, and parameters. Copy Geometry features can also be toggled to be independent or dependent on the original references for ultimate control.

Steps to Creating a Copy Geometry Feature:

1. Constrain your sub-assembly to the top-level assembly.
2. Activate your sub-assembly and create a skeleton model for it. Open it in a separate window.
3. Copy the Published Geometry information to this skeleton model (Insert -> Shared Data -> Copy Geometry from other Model).
4. Select the top-level assembly skeleton file or the top-level assembly and define its location.
5. Define the Publish Geometry element and highlight it the menu.

### Create Model Geometry and Populate the Assembly

Once the information is captured from the top-level skeleton model into the sub-assemblies, you can use the Copy Geometry feature as a reference to build parts and begin populating your assembly.

### Conclusion

Top down design provides many benefits including reduced design time, improved quality and top-level change control. By following these simple tips, your team can realize these benefits first hand.