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CATIA V5-6R2018 Advanced Assembly Design and Management

Learning Guide

1st Edition

ASCENT - Center for Technical Knowledge®
CATIA V5-6R2018: Advanced Assembly Design and Management

1st Edition

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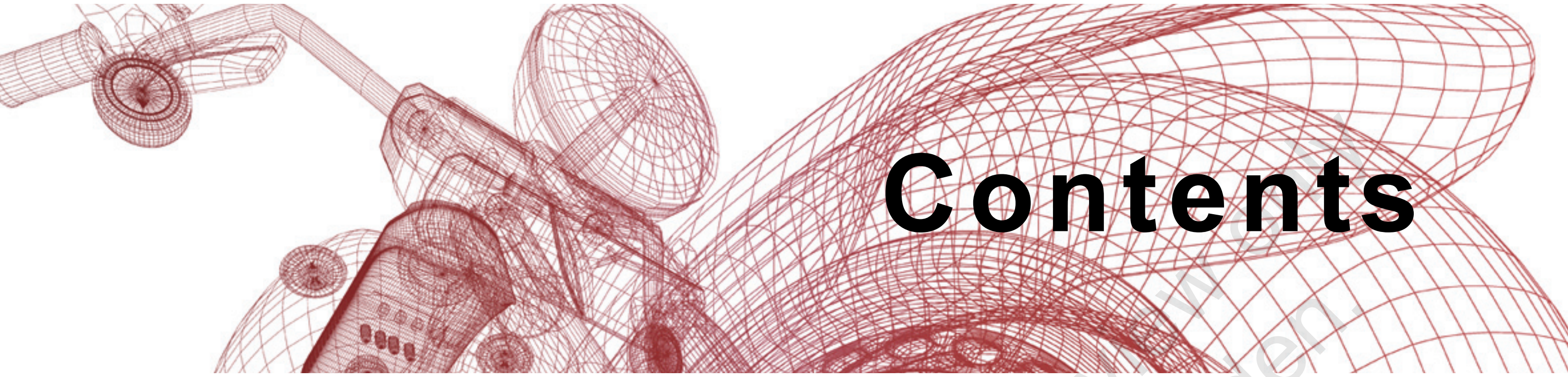
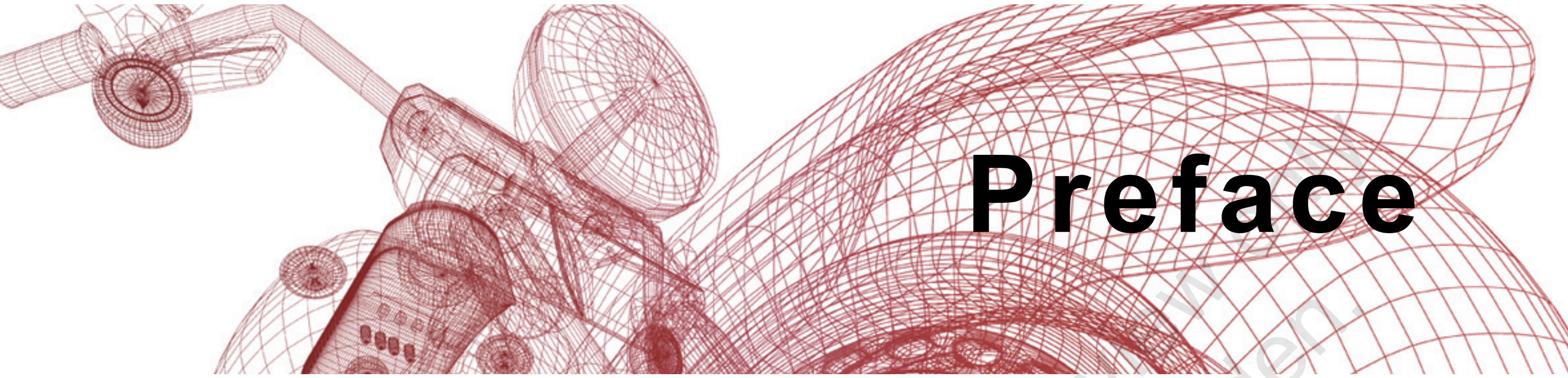


Contents



Preface

The CATIA V5-6R2018: Advanced Assembly Design and Management learning guide builds on the assembly functionality introduced in the CATIA: Introduction to Modeling course. Students gain a full understanding of how to design and manage a complex assembly in the CATIA software while concentrating on techniques that maximize the capabilities of the Assembly workbench. This extensive hands-on course contains numerous labs focused on process-based practices to give you practical experience and improve design productivity.

Topics Covered

- Assembly operations (reconnecting constraints, specification tree customization, save operations, Desk Command, etc.)
- Skeleton Modeling
- Contextual Design
- Publications
- Link Management
- Collaborative Design
- Component Degrees of Freedom
- Assembly Duplication (multi-instantiation, component symmetry, reuse patterns, etc.)
- Assembly analysis (measurements, clash, sectioning a model, etc.)

Prerequisites

- Access to CATIA V5-6R2018 software. The practices and files included with this guide might not be compatible with prior versions.
- Completion of the *CATIA V5-6 R2018: Introduction to Modeling* course and an additional 80 hours of CATIA experience are recommended.

Note on Software Setup

This guide assumes a standard installation of the software using the default preferences during installation. Lectures and practices use the standard software templates and default options for the Content Libraries.

Lead Contributor: Scott Hendren

Scott Hendren has been a trainer and curriculum developer in the PLM industry for over 20 years, with experience on multiple CAD systems, including Pro/ENGINEER, Creo Parametric, and CATIA. Trained in Instructional Design, Scott uses his skills to develop instructor-led and web-based training products.

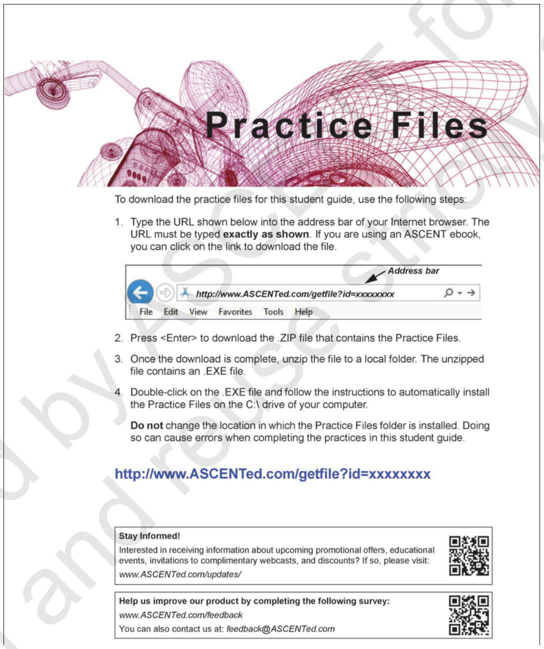
Scott has held training and development positions with several high profile PLM companies, and has been with the Ascent team since 2013.

Scott holds a Bachelor of Mechanical Engineering Degree as well as a Bachelor of Science in Mathematics from Dalhousie University, Nova Scotia, Canada.

Scott Hendren has been the Lead Contributor for CATIA: Advanced Assembly Design and Management since 2013.

In this Guide

The following images highlight some of the features that can be found in this guide.


Practice Files

To download the practice files for this student guide, use the following steps:

1. Type the URL shown below into the address bar of your Internet browser. The URL must be typed **exactly as shown**. If you are using an ASCENT ebook, you can click on the link to download the file.

2. Press <Enter> to download the .ZIP file that contains the Practice Files.
3. Once the download is complete, unzip the file to a local folder. The unzipped file contains an .EXE file.
4. Double-click on the .EXE file and follow the instructions to automatically install the Practice Files on the C:\ drive of your computer.

Do not change the location in which the Practice Files folder is installed. Doing so can cause errors when completing the practices in this student guide.

http://www.ASCENTed.com/getfile?id=xxxxxxxx

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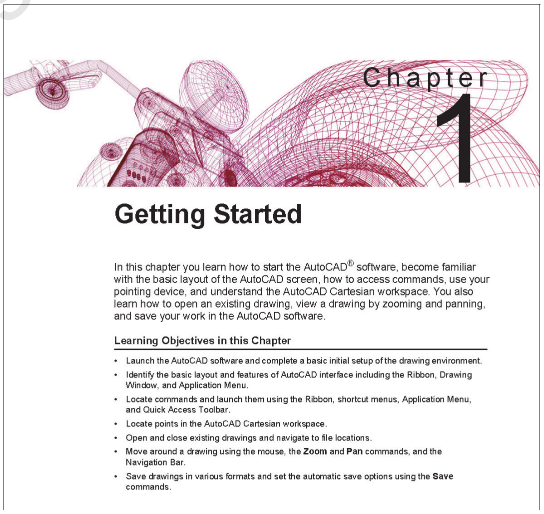
Link to the practice files

Practice Files

The Practice Files page tells you how to download and install the practice files that are provided with this guide.


Chapter 1

Getting Started

In this chapter you learn how to start the AutoCAD® software, become familiar with the basic layout of the AutoCAD screen, how to access commands, use your pointing device, and understand the AutoCAD Cartesian workspace. You also learn how to open an existing drawing, view a drawing by zooming and panning, and save your work in the AutoCAD software.

Learning Objectives in this Chapter

- Launch the AutoCAD software and complete a basic initial setup of the drawing environment.
- Identify the basic layout and features of AutoCAD interface including the Ribbon, Drawing Window, and Application Menu.
- Locate commands and launch them using the Ribbon, shortcut menus, Application Menu, and Quick Access Toolbar.
- Locate points in the AutoCAD Cartesian workspace.
- Open and close existing drawings and navigate to file locations.
- Move around a drawing using the mouse, the **Zoom** and **Pan** commands, and the Navigation Bar.
- Save drawings in various formats and set the automatic save options using the **Save** commands.

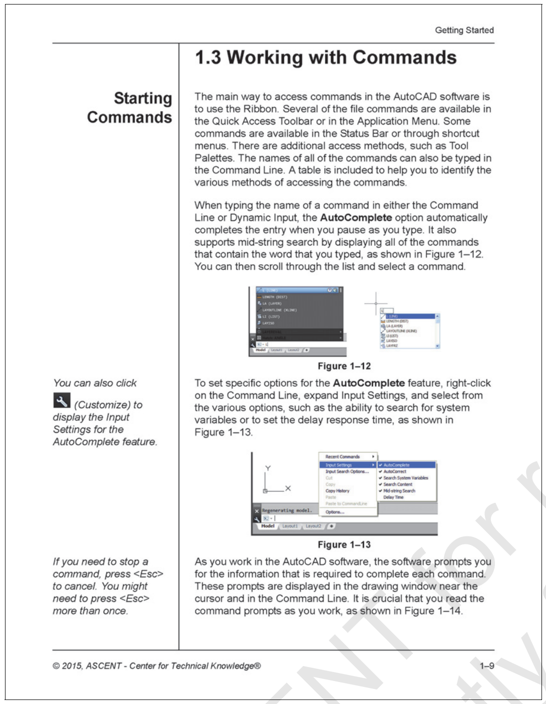
Chapters

Each chapter begins with a brief introduction and a list of the chapter's Learning Objectives.

Learning Objectives for the chapter

Side notes

Side notes are hints or additional information for the current topic.



Getting Started

1.3 Working with Commands

Starting Commands

The main way to access commands in the AutoCAD software is to use the Ribbon. Several of the file commands are available in the Quick Access Toolbar or in the Application Menu. Some commands are available in the Status Bar or through shortcut menus. There are additional access methods, such as Tool Palettes. The names of all of the commands can also be typed in the Command Line. A table is included to help you to identify the various methods of accessing the commands.

When typing the name of a command in either the Command Line or Dynamic Input, the **AutoComplete** option automatically completes the entry when you pause as you type. It also supports mid-string search by displaying all of the commands that contain the word that you typed, as shown in Figure 1–12. You can then scroll through the list and select a command.

Figure 1–12

You can also click (Customize) to display the Input Settings for the AutoComplete feature.

To set specific options for the **AutoComplete** feature, right-click on the Command Line, expand Input Settings, and select from the various options, such as the ability to search for system variables or to set the delay response time, as shown in Figure 1–13.

Figure 1–13

If you need to stop a command, press <Esc> to cancel. You might need to press <Esc> more than once.

As you work in the AutoCAD software, the software prompts you for the information that is required to complete each command. These prompts are displayed in the drawing window near the cursor and in the Command Line. It is crucial that you read the command prompts as you work, as shown in Figure 1–14.

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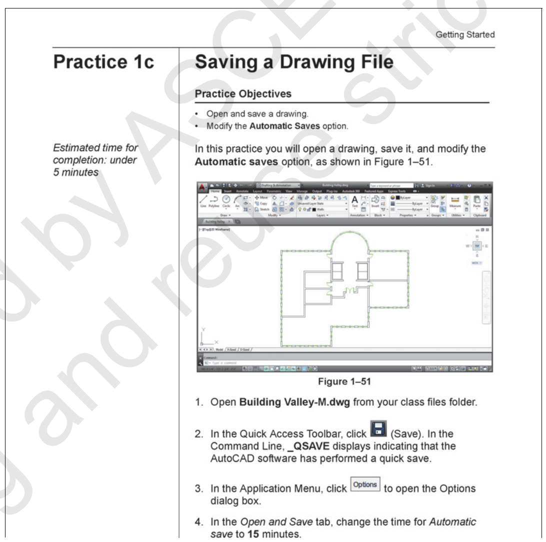
Instructional Content

Each chapter is split into a series of sections of instructional content on specific topics. These lectures include the descriptions, step-by-step procedures, figures, hints, and information you need to achieve the chapter's Learning Objectives.

Practice Objectives



Getting Started

Practice 1c Saving a Drawing File

Practice Objectives

- Open and save a drawing.
- Modify the **Automatic Saves** option.

Estimated time for completion: under 5 minutes

In this practice you will open a drawing, save it, and modify the **Automatic saves** option, as shown in Figure 1–51.

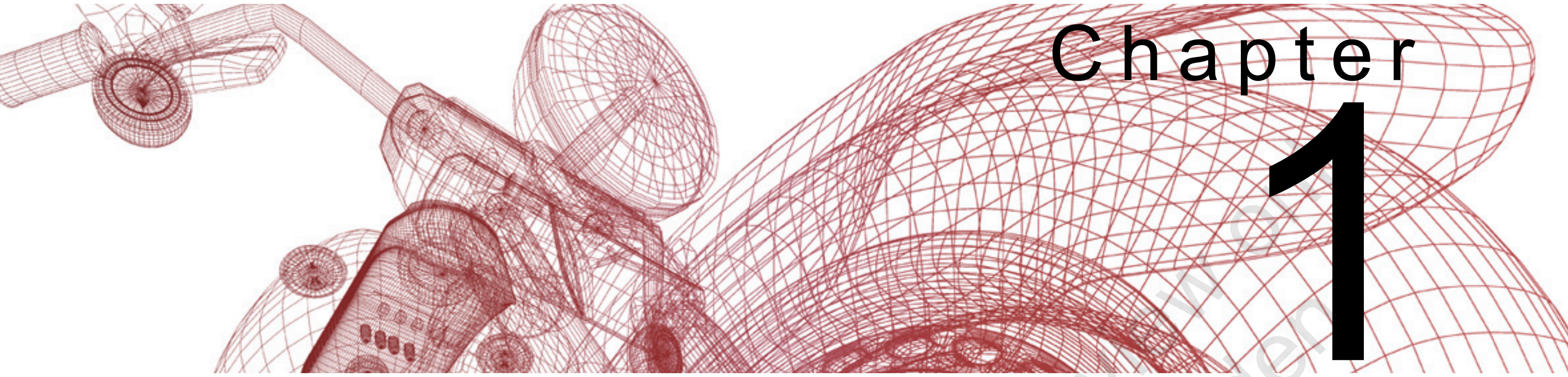
Figure 1–51

1. Open **Building Valley-M.dwg** from your class files folder.
2. In the Quick Access Toolbar, click (Save). In the Command Line, **_QSAVE** displays indicating that the AutoCAD software has performed a quick save.
3. In the Application Menu, click Options to open the Options dialog box.
4. In the *Open and Save* tab, change the time for *Automatic save* to **15** minutes.

Practices

Practices enable you to use the software to perform a hands-on review of a topic.

Some practices require you to use prepared practice files, which can be downloaded from the link found on the Practice Files page.



Chapter 1

Assembly Operations

The Assembly Design workbench enables you to explore various design configurations and locate missing files. Operations, such as assembly constraints and constraint creation modes, help you to create a flexible assembly. This chapter introduces operations in the Assembly Design workbench, and compares top-down design techniques to bottom-up design techniques when creating assemblies.

Learning Objectives in this Chapter

- Review the Assembly Design Workbench and assembly related terms and definitions.
- Understand the Assembly Specification Tree.
- Learn how to work with assembly annotations.
- Learn how to work with assembly constraints.
- Understand the Save operations and the Desk command.

1.1 Assembly Design Workbench

When designing parts in the Part Design workbench, features are created and positioned parametrically with respect to each other and to other reference features. Parts that belong to an assembly can be assembled and positioned parametrically in a CATProduct file using the Assembly Design workbench.

Accessing the Assembly Design Workbench

By default, CATIA opens in the Assembly Design workbench. You can also activate the Assembly Design workbench by selecting **Start>Mechanical Design>Assembly Design** or **File>New>Product**.

When the Assembly Design workbench is activated, various assembly-specific toolbars open. The Product Structure toolbar shown in Figure 1–1 enables you to assemble components, create part and product files in context, replace components, manage an assembly, and multi-instantiate components.

Figure 1–1

1.2 Terms and Definitions

Skeleton

A skeleton is a CATIA part model that is used to aid the construction of a complex assembly model. The part is used as a storage location for any critical design information, such as industrial design surfaces (e.g., A-side or masterline surfaces), and location and sizing information. The skeleton information is shared with the other components in the assembly to drive the design. The skeleton model is always the first component in an assembly and should only contain wireframe and surface geometry so that assembly mass properties are not affected.

Contextual Design

Contextual design involves the creation of part-level geometry in the context of an assembly. By visualizing all of the components of an assembly while building a part, it is possible to share critical design information between components and ensure that components do not interfere with each other.

The contextual design approach is driven by the creation of external references between assembly components. These links add an extra level of complexity to the assembly and must be maintained throughout the design process to obtain maximum benefits. Building an assembly contextually requires good communication in the design team and a clear reference structure in the assembly.

Bottom-Up Design

In a traditional bottom-up design approach, part geometry is created independent of the assembly or any other component. Any design criteria established before the part is modeled are not shared between models. Once all of the part models are completed, they are brought together for the first time in the assembly. At this point in the design, problems often result with the assembly because engineering information is not correctly shared or communicated. Problems can include interference between components, misalignment between components, or incomplete design. In addition, any modifications to components must be manually propagated throughout the assembly.

Top-Down Design

The top-down design approach places critical information in a top-level assembly and then communicates that information to lower levels of the product structure. The first step in creating a top-down design model is to create an initial assembly structure. Design information is placed in this assembly through the use of skeleton models and parameters that are controlled by design tables. Any changes made to the top-level information are automatically propagated to all affected components.

Top-down design techniques simulate a design team and facilitate concurrent engineering. The top-down design approach forces you to consider all areas of a final model before creating any geometry. Consider the following questions when using this technique:

- What does the assembly do?
- How does a specific model interface with other components in the assembly?
- What are the inputs and outputs of the assembly?

Planning the assembly using the top-down design approach helps to create clean, reusable geometry that interfaces correctly with the rest of the assembly.

Collaborative Design

Collaborative design involves two or more people simultaneously developing geometry for an assembly. For example, when designing a car, several departments contribute to the finished product.

A major concern in a collaborative environment is the loss of data or duplication of efforts. If two people open a part model at the same time, the last person to save defines the latest revision, while the first person's modifications are lost. Communication is one defence against these types of setbacks. Another solution is to install a Product Data Management (PDM) application.

Product Data Management

Product Data Management (PDM) is a type of software that organizes and manages files in a database. Files in a PDM system are related to the development of a product. These files are stored on a server commonly referred to as the vault. From here, you can open files and save them back to the server. The PDM system keeps track of and controls all file operations. Only one person can work on a file at a time. All other users can only display the file, but not make changes to it.

Common capabilities of a PDM system include:

- Tracking revisions of a document.
- Advanced tools to search documents in the database.
- Viewing file information.
- Managing change orders.
- Managing bill of materials.
- Permissions control over files.

1.3 Assembly Specification Tree

This section discusses the following methods of customizing the specification tree for a CATIA Product:

- Node Customization
- Graph Tree Reordering

Node Customization

When working in the Assembly Design workbench, the specification tree displays component part numbers and instance numbers by default.

Select **Tools>Options** to open the Options dialog box. Expand **Infrastructure** and select **Product Structure** in the tree. Select the *Nodes Customization* tab. The nodes of the specification tree can be customized to report information, such as **Description**, **Revision**, and **Source** (vendor information).

Figure 1–2 shows the *Nodes Customization* tab.

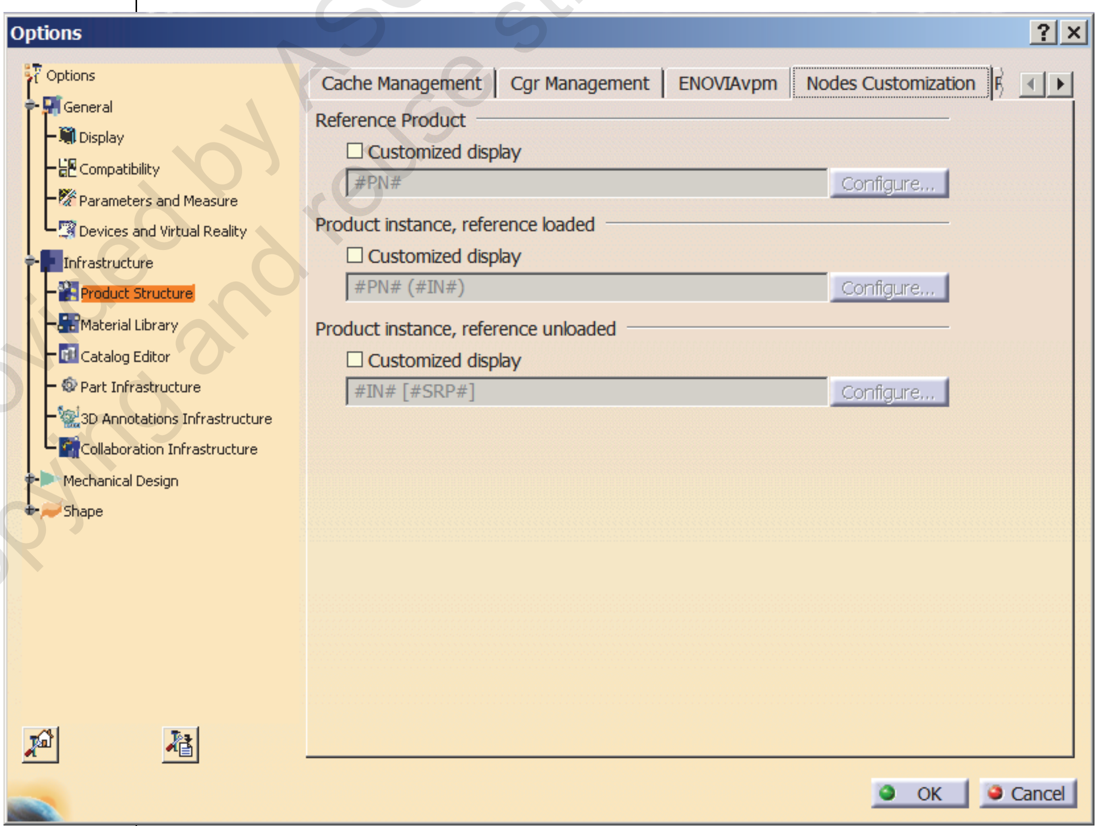


Figure 1–2

Graph Tree Reordering

You can reorder the children of the top level product or any of its subassemblies.

The Graph tree reordering element is useful for reordering components of a product.

How To: Reorder Parts in a Product

1. In Product Structure Tools toolbar, click  (Graph tree reordering) and select the product whose children are to be reordered. The Graph tree reordering dialog box opens, as shown in Figure 1–3.

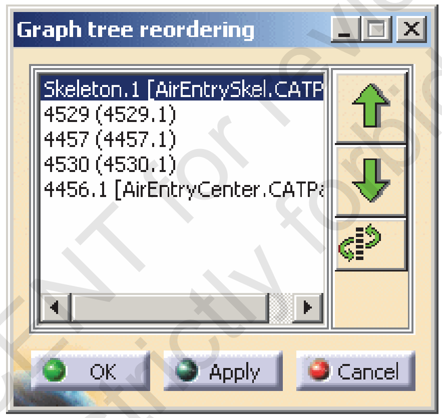


Figure 1–3

2. Use the following options to reorder components:
   - Click  to move the selected components up one step in the product specification tree.
   - Click  to move the selected component down one step in the product specification tree.
   - Click  and another component to switch its position in the specification tree.
3. Click **Apply** to apply changes.
4. Click **OK** to complete the Graph tree reordering feature.

1.4 Assembly Annotations

Annotations can be added to an assembly to identify various parts and components. The Annotations toolbar is shown in Figure 1–4. The following types of annotations can be created:

- Weld Feature
- Text with Leader
- Flag Note with Leader
- Front View/Annotation Plane
- 3D-Annotation-Query Switch On/Switch Off

Figure 1–4

Click  (Text with Leader) to create a text note and then select an element to attach the leader. The Text Editor dialog box opens, as shown in Figure 1–5 in which you can enter text.

To add another line of text to a note, press <Shift>+<Enter>.

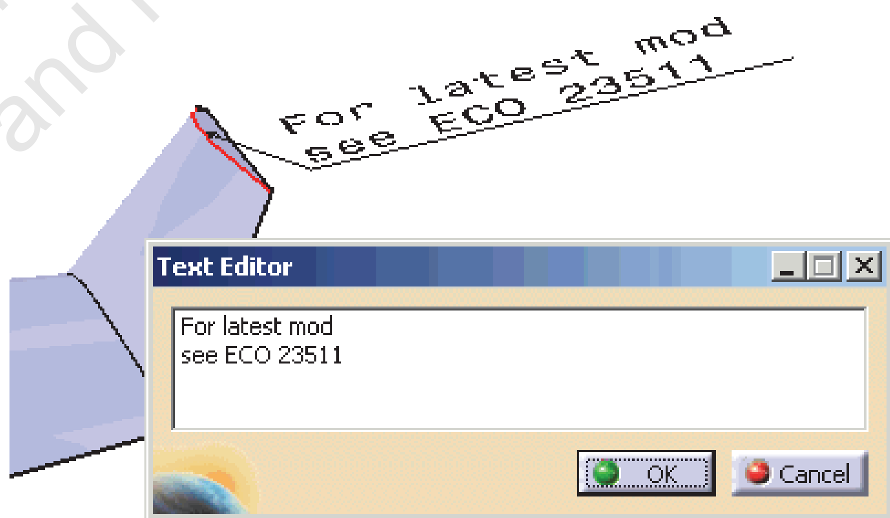


Figure 1–5

Views can be hidden.

Annotation features are added under the Annotation node in the specification tree. For example, a front view is automatically created with annotations, as shown in Figure 1–6.

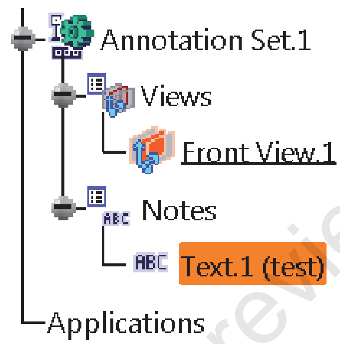


Figure 1–6

Multiple notes can be associated with a single view. A view is useful for displaying all of the notes with one model orientation setting.

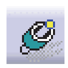
1.5 Assembly Constraints

When creating features for part models, parent/child relationships result from geometrical, dimensional, and depth option references between features. When working with parts in a Product file, parent/child relationships are established through assembly constraints.

Assembly constraints are created using the Constraints toolbar shown in Figure 1–7.

Figure 1–7

The constraints are described as follows:

Icon	Description
	**Coincidence:** Aligns axes, planar surfaces, planes, and points.
	**Contact:** Mates two planar surfaces and can force curved surfaces to touch.
	**Offset:** Specifies an offset distance between two planar elements.
	**Angle:** Permits a keyed-in value between planar selections. Parallel and perpendicular can also be specified.
	**Fix:** Constrains a component in 3D space. This option constrains all six degrees of freedom.
	**Fix Together:** Prompts you for a name and applies a Fix constraint between two or more components.
	**Quick Constraint:** Enables the system to automatically select the constraint to use, based on your selection. This constraint can be changed later.

To change the type of existing constraint, click  (Change Constraint) and select the new constraint type in the Possible Constraints dialog box, as shown in Figure 1–8.

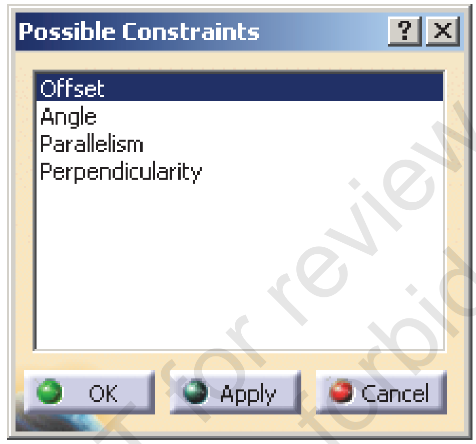


Figure 1–8

Constraint Creation

Three different constraint creation modes can be activated. The system stays in the selected mode until a different mode has been selected. The Constraint Creation toolbar is shown in Figure 1–9.

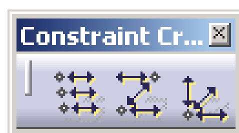
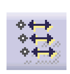
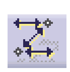


Figure 1–9

The three constraint creation modes are described as follows:

Mode	Description
	**Default mode:** Selects references selected between two components.
	Chain mode: Selects references from multiple components to be incrementally offset.
	Stack mode: Selects a common reference for multiple components.

1.6 Reconnecting Constraints

If an incorrect constraint reference is selected or if the design requires a change to the references of an existing constraint, the constraint must be reconnected. Two components with constraints that need to be reconnected are shown in Figure 1–10.

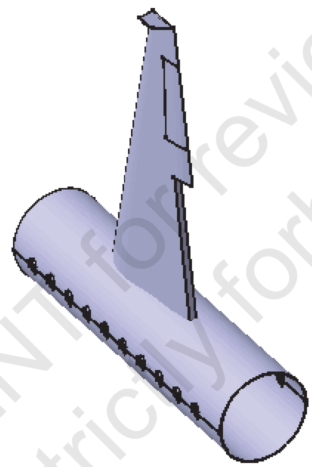


Figure 1–10

General Steps

Use the following general steps to reconnect a constraint:

1. Edit the constraint.
2. Select new reference(s).
3. Update the assembly.

Step 1 - Edit the constraint.

Double-click on the constraint in the specification tree. Click **More** in the Constraint Definition dialog box, as shown in Figure 1–11.

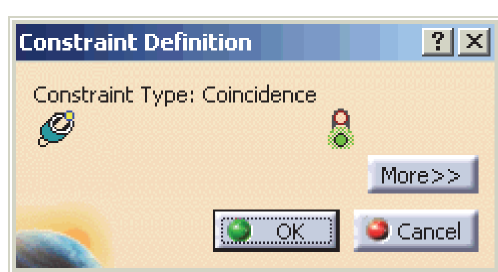


Figure 1–11

Double-click on the *Connected in the Status* column of the reference to be changed. The Plane reference of Fuselage part is being redefined, as shown in Figure 1–12.

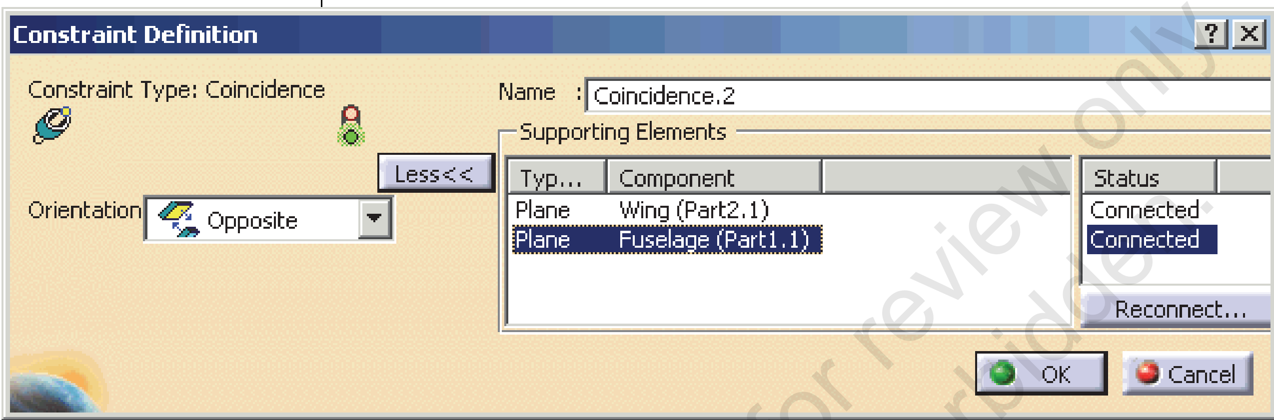


Figure 1–12

Step 2 - Select new reference(s).

Select a new element to be referenced in the specification tree or on the display. Click **OK** to complete the constraint definition.

Step 3 - Update the assembly.

Click  (Update All) to update the assembly constraints. The updated assembly displays, as shown in Figure 1–13.

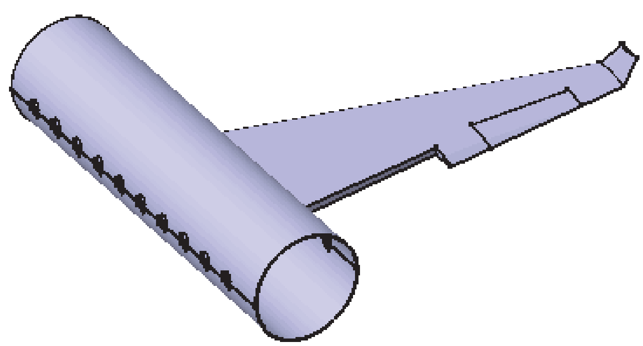


Figure 1–13

1.7 Save Operations

New or modified files should be saved frequently to prevent data loss. Files can be saved using a variety of options:

- Save
- Save As
- Save All
- Save Management
- Send To

Save

To save a file without renaming it, click  (Save) in the Standard toolbar or select **File>Save**. Using this option requires no further input from you.

Save As

If the file is being saved for the first time, the **Save As** option is performed automatically. You can also use this option to rename a file or save a file to another format. Select **File>Save As** to open the Save As dialog box, as shown in Figure 1–14. The current name of the file displays in the *File name* field. To rename the file, replace the name in the *File name* field with the required name. Click **Save** to save the file to the hard drive.

By default, the file is saved as a .CATProduct file. You can change the type of file by selecting the file format in the ***Save as type*** *drop-down list.*

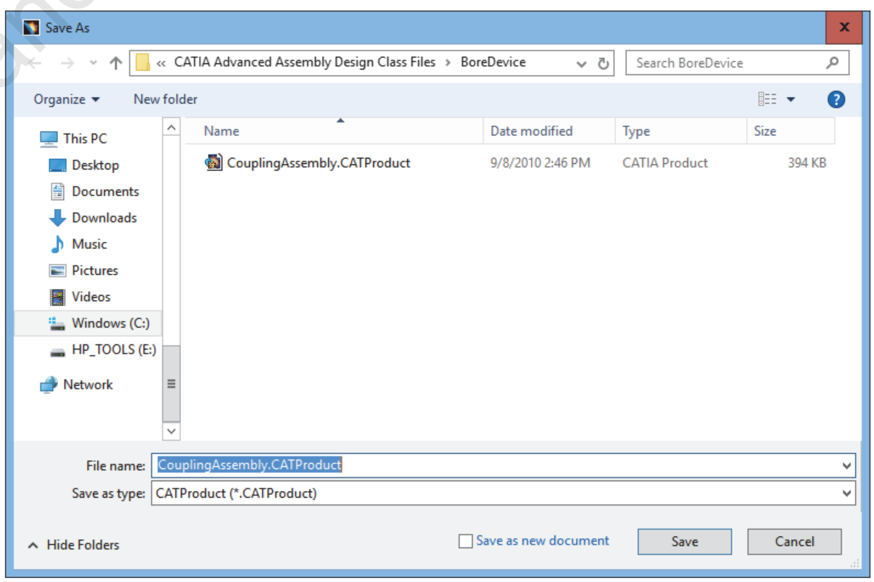


Figure 1–14

Save All

The **Save All** option performs the **Save** command on all open modified documents. This option enables you to save several open documents simultaneously. To perform the operation, select **File>Save All**. If the **Save** operation can be performed on the documents without any user input, a prompt box opens, as shown in Figure 1–15. Click **Yes** to save all of the modified open documents.

If any of the files to be saved requires additional user input, the prompt box shown in Figure 1–16 opens. Click **OK** to continue.

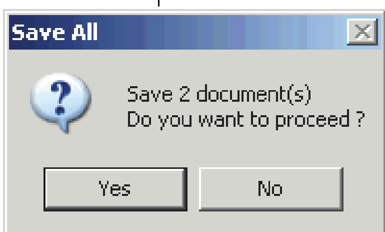


Figure 1–15

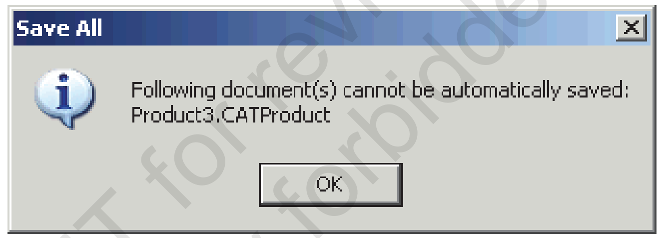


Figure 1–16

If documents cannot be automatically saved, the Save All dialog box opens. It lists all of the open modified files that require additional input to be saved. For example, in Figure 1–17 two files require additional input before they can be saved. The first file is a new file, indicating that it has never been saved to the hard drive. The second file is a read-only file and cannot be saved to the same location with the same name. In both cases, select the file and click **Save As** to perform a Save As operation on the selected file. Once all of the files listed in the window have had a **Save As** performed on them, click **OK** to complete the **Save All**.

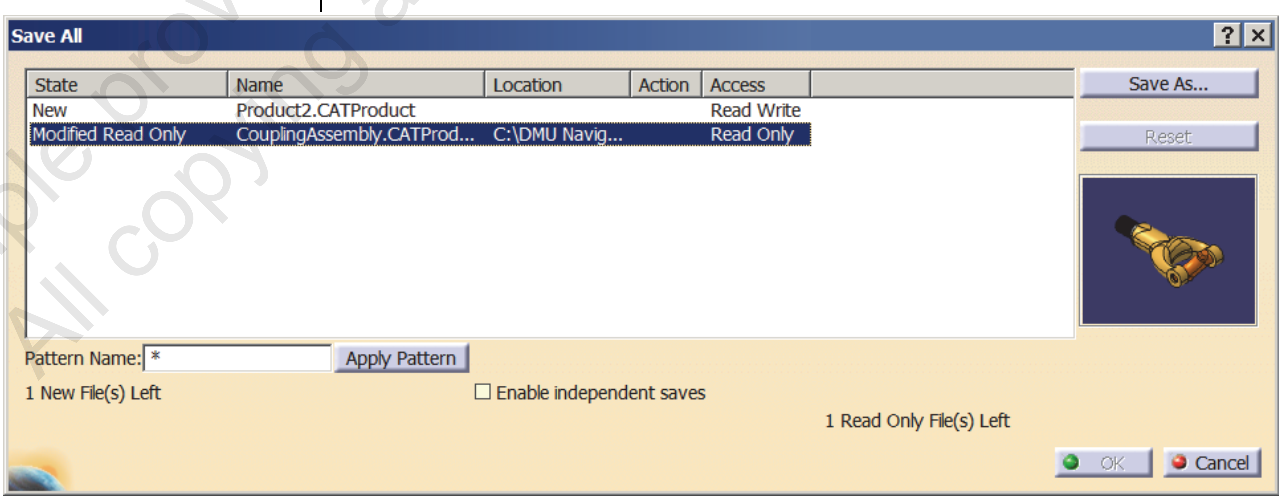


Figure 1–17

Save Management

The **Save Management** option enables you to control where all open files are saved. This option is useful when you need to rename multiple files.

How To: Rename the Part Files of a Product

1. Select **File>Save Management**.
2. Select the part file in the Save Management dialog box, as shown in Figure 1–18.

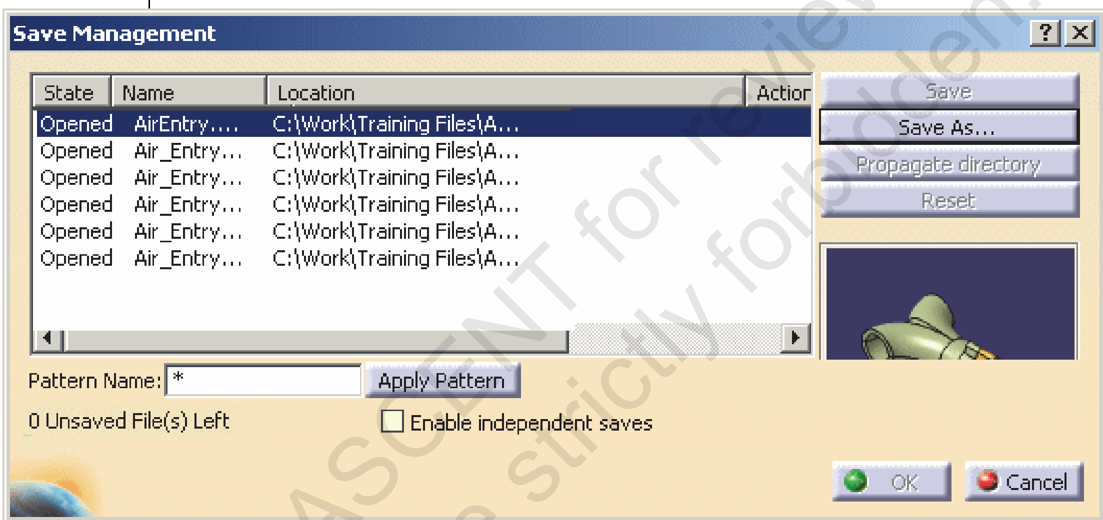


Figure 1–18

3. Click **Save As** and enter new name for the part file.

Save Management is also useful for exploring an alternative product development path. **Propagate directory** enables you to create a copy of a complete assembly in a different directory. The part and product files from the new directory can then be modified and re-configured without affecting the original product and part files.

How To: Propagate a Directory

1. Select a product file in the Save Management dialog box.
2. Click **Save As**. Specify a different directory to save the product file.
3. Click **Propagate directory**. The system saves a copy of all of the part and product files associated with the selected product file to the new directory.

You are prompted to use Save Management when attempting to save a file with links to other modified documents. For example, when a product is saved and contains a part that has been altered, the prompt box shown in Figure 1–19 opens.

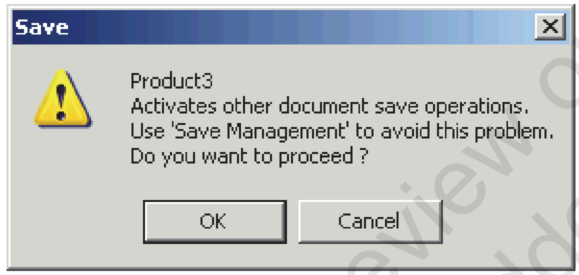


Figure 1–19

If you proceed with the **Save** option, only the selected document is saved and not the other modified documents. Instead, try clicking **Cancel** to abort the **Save** option and then using the **Save Management** or **Save All** option to avoid problems.

Send To

The **Send To** option copies a product and all of its linked files to a specified directory or attaches them to an e-mail. This option ensures that all of the files required to open a product file are included in an e-mail or moved with the product file.

How To: Perform a Send To Operation

1. Select **File>Send To>Mail** or **File>Send To>Directory**. The Send to dialog box opens. The top window lists the selected product file and all of the files linked to it, as shown in Figure 1–20.

*In this example, **Send To Directory** was selected. The Send To Mail dialog box is the same but does not have the Copy to field at the bottom of the dialog box.*

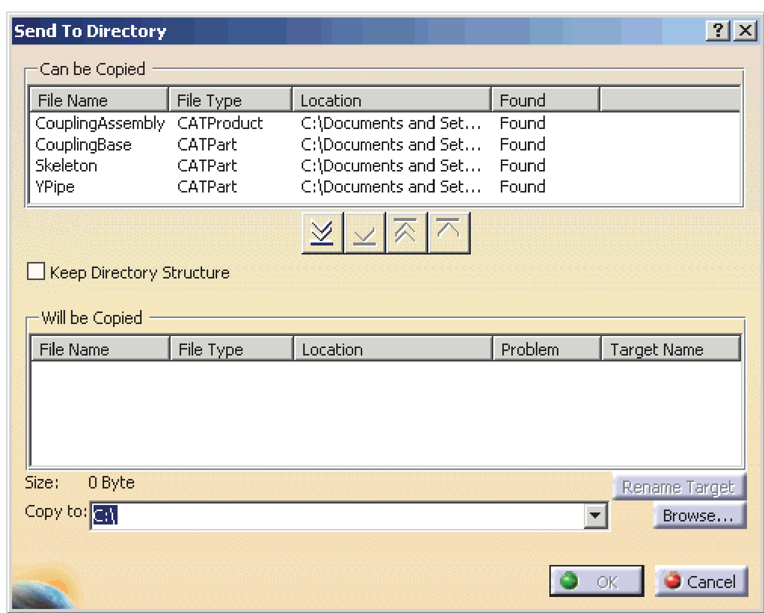


Figure 1–20

2. Click  to copy the product and all of its associated files to the *E-mail* directory. The files move from the top window to the bottom window, as shown in Figure 1–21.

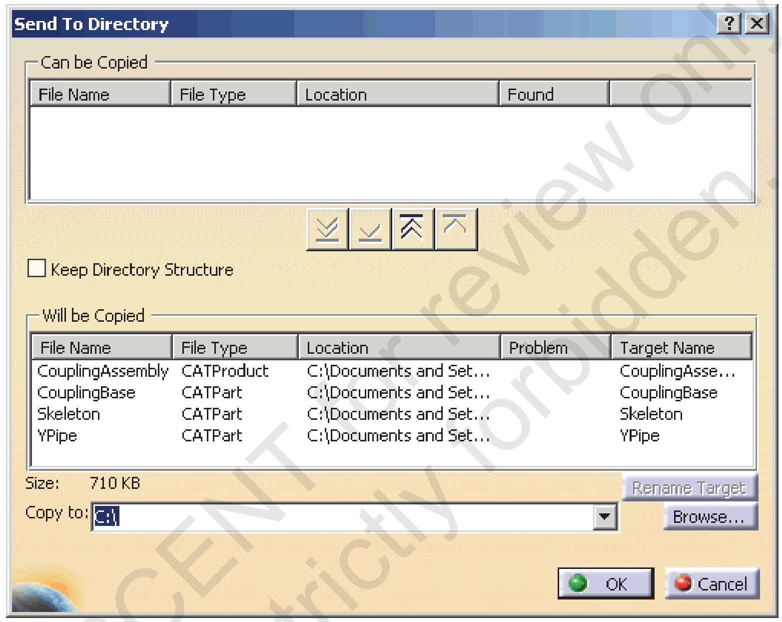


Figure 1–21

3. If the **Send To Directory** operation is performed, the files are copied to the directory indicated in the *Copy to* field. To change the directory, click **Browse** and locate the correct directory. The *Copy To* field updates to reflect the change.
4. Click **OK** to complete the copy.
5. If you have selected to **Send To Mail**, an e-mail opens with the copied files attached. If you have selected **Send To Directory**, a message window opens, notifying you that the copy was successful, as shown in Figure 1–22.

Figure 1–22

1.8 Desk Command

When a CATIA product model is created, file paths to component files (e.g., *.CATPart and *.CATProduct) are written to the Product file. If the system cannot locate these files during retrieval, a message window opens, similar to the one shown in Figure 1–23.

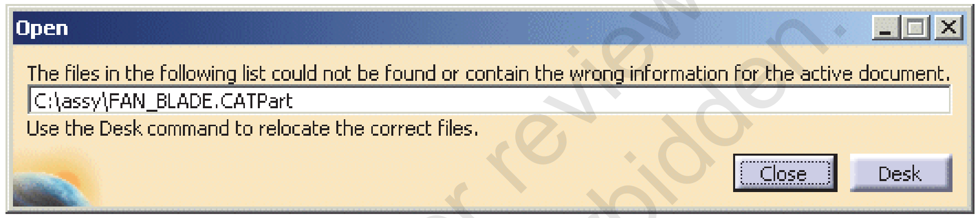


Figure 1–23

If **Close** is clicked, the system does not include the missing component in the assembly.

If **Desk** is clicked, a Desk window opens, displaying the assembly and its components in a tree. Any missing components are highlighted in red. To locate the missing component, right-click on it in the tree and select **Find**, as shown in Figure 1–24.

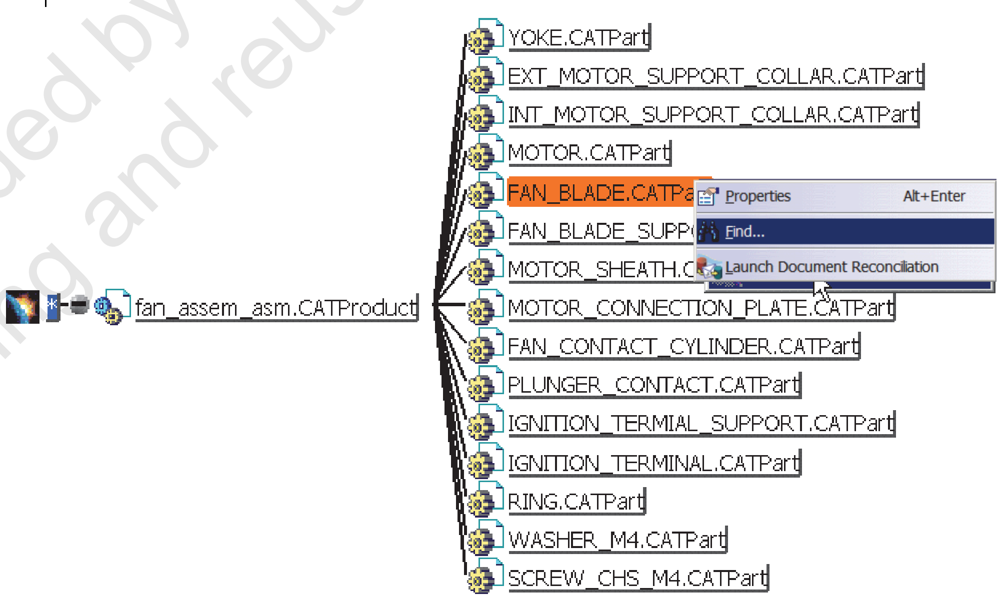


Figure 1–24

6. The system then opens a File Selection dialog box for you to browse for the missing component. Once the component has been located, the Desk window can be closed.

Practice 1a

Assembly Creation

Practice Objectives

- Create a Product file.
- Assemble components.

In this practice, you will create an assembly using a variety of assembly constraints. The completed model displays, as shown in Figure 1–25.

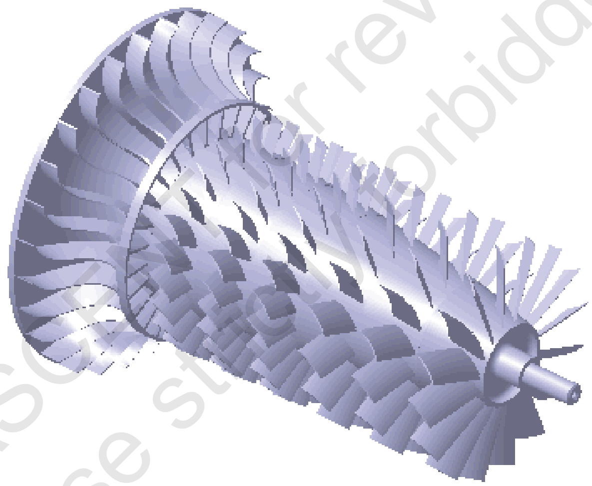
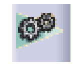


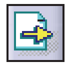
Figure 1–25

Task 1 - Create a Product file.

1. Select **File>New** and create a new Product file.
2. In the specification tree, right-click on **Product1** and select **Properties**.
3. For the part number for the product, enter **71499** and click **OK**.
4. Select **Tools>Options** to open the Options dialog box. Expand **General** and select **Parameters and Measure**.
5. Select the *Units* tab.
6. Set *units for length* to **Millimeters(mm)**.
7. Save the product in the Turbine directory with the name **CompressorRotor**.

Task 2 - Assemble Impeller.CATPart.

1. Ensure that the Assembly Design workbench  is active. If not, select **Start>Mechanical Design>Assembly Design**.

2. Click  (Existing Component). In the specification tree, select **71499**.

3. In the *Turbine* directory, select **Impeller.CATPart**.

4. In the Constraints toolbar, click  (Fix Component) and select the **Impeller** model. The component displays with the fix component anchor symbol, as shown in Figure 1–26.

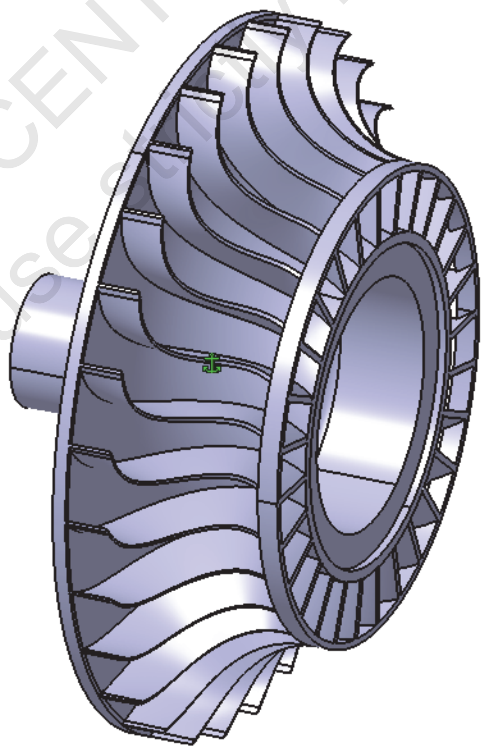


Figure 1–26

Task 3 - Define Node Customization for the specification tree.

1. Select **Tools>Options** to open the Options dialog box.

2. Expand **Infrastructure** and select **Product Structure**.

3. Select the *Nodes Customization* tab. The Options dialog box opens, as shown in Figure 1–27.

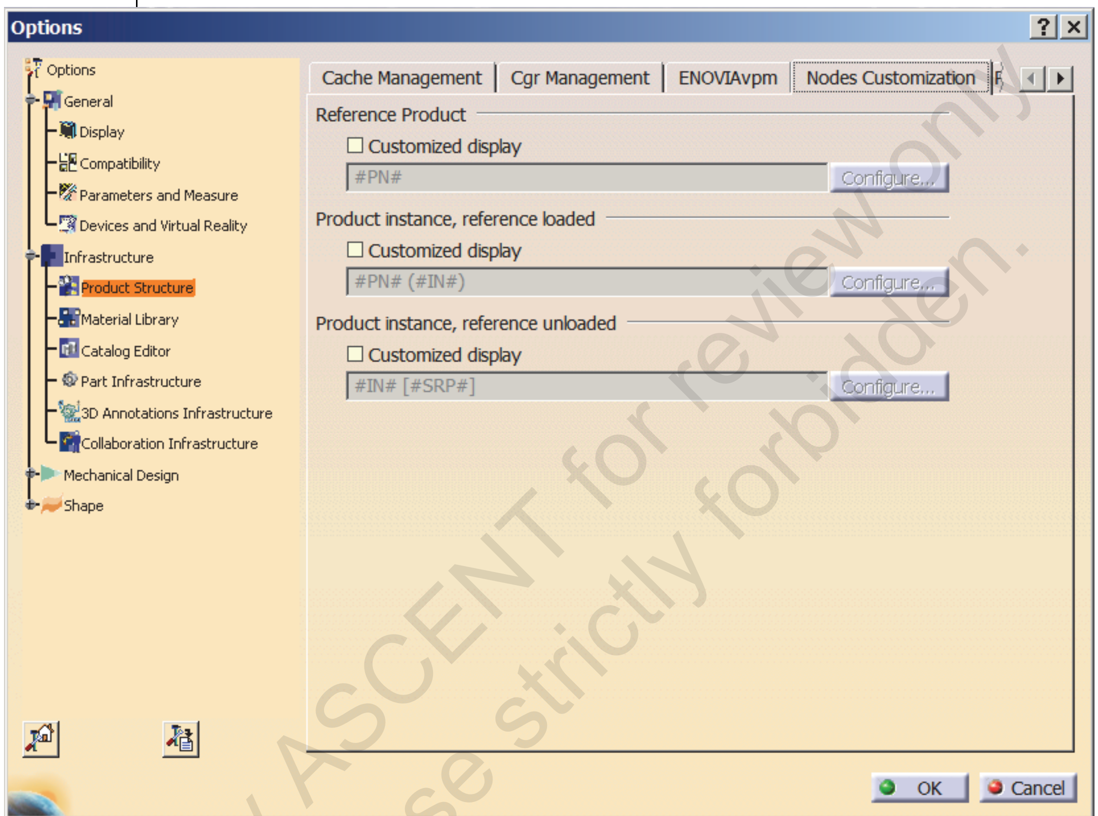


Figure 1–27

4. In the *Product instance, reference loaded* field, select **Customized display**, as shown in Figure 1–28.

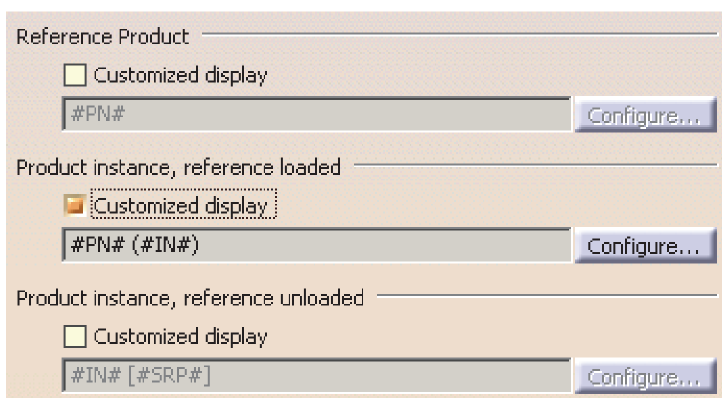


Figure 1–28

5. Click **Configure**. The Configure customized display dialog box opens, as shown in Figure 1–29.

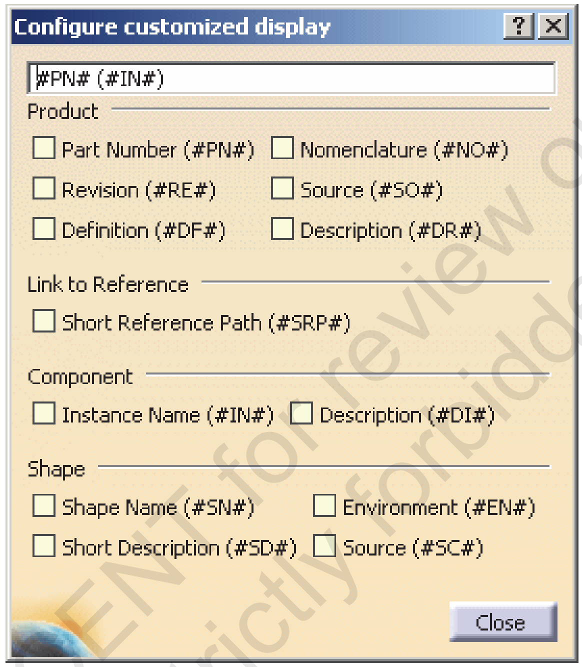


Figure 1–29

6. Clear the contents in the upper field, as shown in Figure 1–30.

Figure 1–30

7. Select the **Part Number (#PN#)**

8. Add a left side bracket and select **Short Description (#SD#)**.

The Short Description is the filename of the part. Products do not have short descriptions.

9. Add a right side bracket. The *Display* field displays, as shown in Figure 1–31.

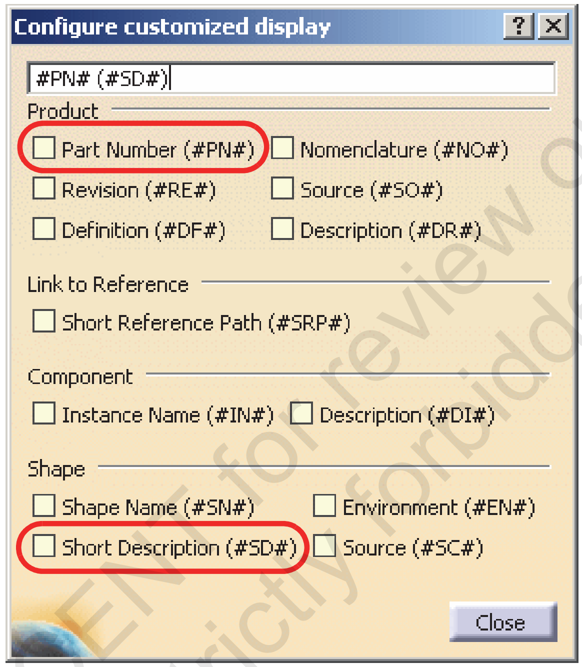


Figure 1–31

10. Click **Close** to close the Configure customized display dialog box.

11. Click **OK** to close the Options dialog box. The specification tree displays, as shown in Figure 1–32.

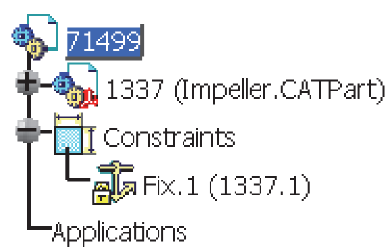


Figure 1–32

Task 4 - Assemble 6thStage.CATPart.

1. In the specification tree, right-click on **71499** and select **Components>Existing Component**.

2. Open **6thStage.CATPart**. The model displays in its default location on top of the impeller.

3. Use the compass to reposition **6thStage**. Right-click on the red box on the compass and select **Snap Automatically to Selected Object**.

4. In the specification tree, select **1296**. The compass snaps to the component.

5. Drag the compass to reposition **6thStage**, as shown in Figure 1–33.

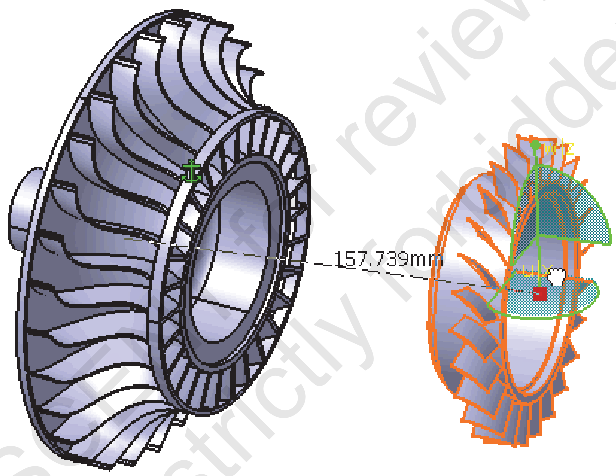


Figure 1–33

6. Drag the compass off **6thStage**.

7. Click  (Coincidence Constraint).

8. Select the implicit axis of **6thStage** and the **Impeller**, as shown in Figure 1–34.

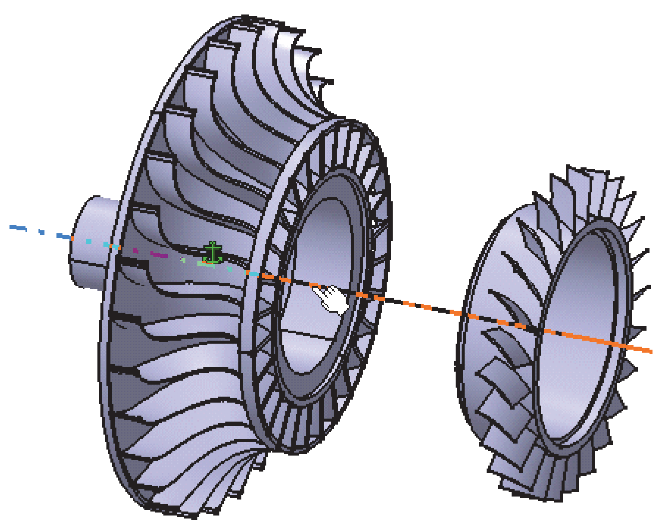
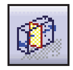


Figure 1–34

9. Click  (Contact Constraint).

10. Select the surface on **Impeller** shown in Figure 1–35.

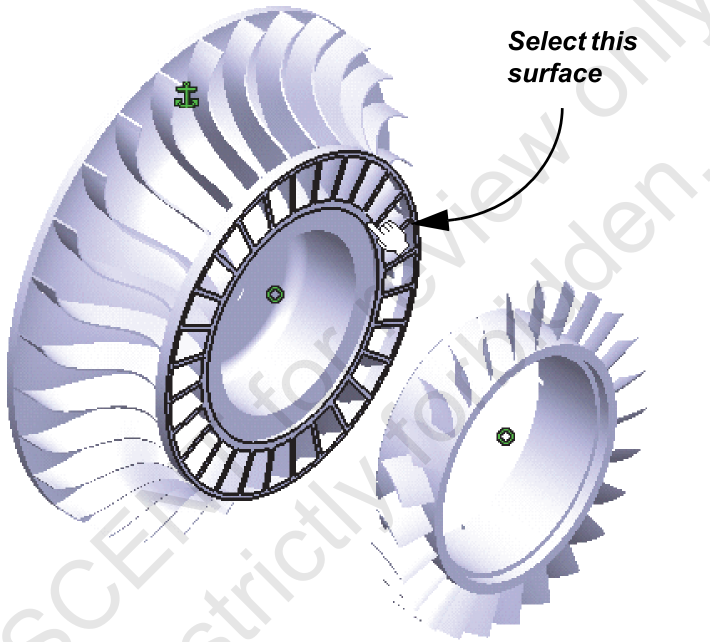


Figure 1–35

11. Reorient the model and select the surface on **6thStage**, as shown in Figure 1–36.

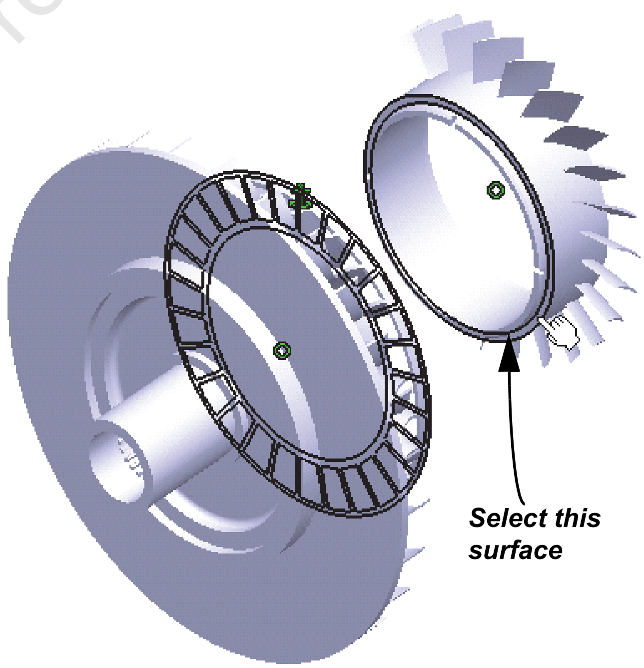
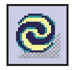


Figure 1–36

12. Click  (Update All) to update the assembly, as shown in Figure 1–37.

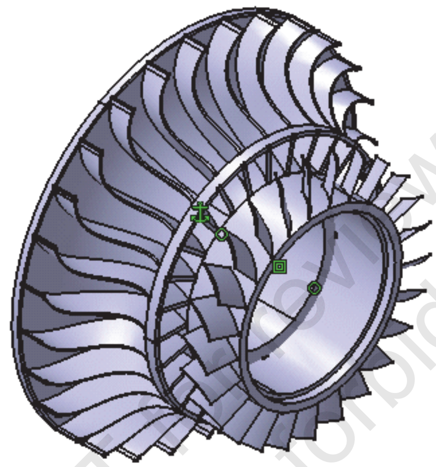


Figure 1–37

Task 5 - Assemble 5thStage.CATPart.

1. Insert the **5thStage.CATPart**.
2. Use the compass to reposition **5thStage**.
3. Create a Coincidence constraint between the implicit axis of **5thStage** and **6thStage**.
4. Create a Contact constraint between the two surfaces shown in Figure 1–38.

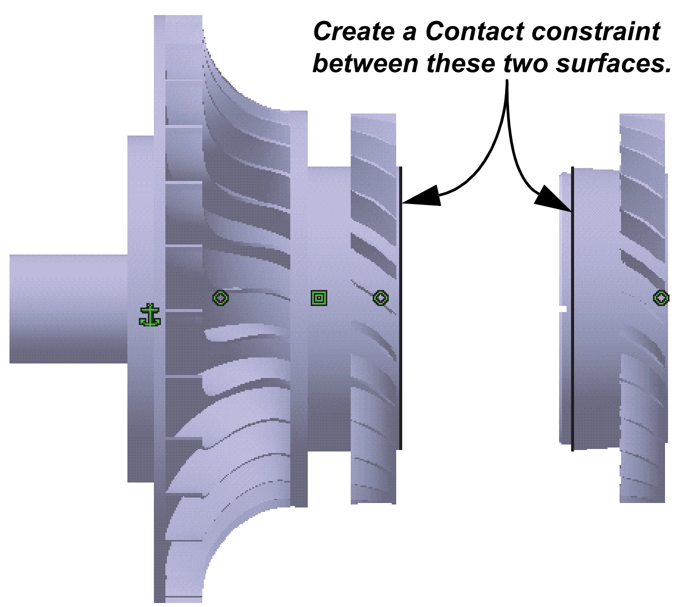


Figure 1–38

5. Add a third constraint to orient **5thStage**. Zoom in on the locating tab of **6thStage**, as shown in Figure 1–39.

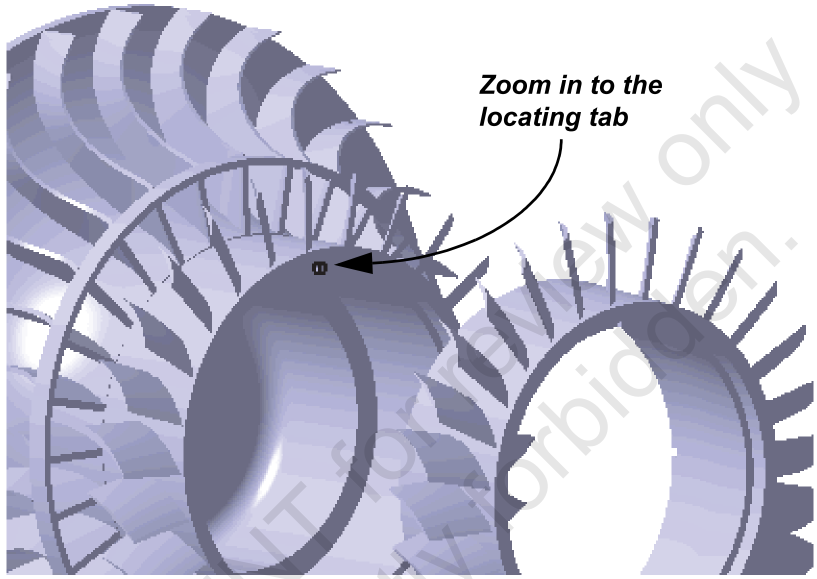


Figure 1–39

6. Click  (Angle Constraint) and create a Parallelism constraint between the surface of **6thStage** shown in Figure 1–40 and the surface of **5thStage** shown in Figure 1–41. Ensure that the *Orientation* is set to **Same**.

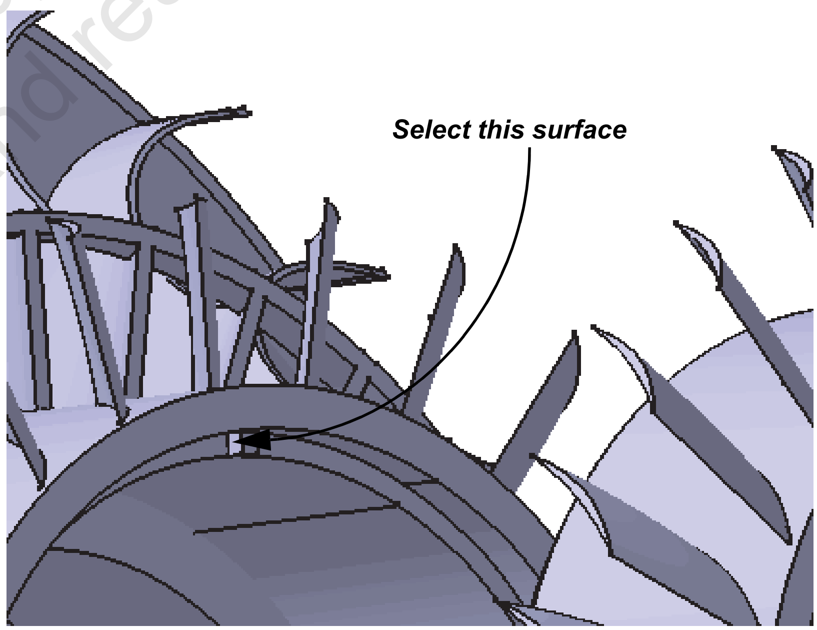


Figure 1–40

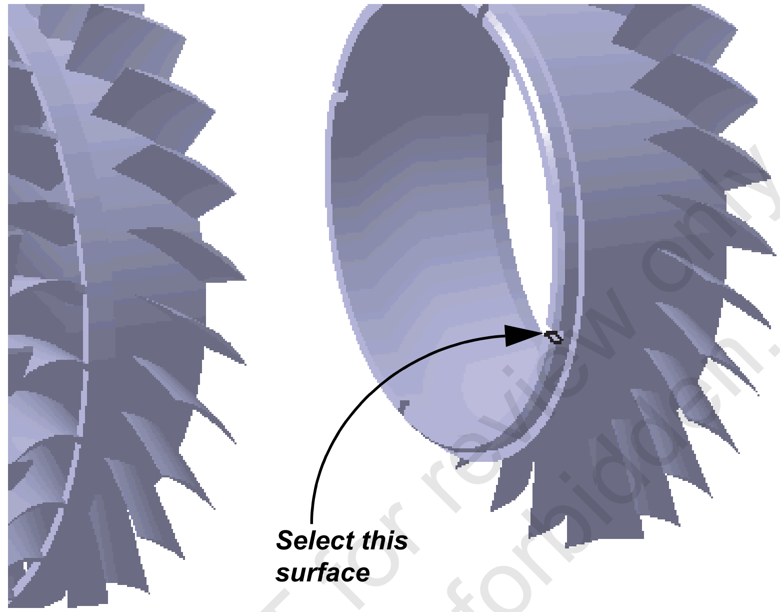


Figure 1–41

7. Click  (Update All).

8. Click  (Isometric View).

   The assembly displays, as shown in Figure 1–42.

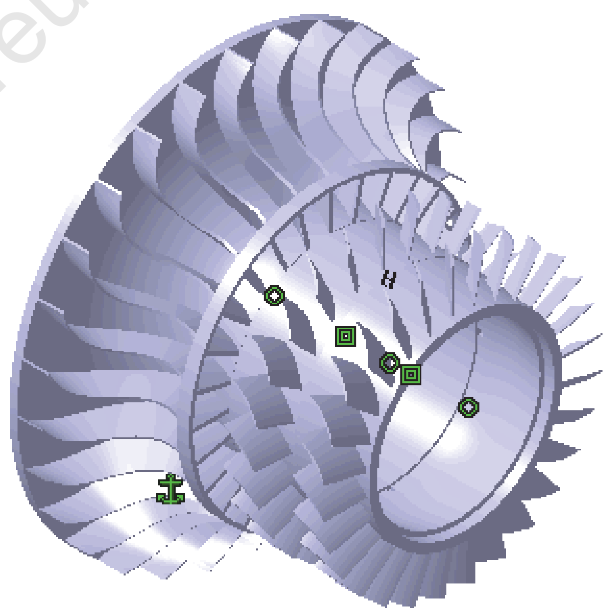


Figure 1–42

Task 6 - Assemble the remaining turbine wheel components.

1. Insert the remaining components of the assembly. When selecting the components to open, use <Ctrl> to select **4thStage**, **2nd3rdStage**, and **1stStage**.

2. Use the compass to drag each component out to new locations, as shown in Figure 1–43.

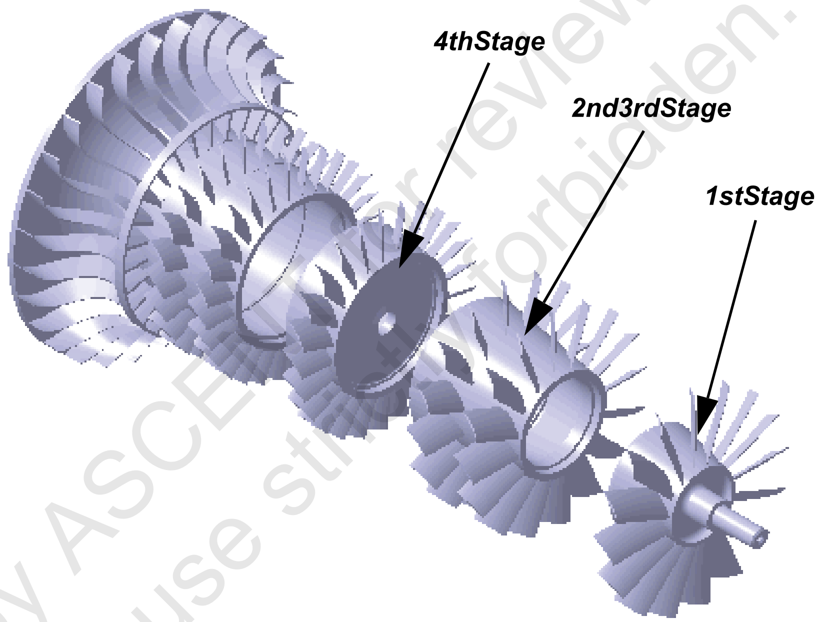
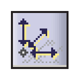


Figure 1–43

Task 7 - Change the constraint creation mode.

1. In the Constraint Creation toolbar, click  (Stack Mode).

2. Double-click on  (Coincidence Constraint).

3. Select the axis of **1337**, as shown in Figure 1–44.

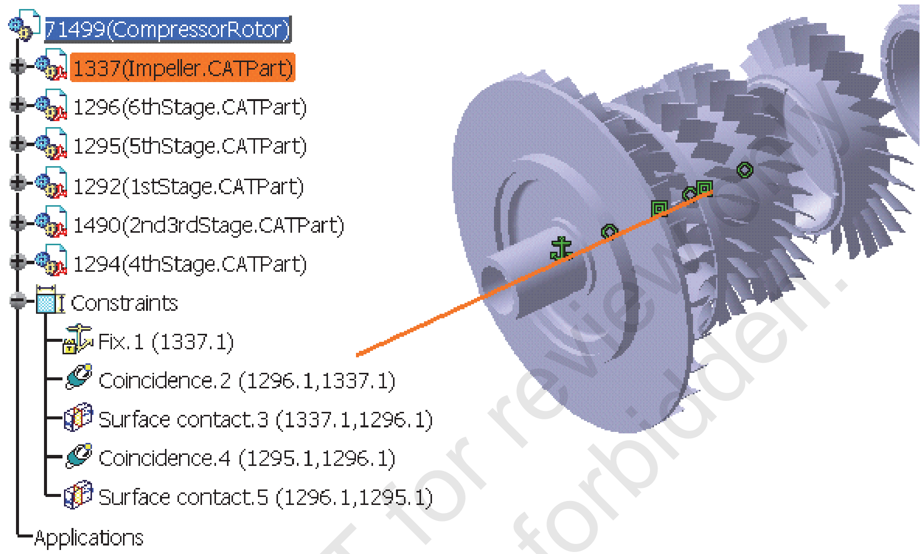


Figure 1–44

4. Select the axes of the three turbine wheels, as shown in Figure 1–45.

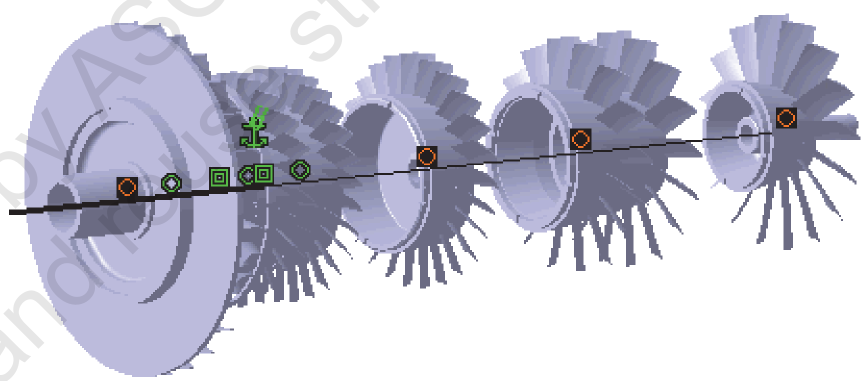
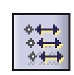


Figure 1–45

5. Set the constraint creation mode to  (Default).

6. Create three Parallelism constraints between the turbine wheels.

7. Create three Contact constraints between the turbine wheels.

8. Click  (Update All).

9. In the specification tree, right-click on the Constraints node and select **Hide/Show**. The updated model displays, as shown in Figure 1–46.

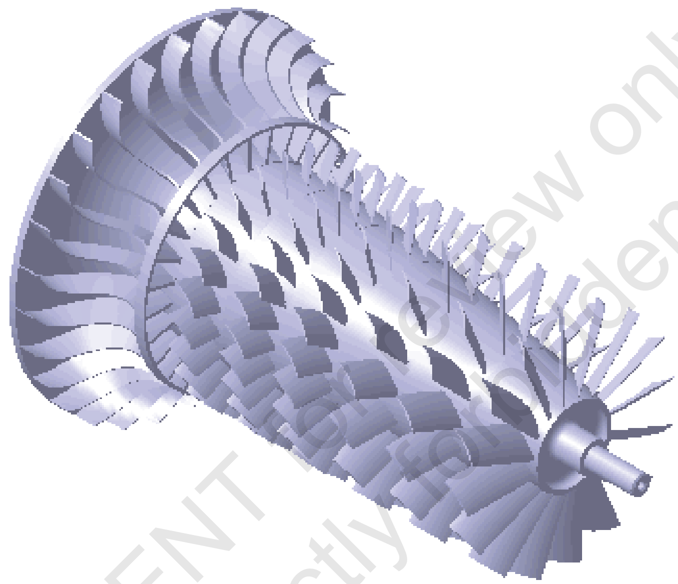


Figure 1–46

Task 8 - Assemble TieBolt.CATPart.

When selecting components to hide, select them in the specification tree, otherwise you might only hide the PartBody and not the complete instance of the part.

1. Hide all of the components except **Impeller** and **1stStage**, as shown in Figure 1–47.

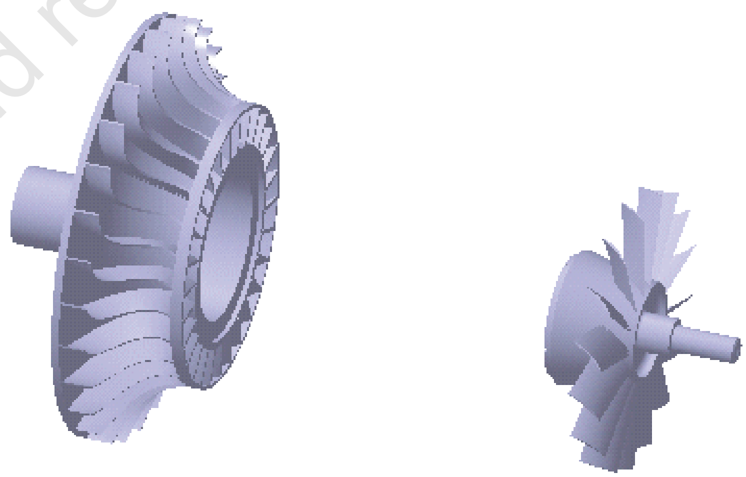


Figure 1–47

2. Insert **TieBolt.CATPart**.

3. Apply a Contact constraint to the surface of **TieBolt**, as shown in Figure 1–48 and to the back surface of **Impeller**, as shown in Figure 1–49.

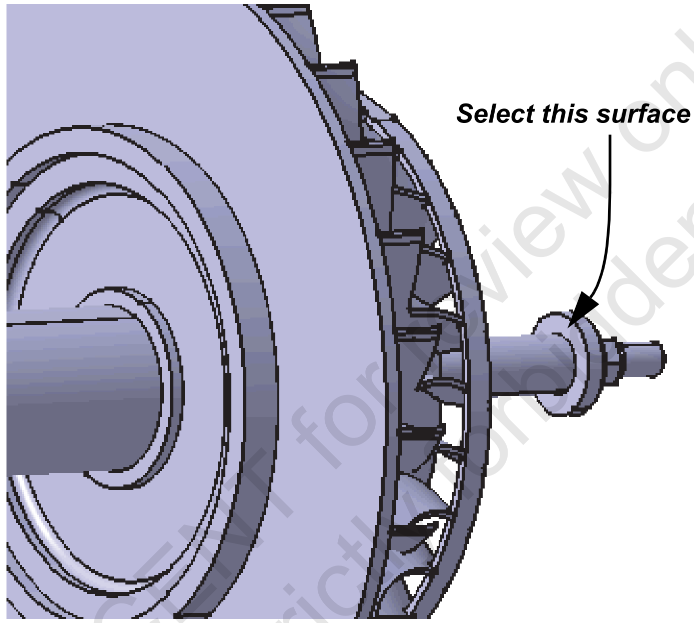


Figure 1–48

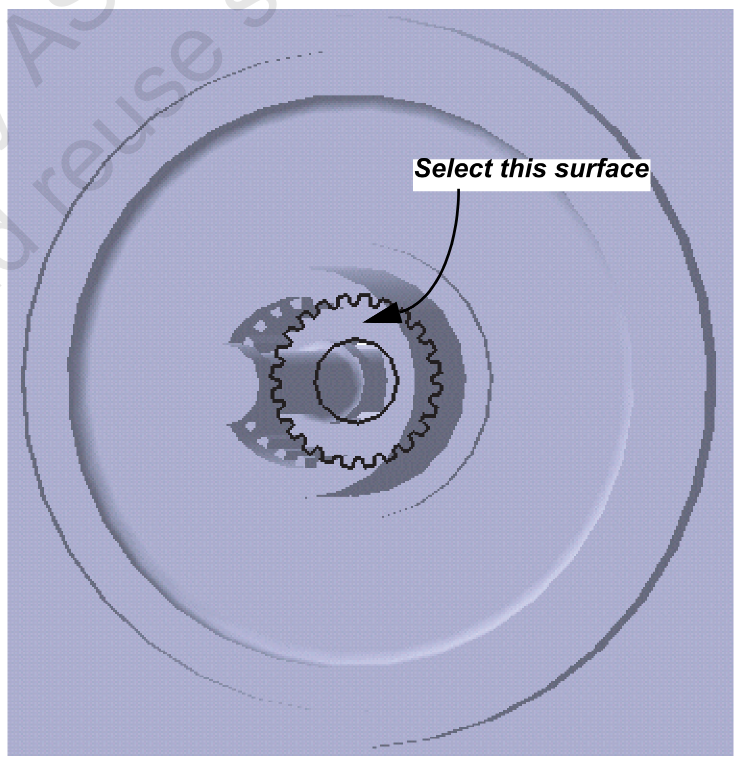


Figure 1–49

4. Constrain the axes of **TieBolt** and **Impeller** and update the assembly. The assembly displays, as shown in Figure 1–50.

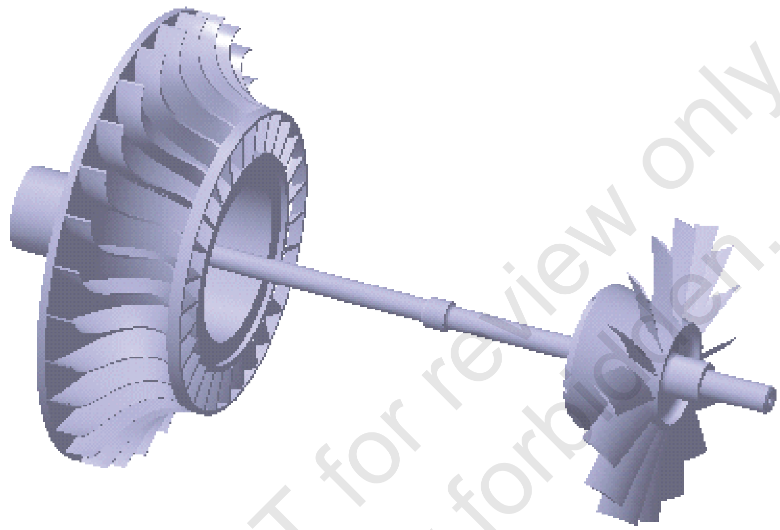


Figure 1–50

Task 9 - Change update options and assemble Coupling.CATPart.

1. Select **Tools>Options**. Expand Mechanical Design and select **Assembly Design**.

2. Select the *General* tab and select the **Automatic** option in the *Update* area, as shown in Figure 1–51.

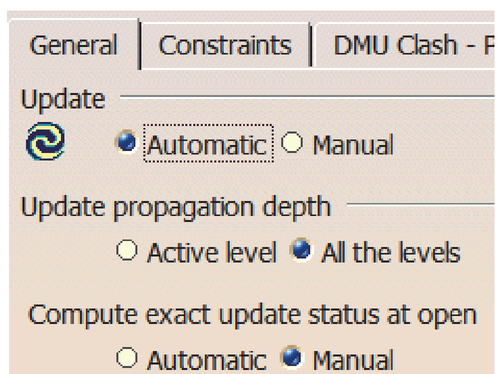


Figure 1–51

3. Click **OK** to close the Options dialog box.

4. Hide the **Impeller (1337)**.

5. Assemble **Coupling.CATPart** to the impeller end of the tiebolt, as shown in Figure 1–52.

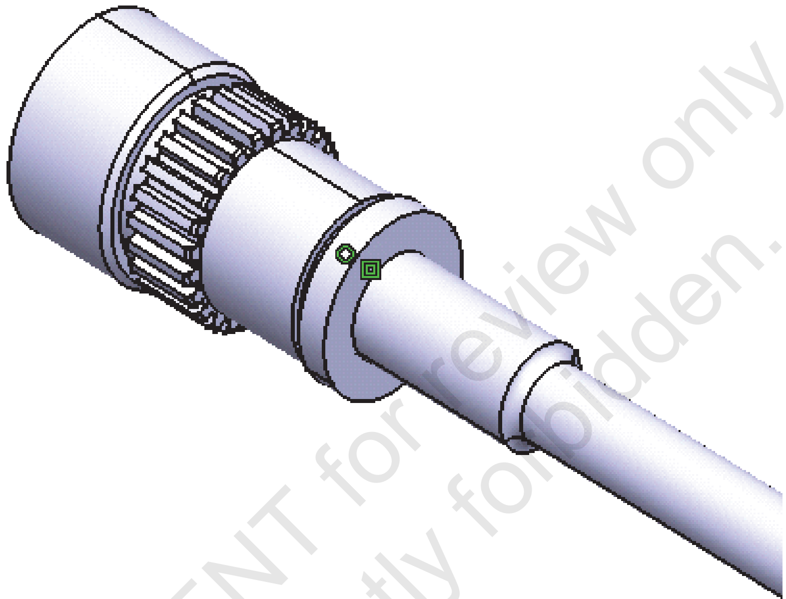


Figure 1–52

Note that the system automatically updates the constraints on creation.

Task 10 - Restore options.

1. Select **Tools>Options**. The Options dialog box opens, displaying the *General* tab in **Assembly Design**.

2. Click  (Reset parameters values to default ones). The Reset dialog box opens, as shown in Figure 1–53.

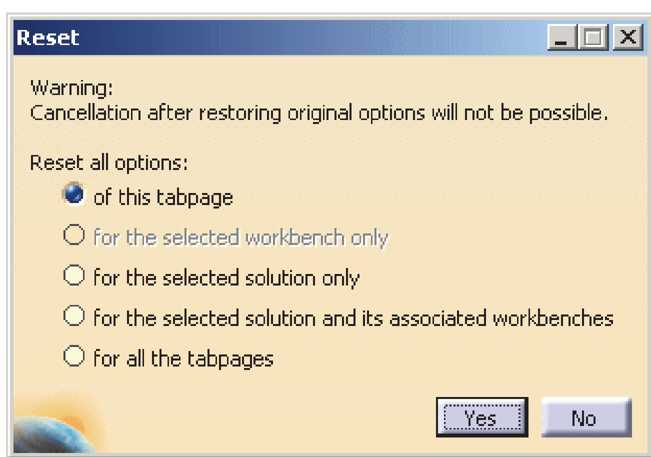


Figure 1–53

3. Click **Yes** to restore the options in the *General* tab.
4. Select **Infrastructure>Product Structure>***Nodes Customization* **tab** and repeat Steps 2 and 3 to restore the settings.
5. Click **OK** to close the Options dialog box.
6. Show all of the components,
7. Drag the compass away from the parts.
8. Select **View>Reset Compass**.
9. Right-click on the compass and clear the **Snap Automatically to Selected Object** option.
10. Save the model and close the file.