Pro/ENGINEER®
Wildfire™ 2.0

Pro/VERIFY™
Help Topic Collection

Parametric Technology Corporation

6,545,671 B1 08-April-2003	5,140,321 18-August-1992	5,557,176 09-November-1996
GB2354685B 18-June-2003	5,423,023 05-June-1990	5,561,747 01-October-1996

Third-Party Trademarks

Adobe is a registered trademark of Adobe Systems. Advanced ClusterProven, ClusterProven, and the ClusterProven design are trademarks or registered trademarks of International Business Machines Corporation in the United States and other countries and are used under license. IBM Corporation does not warrant and is not responsible for the operation of this software product. AIX is a registered trademark of IBM Corporation. Allegro, Cadence, and Concept are registered trademarks of Cadence Design Systems, Inc. Apple, Mac, Mac OS, and Panther are trademarks or registered trademarks of Apple Computer, Inc. AutoCAD and Autodesk Inventor are registered trademarks of Autodesk, Inc. Baan is a registered trademark of Baan Company. CADAM and CATIA are registered trademarks of Dassault Systemes. COACH is a trademark of CADTRAIN, Inc. DOORS is a registered trademark of Telelogic AB. FLEX*lm* is a trademark of Macrovision Corporation. Geomagic is a registered trademark of Raindrop Geomagic, Inc. EVERSYNC, GROOVE, GROOVEFEST, GROOVE.NET, GROOVE NETWORKS, iGROOVE, PEERWARE, and the interlocking circles logo are trademarks of Groove Networks, Inc. Helix is a trademark of Microcadam, Inc. HOOPS is a trademark of Tech Soft America, Inc. HP-UX is a registered trademark and Tru64 is a trademark of the Hewlett-Packard Company. I-DEAS, Metaphase, Parasolid, SHERPA, Solid Edge, and Unigraphics are trademarks or registered trademarks of Electronic Data Systems Corporation (EDS). InstallShield is a registered trademark and service mark of InstallShield Software Corporation in the United States and/or other countries. Intel is a registered trademark of Intel Corporation. IRIX is a registered trademark of Silicon Graphics, Inc. LINUX is a registered trademark of Linus Torvalds. MatrixOne is a trademark of MatrixOne, Inc. Mentor Graphics and Board Station are registered trademarks and 3D Design, AMPLE, and Design Manager are trademarks of Mentor Graphics Corporation. MEDUSA and STHENO are trademarks of CAD Schroer GmbH. Microsoft, Microsoft Project, Windows, the Windows logo, Windows NT, Visual Basic, and the Visual Basic logo are registered trademarks of Microsoft Corporation in the United States and/or other countries. Netscape and the Netscape N and Ship's Wheel logos are registered trademarks of Netscape Communications Corporation in the U.S. and other countries. Oracle is a registered trademark of Oracle Corporation. OrbixWeb is a registered trademark of IONA Technologies PLC. PDGS is a registered trademark of Ford Motor Company. RAND is a trademark of RAND Worldwide. Rational Rose is a registered trademark of Rational Software Corporation. RetrievalWare is a registered trademark of Convera Corporation. RosettaNet is a trademark and Partner Interface Process and PIP are registered trademarks of “RosettaNet,” a nonprofit organization. SAP and R/3 are registered trademarks of SAP AG Germany. SolidWorks is a registered trademark of SolidWorks Corporation. All SPARC trademarks are used under license and are trademarks or registered trademarks of SPARC International, Inc. in the United States and in other countries. Products bearing SPARC trademarks are based upon an architecture developed by Sun Microsystems, Inc. Sun, Sun Microsystems, the Sun logo, Solaris, UltraSPARC, Java and all Java based marks, and “The Network is the Computer” are trademarks or registered trademarks of Sun Microsystems, Inc. in the United States and in other countries. TIBCO, TIBCO Software, TIBCO ActiveEnterprise, TIBCO Designer, TIBCO Enterprise for JMS, TIBCO Rendezvous, TIBCO Turbo XML, TIBCO Business Works are the trademarks or registered trademarks of TIBCO Software Inc. in the United States and other countries. WebEx is a trademark of WebEx Communications, Inc.

Third-Party Technology Information

Certain PTC software products contain licensed third-party technology: Rational Rose 2000E is copyrighted software of Rational Software Corporation. RetrievalWare is copyrighted software of Convera Corporation. VisTools library is copyrighted software of Visual Kinematics, Inc. (VKI) containing confidential trade secret information belonging to VKI. HOOPS graphics system is a proprietary software product of, and is copyrighted by, Tech Soft America, Inc. G-POST is copyrighted software and a registered trademark of Intercim. VERICUT is copyrighted software and a registered trademark of CGTech. Pro/PLASTIC ADVISOR is powered by Moldflow technology. Moldflow is a registered trademark of Moldflow Corporation. The JPEG image output in the Pro/Web.Publish module is based in part on the work of the independent JPEG Group. DFORMD.DLL is copyrighted software from Compaq Computer Corporation and may not be distributed. METIS, developed by George Karypis and Vipin Kumar at the University of Minnesota, can be researched at http://www.cs.umn.edu/~karypis/metis. METIS is © 1997 Regents of the University of Minnesota. LightWork Libraries are copyrighted by LightWork Design 1990–2001. Visual Basic for Applications and Internet Explorer is copyrighted software of Microsoft Corporation. Parasolid © Electronic Data

Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494 USA

Table Of Contents

Using Additional Modules

Computer-Aided Verification Option

Pro/VERIFY

Basics of Pro/VERIFY

About Pro/VERIFY

Pro/VERIFY is an optional Pro/ENGINEER module that you can use to classify and quantify flaws generated in manufactured models when they are compared to design specifications.

Typically, you use a laser scanner to scan a manufactured model. The scanner outputs a file containing a cloud of point representations of the model—usually several hundred thousand in one scan. These point representations are generally spaced very closely to detect and measure surface geometry features that are approximately 15/1000 of an inch. Many laser scanners can achieve accuracy to approximately 25 microns.

Pro/ENGINEER reads in the scan data file and then compares the sampled model to the (nominal) design model, using computer-aided verification techniques. You can quantify the following types of deviations:

- Global defects, such as warping and shrinkage
- Mid-level defects, such as those caused by improper EDM setups
- Highly local defects, caused by surface imperfections

Scan data sets can be in any of the following formats:

- ASCII (`filename.pts`)
- IBL (`filename.ibl`)
- IGES (`filename.igs`)
- VDA (`filename.vda`)

To maintain compatibility with Pro/ENGINEER formats, Pro/VERIFY first creates an assembly. However, Pro/ENGINEER can distinguish Verify assemblies from Pro/ENGINEER assemblies.

The Verify assembly has the following characteristics:

- The base component is the design model.
- Each scaled and placed scan set is treated as a component part. When you save the assembly, its scan sets are also saved as separate parts.

Each fit and deviation case is treated as a feature of the assembly.

To Use Pro/VERIFY

Use Pro/VERIFY with an assembly.

1. Read in a set of sampled point data (often described as a point cloud) from the manufactured model, scale it, and roughly place it on the design model. Pro/ENGINEER regards the scaled and placed data as a scan set and considers it to be a component part of the Verify assembly.

   Repeat as required with other sets of sampled data.

2. Use built-in tools to filter the scan sets.

3. Specify one or more precise fits of the data sets to the design model. Then, based on those precise fits, specify the deviation cases that you want analyzed.

4. Calculate the deviation cases.

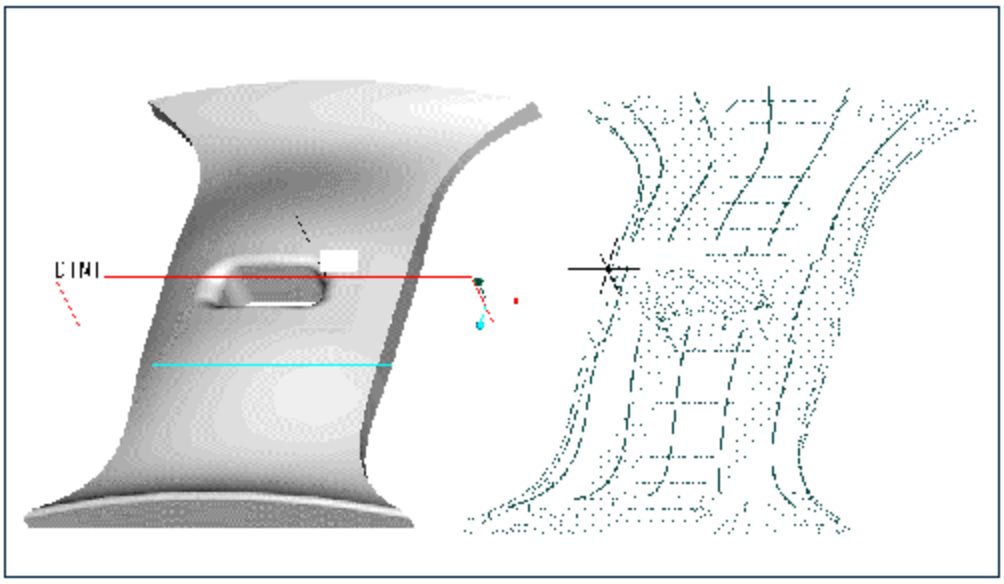
5. Display graphically, plot, tabulate, or export the results.

Example: Typical Scan File Extract

Extract from a Typical Scan File

```
-1.3728 -.8386 .2323
-1.3728 -.8446 .2318
-1.3728 -.8506 .2312
-1.3728 -.8556 .2313
 1.3728  .8626 .2309
-1.3728 -.8686 .2302
-1.3728 -.8745 .2302
-1.3728 -.8806 .2298
-1.3728 -1.0563 .2165
-1.3668 -1.0020 .2311
-1.3668 -.9950 .2315
-1.3668 -.9900 .2322
-1.3668 -.9840 .2322
-1.3668 -.8581 .2393
```

Design Model and Point Cloud before Assembly



Working with Verification Assemblies

About Creating the Verification Assembly

A verification assembly consists of the design model and one or more sets of scan data. The design model is the base component.

Each scan set is regarded as an additional component part. You place each scan set on the design model by using procedures and constraints similar to those used in assembling physical components.

To Create a Verification Assembly

1. Click **File > New**. The **New** dialog box opens.
2. In **Type**, select **Assembly** and in **Sub-type**, select **Verify**.
3. Enter the file name or use the existing file name format (`asm0001`) and click **OK**. The **VERIFICATION** menu opens.
4. Click **VERIFICATION > Design Mdl**. The **DESIGN MDL** menu opens with **Assemble** preselected and the **Open** dialog box opens.
5. Select a design model to retrieve. Click the name of the file type (**Assembly** or **Part**) to filter the files and then select the path to the directory where the file is stored.

After Pro/ENGINEER retrieves the design model, the base Verify assembly is complete.

Verification Menu Commands

- **Design Mdl**—Selects a base component (a design model)
- **Scan Set**—Creates, redefines, and deletes scan set components
- **Fit Scan**—Set ups, redefines, and deletes scan fitting cases
- **Deviation**—Creates, redefines, and deletes deviation cases
- **Results**—Computes and displays verification results

To Save a Verify Assembly

After you have placed a scan set on the assembly, you can save a Verify assembly.

1. Click **File > Save**. The assembly file is saved as an `.asm` file, and the scan set file is saved automatically in a separate file as a `.prt` file.

To Retrieve an Existing Verify Assembly

You can retrieve a Verify assembly after you have placed a scan set on the assembly and saved the file.

1. Click **File > Open**. The **Open** dialog box opens.
2. Select a file of **Type Assembly** (`asm0001.asm`) with a **Sub-type** of **Verify** from the directory path. The Verify assembly opens. The **VERIFICATION** menu also opens.

Working with Scan Sets

About Scan Sets

Typically, you use a laser scanner to scan a manufactured model. The scanner outputs a file containing a cloud of point representations of the model—usually several hundred thousand in one scan. These point representations are generally spaced very closely to detect and measure surface geometry features that are approximately 15/1000 of an inch. Many laser scanners can achieve accuracy to approximately 25 microns.

Pro/ENGINEER reads in the scan data file and then compares the sampled model to the (nominal) design model, using computer-aided verification techniques. Scan sets of data can be in any of the following formats:

- ASCII (`filename.pts`)
- IBL (`filename.ibl`)
- IGES (`filename.igs`)
- VDA (`filename.vda`)

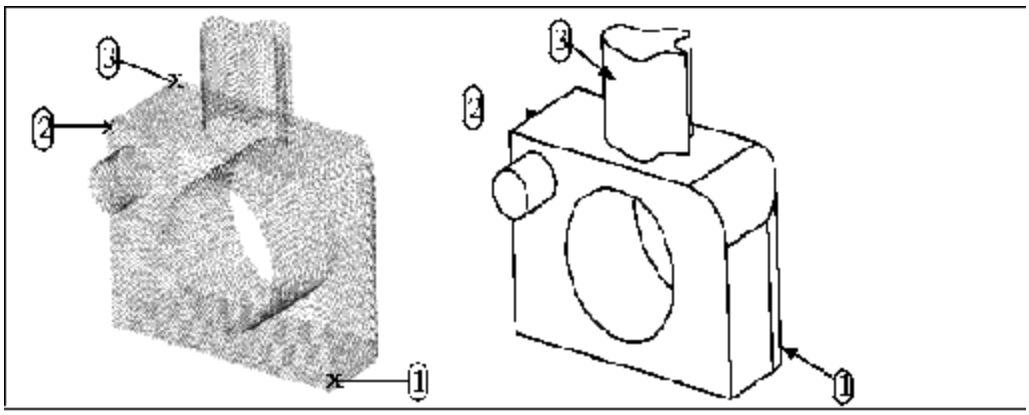
To Create a Scan Set

You can create a scan set from a point cloud and assemble it to a design model.

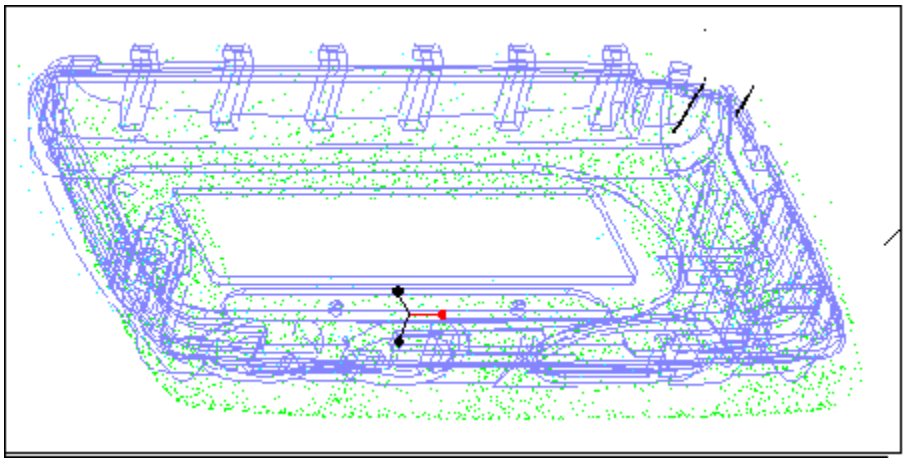
1. Click **VERIFICATION > Scan Set** to open the **SCAN SET** menu.
2. Click **Create** and type a part name for the scan set. The **Scan Set Create** dialog box opens.
3. Click the Point Set open folder icon to open a point cloud data set. The **Open** dialog box opens. The **Type** filter is set to display the following file types:
    - All files (`*`)
    - IGES—(`.igs` extension)
    - CAV lbl file—(`.lbl` extension)
    - CAV Pts file—(`.pts` extension)
    - VDA file—(`.vda` extension)
4. Select one of the point cloud data sets from these file types, for example, a `.pts` file. Click **OK**.
5. Under **Units**, select a unit of measurement for the scan set. In the option **Other Unit**, you can define and name your own unit.
6. Click **Add** to define an entity (one point defining the center of a sphere in a scan set) in the **Gauge ball centers** option.
7. Select a data point on the displayed scan set.
8. Click **OK**. The name of the entity is listed in the **Gauge ball centers** window. By clicking the name, you can highlight the entity on scan set.
9. You must define at least three entities to define the center of a sphere on the scan set.
10. If it is necessary to reposition the scan set relative to the model, click **Position Scan Set > Position** to open the **Component Placement** dialog box. Use the options on this dialog box to reposition the scan set.
11. Click **OK**.

Example: Create a Scan Set

Point `n` on the scan set is matched with point `n` on the design model.



The scan set can also appear with the assembly as shown next.



Scan Set Menu Commands

The **Scan Set** menu has the following commands:

- **Create**—Creates a scan set component from a point data file
- **Delete**—Removes a scan set component from the verification assembly
- **Redefine**—Respecifies elements that define a scan set
- **Split**—Splits one scan set into two
- **Merge**—Merges two scan sets into one
- **Suppress**—Suppresses scan set or sets
- **Resume**—Resumes working with the scan set or sets
- **Delete Pnts**—Permanently removes points from a scan set

Display Options with Scan Sets

The **Component Placement** dialog box contains the following buttons:

- **Separate Window**—Displays the scan set in a separate component window
- **Assembly**—Displays the scan set in the main Pro/ENGINEER window with the assembly

To Place the Scan Set on the Design Model

1. Select a **Constraint Type** from the **Component Placement** dialog box:
    - **Align**—Selects a data point from the scan set for alignment to a design model point
    - **Coord Sys**—Selects a coordinate system from one model of the scan set of the design model to be aligned
    - **Pnt On Srf**—Selects a data point from the scan set to constrain to a design model surface
    - **Default**—Selects a fully constrained placement status
2. For the first constraint, click **Align** and then click a command on the **GET SELECT** menu (for example, **Pick**) to select a data point from the scan set for alignment to a design model point.
3. Click **CONFIRM > Accept** when the design model point turns red. This command is used only with **Query Sel**.
4. With the **GET SELECT** menu, select an aligning point or vertex on the design model. A red cross appears on the location.
5. For the second constraint, again select a data point from the scan set. The Scan Set Point appears in the message area, for example:

    ```
    X= 0.8148, Y= 2.0909, Z=0.3269 in model MF_PART
    ```

5. Click **CONFIRM > Accept** or **Next**.
6. For the third restraint, click **Align** or **Pnt On Srf**.
7. To preview the placement, click **Preview**. You can change the placement later.

To Change Scan Set Placement

1. Click **SCAN SET > Redefine.** the **SEL SCAN** menu.appears.
2. Select the scan set.
3. Click the **Position** box and click **Done**. The **Component Placement** dialog box opens.

To Create a Scan Set Output File

1. Click **FIT SCAN > Out Trf Pts**. The **Output Transformed Points** dialog box opens.
2. Click **OK**. The **GET COORD S** menu and the **GET SELECT** menu open so you can create or select a coordinate system.
3. Choose one of the commands from the **GET COORD S** menu. After you select a command, the transformed points are saved to a Pts file.
    - **Create**—Creates a coordinate system with input from the **OPTIONS** menu
    - **3 Planes**—Creates a coordinate system that passes through 3 planes
    - **Pnt + 2Axes**—Creates a coordinate system when you specify the origin and 2 axes
    - **2 Axes**—Creates a coordinate system when you specify 2 axes
    - **Offset**—Creates a coordinate system offset from another coordinate system
    - **Offs By View**—Creates a coordinate system parallel to the screen
    - **Pln + 2Axes**—Creates a coordinate system when you specify 1plane and 2 axes
    - **Orig + ZAxis**—Creates a coordinate system when you specify the origin and z-axis
    - **From File**—Creates a coordinate system by reading data from a file
    - **Default**—Displays the model's default coordinate system
    - **Done**—Continues with the current process
    - **Quit**—Quits the current process
    - **Select**—Selects a coordinate system from the current display with input from the **GET SELECT** menu
    - **Default**—Gets the default model coordinate system

To Merge the Spherical Centers of Scan Sets

1. Click **VERIFICATION > Scan Set > Create**.
2. Give the scan set a name, and the **Scan Set Create** dialog box opens.
3. Click the **Point Set** open folder icon to open the **File/Open** browser.
4. Select a point cloud from the browser and click **OK**.
5. Under **Units**, select a unit of measurement for the scan set. In the option **Other Unit**, you can define and name your own unit.

6. Click **Add** to define an entity (one point defining the center of a sphere in a scan set) in the **Gauge ball centers** option.
7. Select a data point on the displayed scan set.
8. Click **OK**. The name of the entity is listed in the **Gauge ball centers** window. By clicking the name, you can highlight the entity on scan set.

   You must define at least three entities to define the center of a sphere on the scan set.
9. If it is necessary to reposition the scan set relative to the model, click **Position Scan Set > Position** to open the **Component Placement** dialog box. Use the options on this dialog box to reposition the scan set.
10. Repeat steps 3 through 8 to define a second spherical center of a second scan set.
11. Merge scan sets from the **SCAN SET > Merge** menu.

Merging Scan Sets

If you create more than one point cloud to describe a single part, you would then want to merge the multiple point clouds together into a single scan set to describe your model. The multiple point clouds are created relative to spheres on the fixture on which the part is mounted. The spheres are used as reference points to merge the point clouds together into a single scan set.

To Filter Display of Scan Sets

1. Click **View > Display Settings> Scan Display** to open the **Scan Set Display** dialog box.
2. Select a scan set from the list of scan sets. Deselect it by clicking on it again. You can change a number of scan sets simultaneously by selecting them together.
3. Choose one of the **Display Mode** options:
   - **All Points**—Displays the whole scan set (subject to the density setting).
   - **Pick Box**—Creates a box after you pick two points to form the diagonal corners of a box. Only the points inside the box are displayed. You must pick the box immediately after you choose this mode.
   - **Cross Section**—Displays the points in a specified envelope about a selected planar cross section in the design model and provides a list of available cross sections. You must enter the envelope width in the **Xsec Width** box.
4. To set the density of point display, click on the **Percentage** window, and then select one of the fixed values (from 0 through 100%) from the pull-down menu.
5. Click **Preview** for a preliminary view of the scan displays.

6. Click **Cancel** to close the dialog box without making any changes. Click **OK** to keep the changes and exit the dialog box.

Note: These display settings are not saved. Therefore, if you change them, neither the Verification assembly nor its components are marked as changed.

Filtering Scan Sets

A typical scan set may contain several 100,000 points. Repainting the display of all these points can take some time, especially if you are displaying a number of scan sets. You may often be able to get the information you need from a smaller number of uniformly distributed points. In addition, you may be interested in data from only a certain section of the scan set.

You can filter the display of scan sets with the **Scan Set Display** dialog box. The dialog box lists the active scan sets in the Verification assembly and their current display settings. By default, the scan set display settings are set to **All Points** and a density of either 10% or 5,000 points, whichever is fewer.

If the density is 5,000 points, then the scan set display reflects that value when the scan set is created or brought into session. Any changes you subsequently make to the settings are effective only as long as the scan sets are in session.

Displaying and Blanking the Design Model or Scan Set

You can selectively display or blank the design model or any of the scan sets in a Verification assembly by placing them on layers. The usual rules for displaying or blanking layers in assemblies then apply.

To Delete Points from a Scan Set

1. Choose **Delete Pnts** from the **SCAN SET** menu. The **SEL SCAN** menu appears, listing all the active scan sets in the assembly. If you place the pointer on one of the listed scan sets, a bounding box of the same color appears on the scan set in the assembly.
2. Select one of the scan sets. The **FILTER PnTS** menu appears with the following options:

- **Reduce**—Reduces the number of points in the scan set according to defined criteria
- **Remove—**Permanently removes points from the scan set according to defined criteria
- **Whole Set—**Applies the point reduction to the whole set of scan points
- **Pick Box—**Creates a box after you pick two points on the scan set to act as diagonal corners of a rectangular box. The points contained in the box are deleted
- **Near Plane**—Do one of the following:
    - Select an assembly datum plane.

    - Pick three points to define a plane. The plane is then displayed on the assembly**.**
    - Enter a tolerance around the plane. Points in the resultant 3D space are deleted**.**

- **Half Space**—Select an assembly datum plane or pick three points to define a plane. The plane is displayed on the assembly. Enter a tolerance around the plane. An arrow appears on the assembly, normal to the plane. The **DEL DIR** menu also appears. Click **Flip** to flip the arrow. Click **OK** to accept the direction shown. The system then removes all the points on the arrow side of the plane plus all the points that are within the specified tolerance on the other side of the plane.
    - **Done**—Deletes the selected points and return to the **SCAN SET** menu.
    - **Quit**—Ignores the selections made and returns to the **SCAN SET** menu.

3. Use the menu commands, in any order, to delete as many sets of points as you want. As you define each set, the system highlights it in red.
4. When you have finished, click **Done** to return to the **SCAN SET** menu. The system then removes the points in the selected areas.

Note: You can also use pick boxes to exclude sets of points from the fit features. You can also export a transformed scan set to a file and create the coordinates that you want. You can use this file within Pro/ENGINEER or export it to third-party software for further processing.

Deleting Points from a Scan Set

You can permanently remove data points from an existing scan set (but not from the original scan file) by clicking **Delete Pnts** on the **SCAN SET** menu. For example, you can use this method to exclude anomalous data points from the fixture that held the manufactured model in the scanner.

To Preview the Scan Display

1. Click **View > Display Settings > Scan Display**. The **Scan Set Display** dialog box opens.
2. Select a display in the window.
3. Click **Preview** to obtain a preview of the scan set display and the assembly.

By default, the system displays either 5,000 points or 5% density in the scan set (whichever is fewer).

Fit and Analysis

About Fit and Analysis

Before Pro/VERIFY can make meaningful calculations on the accuracy of the manufactured model, you must precisely fit the scan sets to the design model.

To Create a Fit

You can create a new fit feature with the **Create Fit** command on the **FIT SCAN** menu and reference a previous fit to create your new one.

1. Click **FIT SCAN > Create Fit**. The **SCAN SET FITTING** dialog box opens.
2. Enter the name for the fit feature, for example, `Fit3` and click the green arrow in the message area. The **SEL SCAN** menu opens with a list of scan sets that you can use for the current fit.
3. Click one of the menu commands, for example, **PRT0001_.** The dialog box updates with the new information.
4. Click **OK**. The following system messages appear, for example.
    - `PRT0001_Scan Set is displayed based on Fit3.`
    - `Fit3 Fit feature has been created successfully.`
5. Click one of the elements in the list, then click **Define**.

Note: You do not have to select fit surfaces on the design model to which the scan set is being fitted.

Fit Scan Menu Commands

The **Fit Scan** menu has the following commands:

- **Create Fit**—Defines elements of a fit feature with the **SCAN SET FITTING** dialog box
- **Redefine Fit**—Redefines elements of a fit feature with the **SCAN SET FITTING** dialog box
- **Delete Fit**—Removes a fit feature from the Verification assembly.
- **Update Fit**—Recalculates the fit of a scan set relative to a design model
- **Apply Fit**—Adjusts the scan set placement, according to fit
- **Out Trf Points**—Puts transformed points into a file

To Display a Fit

1. Click **View > Display Settings> Scan Display**. The **Scan Set Display** dialog box opens with a list of all fits that have been defined and applied to the displayed scan set.

2. Select any defined fit from the **Display Fit** list in the dialog box.

To Measure between a Scan Point and Other Geometry

You can select a scan point and measure the distance from it to the surface of a model or to another type of geometry. The measurements always reflect the currently displayed state of the scan set.

1. Create a scan set part, and then define and compute a fit for it.
2. Click **Analysis > Measure** to open the **Measure** dialog box.
3. Select **Distance** from the list in the **Type** box.
4. Select **Scan Point** from the list in **Definition From**.
5. Click a scan point from the displayed scan set. Its xyz coordinates appear in the **Measure** dialog box.
6. Select the type of geometry you want to measure to.
7. Click on the geometry in the displayed part These coordinates appear in the **Measure** dialog box.
8. Click Compute.

Having the Display Fit functionality on the **Scan Set Display** dialog box allows you to display various fits by choosing from a list of all scan fits that have been applied to the scan set.

To Fit a Scan Set to the Design Model

1. Specify the scan set you want to fit. Each fit specification becomes a feature of the Verify assembly. Deviation calculations are based on specified fits.
2. Specify the surfaces on the design model that you want to designate as primary fit surfaces.
3. Optionally, specify the surfaces on the design model that you want to designate as secondary fit surfaces.
4. Optionally, specify the surfaces on the design model that you want to designate as tertiary fit surfaces.
5. Optionally, draw pick boxes around sets of points that you want to exclude from the fit (for example, you can use this method to exclude anomalous data points from the fixture that held the manufactured part in the scanner).

Tip: Fit Scan Set to Design Model

Select as few surfaces as is reasonable for fitting. Too many fit surfaces can cause overfitting, which can decrease the value of deviation results.

Note: For the best results, select surfaces that will fully constrain all six degrees of freedom.

The system fits the scan set to all the selected surfaces simultaneously. It gives the greatest weight to the primary surfaces and the least weight to the tertiary surfaces. This functionality is very useful in obtaining deviation results about features that have been dimensioned with primary, secondary, and tertiary geometric tolerances.

Imagine a hole in the top of a cube, which is located by linearly dimensioning to the right plane (D1), the front plane (D2), and with a depth dimensioned from the top plane (D3). If geometric tolerances are assigned D1, D2, and D3 as primary, secondary, and tertiary respectively, these must be kept in mind when fitting the scan set. The fit should reference the right side of the cube as a primary surface, the front side as a secondary surface, and the top as a tertiary surface. This yields the most accurate information about deviations of the hole feature.

In situations where another feature—which was dimensioned using different references—is on top of the box, you can create another fit feature using the appropriate surfaces as references.

After you have selected surfaces for a fit feature, you can designate points that you want the system to ignore during the fit calculation. This tactic is advantageous when some points are known to deviate from the design model (so including them in the fit calculation would be detrimental) but their exact deviation is still unknown (but can be discerned during deviation analysis).

After you click **OK** in the **Scan Set Fitting** dialog box, Pro/ENGINEER calculates the fit and then adjusts the placement of the scan set accordingly. If multiple fits exist, you can display them with the **Display Fit** option on the **Scan Set Display** dialog box.

If you redefine a scan set, and the fit feature needs to be recomputed, you should use the **Update Fit** option. The system retains all of the previously specified references, but it recalculates the scan set placement to reflect the changes made to the scan set.

To Offset Scan Set by Radius of CMM Probe

Continuous contact CMM probes are often used to collect the point data in a scan set, and in order for the data to be accurate it must be offset by the radius of the CMM probe sphere.

1. Click **VERIFICATION > Fit Scan** to open the **FIT SCAN** menu.
2. Click **Create Fit** to open the **SCAN SET FITTING** dialog box.
3. Enter a name for the fit feature.
4. Select the scan set from the **SEL SCAN** list.
5. Click **CMM probe radius** in the **SCAN SET FITTING** dialog box.
6. Enter the radius of the CMM. The radius is incorporated as an offset into the scan set fit.

Analyzing Deviation

About Deviation Analysis

Two types of deviation analysis can be performed: Distance to Surfaces and Geometric. In both cases, the system analyzes the deviation between the selected reference surfaces in the design model and the corresponding best-fit surfaces in the scan sets.

- **Distance to Surfaces**—Calculates the distance from points in the scan set best-fit surfaces to the corresponding reference surfaces in the design model
- **Geometric**—Calculates the size (where applicable) of the scan set best-fit surfaces and also the location and orientation of those surfaces relative to their reference surfaces in the design mode

For different types of surfaces, best-fit is calculated in different ways:

- **Plane Surface**—Calculates the angle between the best-fit and reference planes
- **Cylindrical Surface**—Calculates the following:
    - The radius and cylinder length of the best-fit surface
    - The angle between the best fit surface's axis and that of the reference surface
- **Conical Surface**—Calculates the following:
    - The cone half angle and cone height of the best-fit surface
    - The angle between the axes of the best fit and reference surfaces
    - The shift of the best-fit surface's apex from that of the reference surface
- **Spherical Surface**—Calculates the following:
    - The radius of the best-fit surface
    - The shift of the best-fit of the surface center from that of the reference surface

To Set Up a Deviation Case

1. Click **VERIFICATION > Deviation**. The **DEVIATION** menu opens.
2. Click **DEVIATION > Create**. The **SCAN SET DEVIATION** dialog box opens.

Deviation Menu Commands

The **Deviation** menu has the following commands:

- **Create**—Creates deviation features with the **SCAN SET DEVIATION** dialog box
- **Redefine**—Redefines elements of a deviation feature with the **SCAN SET DEVIATION** dialog box

- **Delete**—Removes a deviation feature

Verification Results

About Verification Results

After you have set up deviation cases, you can run them. The results of each analysis are written to a file (`deviation-case.cdv` for Distance to Surfaces features and `deviation-case.dat` for Geometric features).

Later, you can read in the Distance from Surfaces results and analyze them graphically. You can also tabulate them, plot them, or export them to a text file, so that they can be analyzed by other tools, such as spreadsheets.

Deviation features are computed and output through the **Ver Results** menu. That menu has the following commands:

- **Compute**—Computes selected deviation features
- **Display**—Graphically displays computed Distance to Surfaces deviation feature results
- **Tabulate**—Tabulates computed Distance to Surfaces deviation feature results
- **Export**—Writes computed **Distance to Surfaces** deviation feature results to a text file
- **Done/Return**—Returns to the **VERIFICATION** menu

To Compute Deviation Features

1. On the **VER RESULTS** menu, click **Compute**. The **SELDEV** menu appears.
2. Select the features you want to analyze and then click **Done Sel**.

   The system calculates in order the selected deviation features. The message window indicates which feature is being processed and its deviation type. The working indicator (clock) also appears on the screen and the STOP sign appears in the message window.
3. If you want to stop the calculations, click the STOP sign.

To Graphically Display Deviation Feature Results

1. Click **Display** from the **VER RESULTS** menu. The **DISP RESULTS** menu appears with the following commands:
   - **Sel Deviatn**—Opens the **SELECT FILE** menu with its filter set for `*.cdv` files (Distance to Surfaces deviation feature analysis results). Select the file.
   - **Setup Disp**—Sets up colors and ranges for point display. This command is unavailable until a deviation feature is displayed on the screen.

- **Plot Disp**—Plots the Distance to Surface analysis results that were displayed on the screen. This command is unavailable until a deviation feature is displayed on the screen.
- **Done/Return—**Returns to the **VER RESULTS** menu.

2. Click **Sel Deviatn** and select a deviation feature. The system reads the file. A COLOR RANGE palette appears. The system paints the design model in colors that represent the deviations of the points in the scan set.

To Plot Computed Deviation Case Results

1. On the **DISP RESULTS** menu, click **Plot Disp** while a deviation case is being displayed. The **Plot** dialog box opens.
2. Select the plotter or plot mode from the **Plotter** list. You can repaint the current plot on the screen, send it to a postscript file, or send it to a plotter.
3. Set up the plotter by clicking **Plot Setup**. The **Plot Setup** dialog box opens. When you have finished, click **OK** to return to the **Plot** dialog box.
4. Select the paper size or the height and width of the plot and the units.
5. Enter the name of the plot file.
6. Click **OK**. The system scales the plot and saves it in the specified file. You can either display the file's contents or send the file to the plotter.

To Tabulate Deviation Cases

1. Click **Tabulate** from the **VER RESULTS** menu. The **Tabulate Deviation Results** dialog box opens.
2. The dialog box panel lists the deviation cases that have been calculated. Click one to select it, and click it again to deselect it. You may select more than one case at a time. The window on the top right shows the number of uniform ranges in which the data is tabulated (the default is 15).
3. Select the range set to be used as follows:
   - **Uniform Ranges**—Uses the number of ranges (from 1 through 50) you select from the menu under the top-right window
   - **Previous Display Range**—Uses the ranges that were last shown in the **Color Range**
4. Click **OK**. The system displays the tabulated results in an information window and also saves them in information files (`deviation-case.inf.#`).

To Export Computed Deviation Results

1. On the **VER RESULTS** menu, click **Export**
2. Select the desired format of the output file by clicking one of commands on the **EXPORT** menu:
   - **Deviation**—Outputs only point deviations to a text file.
   - **Pts & Deviation**—For each point, outputs the xyz coordinates and deviation to a text file.
   - **Pts & Dev Vector**— For each point, outputs the xyz coordinates and the deviation vector to a text file.

   The browser opens with its filter set for `*.cdv` (Distance to Surfaces deviation-case result files).
3. Select the deviation feature file. The system now writes the results to the text file `deviation-case.txt`.

Index