

PC-DMIS Laser Manual

For Version 2019 R2



Generated June 10, 2019
Hexagon Manufacturing Intelligence

Copyright © 1999-2001, 2002-2019 Hexagon Manufacturing Intelligence – Metrology Software, Inc. and Wilcox Associates Incorporated. All rights reserved.

PC-DMIS, Direct CAD, Tutor for Windows, Remote Panel Application, DataPage, DataPage+, and Micro Measure IV are either registered trademarks or trademarks of Hexagon Manufacturing Intelligence – Metrology Software, Inc. and Wilcox Associates, Inc.

SPC-Light is a trademark of Lighthouse.

HyperView is a trademark of Dundas Software Limited and HyperCube Incorporated.

Orbix 3 is a trademark of IONA Technologies.

Unigraphics and NX are either trademarks or registered trademarks of EDS.

Teamcenter is either a trademark or registered trademark of Siemens.

Pro/ENGINEER and Creo are either trademarks or registered trademarks of PTC.

CATIA is either a trademark or registered trademark of Dassault Systemes and IBM Corporation.

ACIS is either a trademark or registered trademark of Spatial and Dassault Systemes.

3DxWare is either a trademark or registered trademark of 3Dconnexion.

The dnAnalytics library v.0.3, copyright 2008 dnAnalytics

lp_solve is a free software package licensed and used under the GNU LGPL below.

nanoflann is a free software package licensed and used under the BSD license below.

NLopt is a free software package licensed and used under the GNU LGPL below.

Qhull is a free software package licensed and used under the license below.

Eigen is a free software package licensed and used under the MPL2 and GNU LGPL licenses below.

RapidJSON is a free software package licensed and used under the MIT license below.

Ipsolve information

PC-DMIS uses a free, open-source package called lp_solve (or Ipsolve) that is distributed under the GNU Lesser General Public License (LGPL).

lpsolve citation data

Description: Open source (Mixed-Integer) Linear Programming system

Language: Multi-platform, pure ANSI C / POSIX source code, Lex/Yacc based parsing

Official name: lp_solve (alternatively lpsolve)

Release data: Version 5.1.0.0 dated 1 May 2004

Co-developers: Michel Berkelaar, Kjell Eikland, Peter Notebaert

Licence terms: GNU LGPL (Lesser General Public Licence)

Citation policy: General references as per LGPL

Module specific references as specified therein

You can get this package from:

http://groups.yahoo.com/group/lp_solve/

Crash Reporting Tool

PC-DMIS uses this crash reporting tool:

"CrashRpt"

Copyright © 2003, Michael Carruth

All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

Neither the name of the author nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES,

INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

nanoflann Library

PC-DMIS uses the nanoflann library (version 1.1.8). The nanoflann library is distributed under the BSD License:

Software License Agreement (BSD License)

Copyright 2008-2009 Marius Muja (mariusm@cs.ubc.ca). All rights reserved.

Copyright 2008-2009 David G. Lowe (lowe@cs.ubc.ca). All rights reserved.

Copyright 2011 Jose L. Blanco (joseluisblancoc@gmail.com). All rights reserved.

THE BSD LICENSE

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY THE AUTHOR "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE AUTHOR BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT

(INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

NLopt Library

PC-DMIS uses the NLopt library (2.4.2). The NLopt library is distributed under the GNU Lesser General Public Licence.

NLopt has this main copyright:

Copyright © 2007-2014 Massachusetts Institute of Technology Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

NLopt also contains additional subdirectories with their own copyrights that are too numerous to list here (see the subdirectories on this project page: <https://github.com/stevengj/nlopt>).

Qhull Library

PC-DMIS uses the Qhull library (2012.1):

Qhull, Copyright © 1993-2012

C.B. Barber

Arlington, MA

and

PC-DMIS Laser: Introduction

The National Science and Technology Research Center for Computation and Visualization of Geometric Structures

(The Geometry Center)

University of Minnesota

email: qhull@qhull.org

This software includes Qhull from C.B. Barber and The Geometry Center.

Qhull is copyrighted as noted above. Qhull is free software and may be obtained via [http](http://www.qhull.org) from www.qhull.org. It may be freely copied, modified, and redistributed under the following conditions:

1. All copyright notices must remain intact in all files.
2. A copy of this text file must be distributed along with any copies of Qhull that you redistribute; this includes copies that you have modified, or copies of programs or other software products that include Qhull.
3. If you modify Qhull, you must include a notice giving the name of the person performing the modification, the date of modification, and the reason for such modification.
4. When distributing modified versions of Qhull, or other software products that include Qhull, you must provide notice that the original source code may be obtained as noted above.
5. There is no warranty or other guarantee of fitness for Qhull, it is provided solely "as is". Bug reports or fixes may be sent to qhull_bug@qhull.org; the authors may or may not act on them as they desire.

Eigen Library

PC-DMIS uses the Eigen Library. This library is primarily licensed under the Mozilla Public Library Version 2.0 (MPL2) license (<https://www.mozilla.org/en-US/MPL/2.0/>) and partly licensed under the GNU Lesser General Public Licence (LGPL). For more information, see Licensing at <http://eigen.tuxfamily.org>.

RapidJSON Information

PC-DMIS uses the RapidJSON software package. The software is used and distributed under this MIT license:

Terms of the MIT License:

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Protocol Buffers Information

PC-DMIS uses Google's protocol buffers mechanism. The code is used and distributed under the terms of this license:

Copyright 2014, Google Inc. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
- Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
- Neither the name of Google Inc. nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT

NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE. Code generated by the Protocol Buffer compiler is owned by the owner of the input file used when generating it. This code is not standalone and requires a support library to be linked with it. This support library is itself covered by the above license.

Non-Negative Least Squares

PC-DMIS uses the Non-Negative Least Squares Algorithm for Eigen:

Copyright © 2013 Hannes Matuschek

It is available at <https://github.com/hmatuschek/eigen3-nnls>. It is subject to the terms of the Mozilla Public License v. 2.0. You can find the license at <http://mozilla.org/MPL/2.0/>.

ZeroMQ libzmq 4.0.4 Library

PC-DMIS uses the libzmq 4.0.4 library by ZeroMQ (<http://zeromq.org>). The code is used and distributed under the terms of the GNU Lesser General Public License V3 (<https://www.gnu.org/licenses/lgpl-3.0.en.html>). For more information on the ZeroMQ license, see <http://zeromq.org/area:licensing>.

Freeicons.png Information

These icons from freeicons.png are used in our help documentation:

- eye icon
- computer icon
- lightbulb icon

IPOPT Large-scale Nonlinear Optimization Library

PC-DMIS uses the IPOPT large-scale nonlinear optimization library which is distributed under the Eclipse Public License (EPL). For details on the IPOPT large-scale nonlinear optimization library, see <https://projects.coin-or.org/Ipopt>.

For details on the Eclipse Public License, please see <https://www.eclipse.org/legal/epl-v10.html>.

Hfb / Miniball Library

PC-DMIS uses the hfb / miniball library for some of its computations. The code is used and distributed under the terms of this Apache 2.0 License:

Copyright 2017 Martin Kutz, Kaspar Fischer, Bernd Gärtner

Licensed under the Apache License, Version 2.0 (the "License");

you may not use this file except in compliance with the License.

You may obtain a copy of the License at

<http://www.apache.org/licenses/LICENSE-2.0>

Unless required by applicable law or agreed to in writing, software

distributed under the License is distributed on an "AS IS" BASIS,

WITHOUT WARRANTIES OR CONDITIONS OF ANY KIND, either express or implied.

See the License for the specific language governing permissions and

limitations under the License.

For details on the hfb / miniball library, see <https://github.com/hbf/miniball>.

For details on the Apache 2.0 License, see <http://www.apache.org/licenses/LICENSE-2.0.html>.

Newuoa Algorithm

PC-DMIS uses the Newuoa algorithm for some of its alignment computations. The code is used and distributed under the terms of this MIT license:

Copyright (c) 2004, by M.J.D. Powell <mjdp@cam.ac.uk>

2008, by Attractive Chaos <attractivechaos@aol.co.uk>

Permission is hereby granted, free of charge, to any person obtaining

a copy of this software and associated documentation files (the

PC-DMIS Laser: Introduction

"Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

For details on the Newuoa algorithm, see <http://mat.uc.pt/~zhang/software.html>.

PDF to PNG Conversion Libraries

PC-DMIS uses functionality from these open-source libraries to convert .pdf files to .png files:

Poppler - Poppler is a PDF rendering library based on the xpdf-3.0 code base. For details on Poppler, see <https://poppler.freedesktop.org/>. Both xpdf and Poppler are licenced under the GNU General Public License (GPL). For license information, see <https://gitlab.freedesktop.org/poppler/poppler/blob/master/COPYING3>. PdfToImage is our software component that uses Poppler. To comply with licensing, PdfToImage is an open-source component and is available for download here: <ftp://ftp.wilcoxassoc.com/PdfToImage/PdfToImage.cpp>.

Cairo - Cairo is a 2D graphics library with support for multiple output devices. For details on Cairo, see <https://cairographics.org/>. It can be redistributed and/or modified under the terms of either the GNU Lesser General Public License (LGPL) version 2.1

(<https://www.gnu.org/licenses/old-licenses/lgpl-2.1.en.html>) or the Mozilla Public License (MPL) version 1.1 (<https://www.mozilla.org/en-US/MPL/1.1/>).

Both Poppler and Cairo depend on the following open-source libraries:

Pixman - Pixman is free open-source and low-level software library for pixel manipulation, providing features such as image compositing and trapezoid rasterization. For details on Pixman, see <http://www.pixman.org/>. You can find licencing information for Pixman from the previous link.

libpng - libpng is a free reference library for reading and writing PNGs. For details on libpng, see <http://www.libpng.org/>. You can find license information for libpng here: <http://www.libpng.org/pub/png/src/libpng-LICENSE.txt>

zlib - zlib is a free available compression library. For details on zlib, see <https://zlib.net/>. You can find license information for zlib here: https://zlib.net/zlib_license.html

FreeType - FreeType is a freely available software library to render fonts. For details on FreeType, see <https://www.freetype.org/>. You can find license information for FreeType here: <https://www.freetype.org/license.html>.

OpenJPEG - OpenJPEG is an open-source JPEG 2000 codec written in C language. For details on OpenJPEG, see <http://www.openjpeg.org/>. The OpenJPEG code is released under the 2-clauses BSD license. You can find that license information here: <https://github.com/uclouvain/openjpeg/blob/master/LICENSE>

Tesseract OCR

PC-DMIS uses the open-source Tesseract OCR (Optical Character Recognition) to recognize Feature Control Frames (FCF). The code for Tesseract OCR is used and distributed under the terms of this Apache license:

The code in this repository is licensed under the Apache License, Version 2.0 (the "License");

you may not use this file except in compliance with the License.

You may obtain a copy of the License at

<http://www.apache.org/licenses/LICENSE-2.0>

Unless required by applicable law or agreed to in writing, software distributed under the License is distributed on an "AS IS" BASIS,

PC-DMIS Laser: Introduction

WITHOUT WARRANTIES OR CONDITIONS OF ANY KIND, either express or implied.

See the License for the specific language governing permissions and limitations under the License.

For details on Tesseract OCR, see <https://sourceforge.net/projects/tesseract-ocr/>.

For details on the Apache 2.0 License, see <http://www.apache.org/licenses/LICENSE-2.0.html>.

Telerik

Portion of the user interface copyright 2015-2019 Telerik AD.

OMPL

PC-DMIS uses the open-source OMPL (Open Motion Planning Library) for some of its Auto Insert Moves calculations. For information on OMPL, see <https://ompl.kavrakilab.org/index.html>. Citations: Zachary Kingston, Mark Moll, and Lydia E. Kavraki, "Decoupling Constraints from Sampling-Based Planners," in *International Symposium of Robotics Research*, Puerto Varas, Chile, 2017.

The code is used and distributed under the terms of this 3-clause BSD license:

Copyright © 2010–2018, Rice University. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
- Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
- Neither the name of the Rice University nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR

CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Table of Contents

PC-DMIS Laser.....	1
PC-DMIS Laser: Introduction.....	1
Attributes for Laser Measurement	2
Getting Started	3
Step 1: Install and Launch PC-DMIS.....	4
Step 2: Define the Laser Sensor	4
Step 3: Define Setup Options for the Laser Sensor.....	6
Step 4: Calibrate the Laser Sensor	23
Step 5: Check the Calibration Results.....	38
Using the Probe Toolbox in PC-DMIS Laser.....	40
Laser Probe Toolbox: Position Probe tab.....	42
Laser Probe Toolbox: Laser Scan Properties tab.....	44
Laser Probe Toolbox: Laser Filtering Properties tab	54
Laser Probe Toolbox: Laser Pixel CG Locator Properties tab	73
Laser Probe Toolbox: Laser Clipping Region Properties tab	77
Laser Probe Toolbox: Feature Extraction tab	79
Laser Probe Toolbox: CWS Parameter tab	95
Laser Probe Toolbox: Laser AF Multiple Creation tab	99
Execution Modes.....	103
Using Asynchronous Execution Mode.....	103
Using Sequential Execution Mode	105
Using Sound Events.....	107

Using the Laser View	107
Using the Scan Line Indicator	111
Understanding the Visualization Tools.....	113
Pointcloud Scanning Colors	116
Using the Laser Toolbars	117
Pointcloud Toolbar	118
QuickCloud Toolbar	124
Mesh Toolbar	124
Using Pointclouds.....	127
Manipulating Pointclouds	129
Pointcloud Graphical Representation	131
COP Command Mode Text.....	134
Pointcloud Point Information	135
Laser Data Collection Settings.....	137
Using the Simulate Pointcloud Function.....	145
Using the Animation Parameters for Pointcloud Simulation	150
Pointcloud Operators.....	153
Manipulating Pointcloud Operators	154
Thickness Colormap	155
Edit the Color Scale	164
SELECT.....	172
CROSS SECTION	174
SURFACE COLORMAP	210

POINT COLORMAP	225
CLEAN.....	231
PURGE.....	235
FILTER	236
Pointcloud EXPORT	238
RESET.....	242
EMPTY	243
Pointcloud IMPORT	244
BOOLEAN	246
Gages	247
Caliper Overview	248
2D Radius Gage Overview.....	262
Pointcloud Alignments	270
Pointcloud/CAD Alignment Dialog Box Description	271
Creating a Pointcloud/CAD Alignment	274
COPCADB Command Mode Text.....	279
Creating a Pointcloud/Pointcloud Alignment	280
COPCOPBF Command Mode Text.....	284
TCP/IP Pointcloud Server.....	285
Extracting Auto Features from Pointclouds.....	290
Defining a Laser Auto Feature by Clicking on a Pointcloud	291
Executing Scan-Extracted Auto Features.....	293
Aligning Measured Auto Features to CAD.....	294

Extracting Auto Features from a Mesh	295
Extracting a Laser Auto Surface Point from a Mesh	298
Creating Auto Features with a Laser Sensor	302
Implementation of QuickFeatures in PC-DMIS Laser	303
Common Laser Auto Feature Dialog Box Options.....	304
Laser Surface Point	309
Laser Edge Point	318
Laser Plane	324
Laser Circle	328
Laser Slot	333
Laser Flush and Gap	339
Laser Polygon.....	356
Laser Cylinder.....	360
Laser Cone	367
Laser Sphere	373
Clearing Auto Feature Scan Data	376
Scanning Your Part Using a Laser Sensor	377
Introduction to Performing Advanced Scans	377
Common Functions of the Scan Dialog Box.....	378
Performing a Linear Open Advanced Scan	395
Performing a Patch Advanced Scan	398
Performing a Perimeter Advanced Scan	401
Performing a Freeform Advanced Scan	406

Performing a Grid Advanced Scan.....	408
Performing a Surface Advanced Scan	411
Performing a Manual Laser Scan on DCC Machines	413
Setting the Machine Speed for Scanning	414
Handling Laser Sensor Errors with On Error Command	415
Using the Mesh Commands	417
Creating a Mesh Feature	419
Creating a Mesh Operator	420
Import Mesh in STL Format	443
Export Mesh in STL Format	444
Empty a Mesh.....	445
Mesh Alignment.....	446
Receive a Mesh from OptoCat.....	460
Index.....	463
Glossary	472

PC-DMIS Laser

PC-DMIS Laser: Introduction

This documentation describes how to use PC-DMIS with your laser sensor to measure features on a part, or to collect data. Laser sensors allow you to collect millions of points of data in one or more clouds of points (COP). PC-DMIS then uses these clouds of points for surface contour maps, export to reverse engineering packages, and creation of constructed features and auto features. This documentation discusses how to use PC-DMIS with a non-contact laser sensor to collect and interpret those clouds of points.

PC-DMIS Laser supports these hardware configurations:

- Perceptron - Digital, V4, V4i, V4ix, and V5
- HP-L-10.6 (CMS106) for DCC
- HP-L-20.8 for DCC and Portable
- HP-L-5.8 for CMM. The supported types are:
 - HP-L-5.8A-SYSTEM (AJ)
 - HP-L-5.8T-SYSTEM (TKJ)



You can use the CMS108 on both DCC and Portable machines.

The main topics in this help document include:

- Attributes for Laser Measurement
- Getting Started
- Using the Probe Toolbox in PC-DMIS Laser
- Execution Modes
- Using Sound Events
- Using the Laser View
- Using the Scan Line Indicator
- Understanding the Visualization Tools
- Pointcloud Scanning Colors
- Using the Laser Toolbars
- Using Pointclouds

- Pointcloud Operators
- Gages
- Pointcloud Alignments
- TCP/IP Pointcloud Server
- Extracting Auto Features from Pointclouds
- Extracting Auto Features from a Mesh
- Creating Auto Features with a Laser Sensor
- Clearing Auto Feature Scan Data
- Scanning Your Part Using a Laser Sensor
- Handling Laser Sensor Errors with On Error Command
- Using the Mesh Commands

If you come across something in the software that isn't covered in this documentation, see the PC-DMIS Core documentation.

Attributes for Laser Measurement

Before you get into the details of non-contact laser sensors, you need to understand their attributes to improve the results that you obtain when you use them for measurement. Laser sensors are excellent at gathering large amounts of data quickly. They are also good for measuring parts that may otherwise deform under the pressure of a tactile probe.

However, keep in mind that other factors, such as sunlight, surface finish, surface reflectivity, and surface color can influence measurements taken with laser sensors. To compensate for some of these factors, you can apply filters to the data to handle the influence. You should understand how and why these items affect the measurement results.

Sunlight

Unlike other non-contact systems, laser sensors are not generally affected by standard industrial lighting. Laser sensors work under varied lighting conditions because the sensor's frequency is tuned to its own laser. Only light that has the same frequency as the laser itself can affect the measurement. Because sunlight contains all frequencies of light, it is important to keep sunlight out of the inspection room.

Surface Finish

Because tactile probes are larger than the deviation in most surface finishes, a tactile probe acts as an averaging filter. When the tactile probe contacts the surface, it gives an average of the highest points on the surface. When you use a laser sensor, the light reflects off the surface of the part. How the light reflects depends a lot on the roughness of the surface, even if it does not appear rough to human sight or touch.

Surface Reflectivity

Generally, surfaces with a matte finish work better than those with a glossy finish. A glossy surface finish usually has directional reflection. Based on the angle of the light, you can get too much or too little light. You might even get a "hot spot" (something that looks like a "blob" in the Graphic Display window). This *blob* is actually the image of the light source. The reflection of light might add some extraneous points to the scan line, but the rest of the points are not affected by the reflection. To compensate for surface reflectivity, you can spray the part with an aerosol powder or paint.

Surface Color

Because the laser is light, the surface color can potentially impact the measurement. Similar to how something colored black absorbs heat from the sun, black surfaces on a part absorb the laser's light and make the measurement of those surfaces difficult. Darker colors have more potential for problems than lighter colors. If your part is too dark, you can apply powder coatings to it to make it easier to sample.

It usually takes time and experience with your particular parts and in your specific environment to determine what settings work best for you. You should experiment with the capabilities of your specific sensor to improve the measurement results.



WARNING: Exercise caution when working with Laser sensors as they can damage your eyes. Consult your laser sensor documentation for safety issues and procedures for a safe work environment.

Getting Started

Before you use PC-DMIS with your laser device, the basic steps below can help you verify that your system is properly prepared.

To get PC-DMIS running with your laser sensor, follow these steps:

If you are using a Perceptron laser on a Romer arm, see the "Using a Romer Portable CMM" section of the PC-DMIS Portable documentation.

Step 1: Install and Launch PC-DMIS

Before you use your laser device, ensure that PC-DMIS has been properly installed on your computer system.

To install PC-DMIS for your laser device:

1. Ensure that the machine that runs the laser sensor is properly set up and configured according to your machine's specifications. Follow the documentation that came with your laser sensor to properly connect the hardware.
2. Ensure that your LMS license or portlock supports the Laser option. This tells the installer to install the necessary Laser components. If you don't have the necessary LMS license or your portlock is not properly configured, please contact your PC-DMIS software distributor.
3. Install PC-DMIS. To do this, see the release notes in the Readme.pdf file.
4. Start PC-DMIS in Online mode by selecting **Start | All Programs | <version> | <version> Online**, where <version> represents your version of PC-DMIS.
5. Open an existing measurement routine, or create a new one. If you create a new measurement routine, the **Probe Utilities** dialog box appears so that you can define your laser sensor in the next step.



The PC-DMIS installer manages the installation of drivers, and so on.

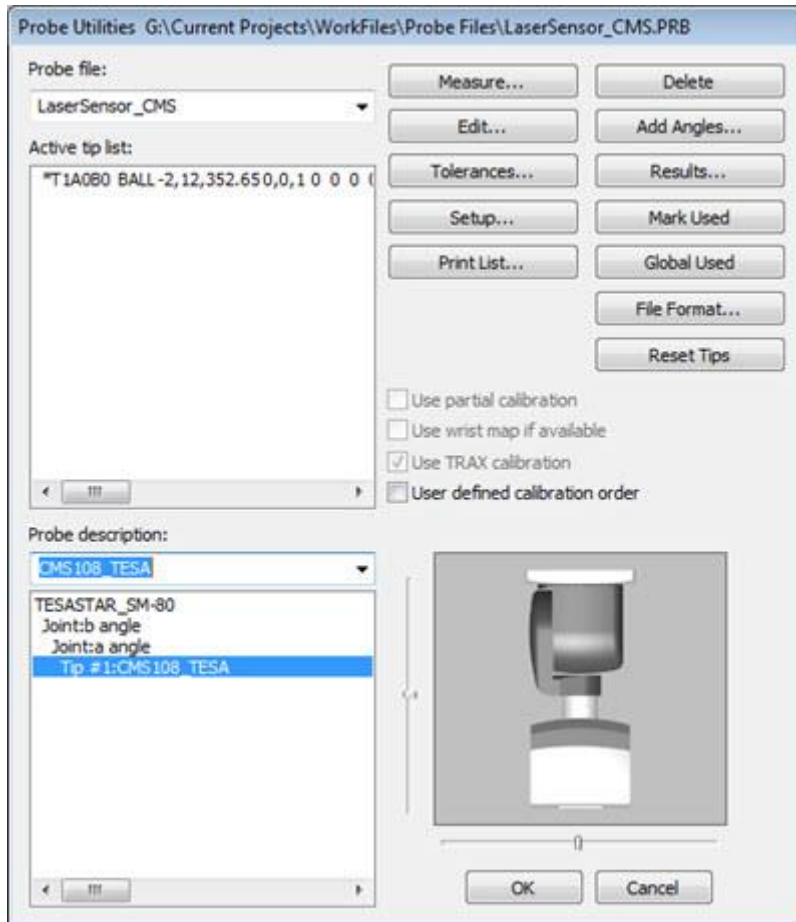
Setting Parameters without a Measurement Routine

Some users may need the ability to change laser parameters without first opening a measurement routine. If needed, you can access the **Laser Sensor** tab for the current laser sensor in the **Setup Options** dialog box by pressing the F5 key or selecting **Edit | Preferences | Setup**. The **Laser Sensor** tab is discussed in Step 3.

Step 2: Define the Laser Sensor

If you don't have a defined laser sensor, use the **Probe Utilities** dialog box to define it. This process creates a probe file.

1. Select the **Insert | Hardware Definition | Probe** menu option to open the **Probe Utilities** dialog box. (This dialog box automatically appears whenever you create a new measurement routine.)



Probe Utilities dialog box

2. In the **Probe file** box, type a name that best describes your laser sensor.
3. From the list of components at the bottom, select the **No probe defined** text to highlight it.
4. From the **Probe Description** list, select the appropriate probe. Most laser sensors connect directly to the PH10M probe head. A CMS108 sensor that you use with a DCC machine connects to a Tesastar probe head. You can mount the CWS or WLS sensor on a wrist with a TKJ connector, or on OPTIV_FIXED on multi-sensor machines.
5. As needed, select additional components in the same manner for "empty connections" until you finish defining the probe. A defined probe shows a tip in the **Active Tip List**.



Once you define the tip, the software no longer shows the probe image. This makes it so that the graphical image of the probe does not obstruct the view of the part during measurement. However, if you want to enable the display of probe components, double-click on the probe component to open the **Edit Probe Component** dialog box. Mark the **Draw this component** check box.

6. If you use a PH10, Tesa, or continuous-type wrists with a C joint, you need to verify that the joint angles are properly adjusted for visual purposes. Otherwise, PC-DMIS can't properly correlate the sensor's data to the machine position. If your probe is not rotated correctly about the joint, you can manually provide the extra rotation. To do this, right-click on the component and change the **Default rotation angle about connection** value to reflect the needed rotation.



The probe file does not define the orientation of the sensor about the joint; it only defines the probe vector.

For additional information on how to define probes, see the "Defining Hardware" section in the PC-DMIS Core documentation.

Step 3: Define Setup Options for the Laser Sensor



If PC-DMIS is configured for the HP-L-20.8 laser sensor at startup, the system looks for the current-mounted probe. If it is *not* the laser HP-L-20.8 sensor, and a probe changer is present, the system assumes that the sensor is in the probe changer and switches on the warm-up power state. This ensures that the sensor is warmed and ready for measuring.

1. If you see the **Probe Utilities** dialog box from the previous step, close it.
2. Select **Edit | Preferences | Setup**, or press **F5**, to open the **Setup Options** dialog box.



There is no tab on the **Setup Options** dialog box for the CWS probe.

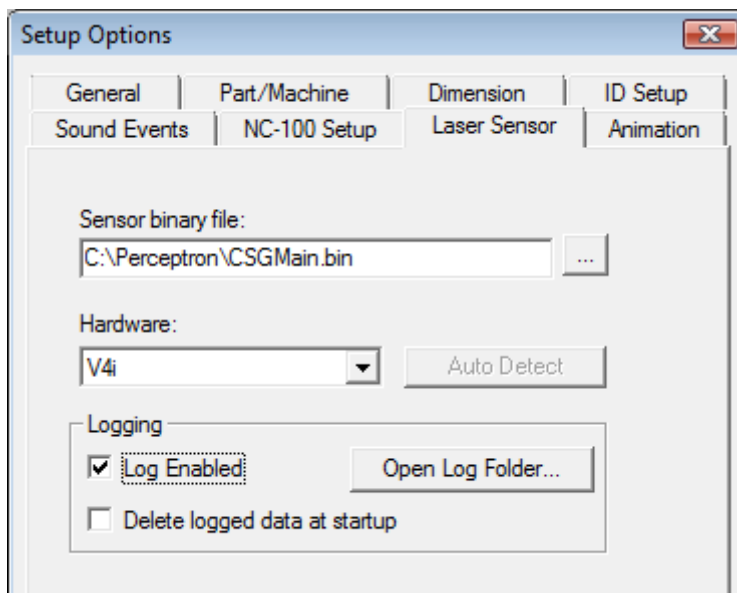
3. Select the **Laser Sensor** tab. The contents of this tab change based on the type of laser sensor that is defined in your LMS license or portlock.
 - Perceptron Sensors
 - CMS Sensors
 - Using Zeiss Eagle Eye 2 with Zeiss I++ DME Server
 - Comparison of HP-L-5.8 and HP-L-10.6 Sensors
4. Follow the setup options instructions below for your laser sensor.

Registry Entries for Laser Sensors

A PH10 wrist can automatically switch between a contact probe and a Perceptron probe. These registry entries control that operation as well as the power on a laser sensor warm-up station:

- `PICSDifferentialSwitchBit`
- `WarmUpStationPowerBit`

Perceptron Sensors



Setup Options dialog box - Laser Sensor tab example pointing to the binary file for Perceptron sensors

Sensor Binary File - You can use the browse button (...) to browse to the location of the CSGMain.bin binary file. This binary file contains the sensor configuration that came with your probe. The process that installs the toolkit and drivers for your probe also installs this binary file.

Hardware list - You can specify the hardware, and PC-DMIS remembers what options (Greysums, V5 Projectors, Flat Target Calibration, and so on) to allow or disallow even when you run PC-DMIS in Offline mode. In Offline mode, all the options for the selected hardware type are available for revision.

AutoDetect - This button checks the hardware attached to your machine. It verifies that the hardware you specified in the **Hardware** list is correct.

Logging area - You can use this area to generate text-based log files that contain communication results between PC-DMIS and the laser sensor when the measurement routine executes. The information it sends to the log files includes scans, nominals of calculated features, and so on. Hexagon Technical Support can use these files to resolve issues that involve your laser sensor.

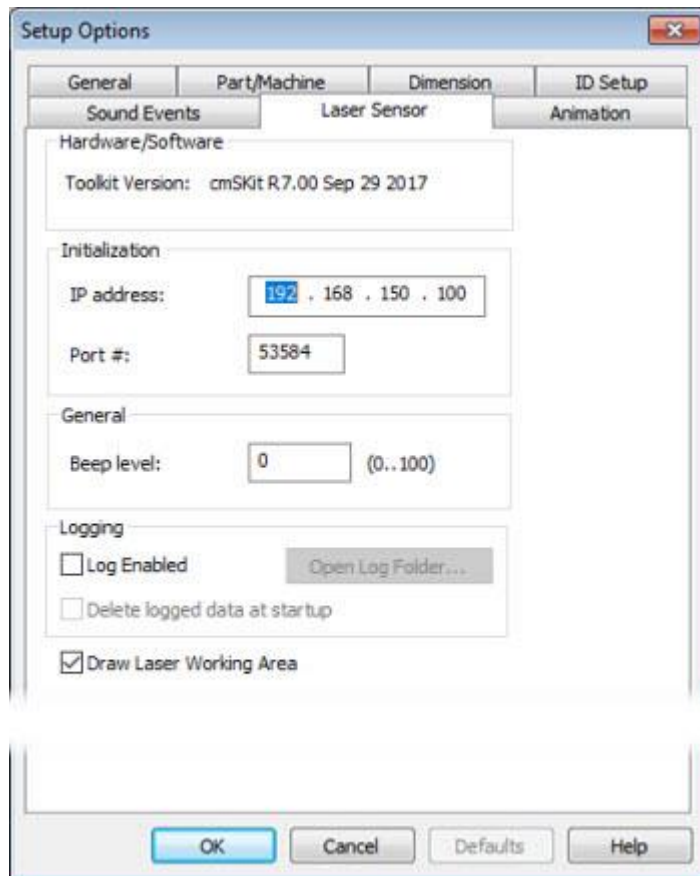
- **Log Enabled** - This check box enables or disables data sent to the log files.
- **Open Log Folder** - This button opens up the folder that contains your log files.



For PC-DMIS 2019 R2, the folder content is located in
C:\ProgramData\Hexagon\PC-DMIS\2019 R2\NCSensorsLogs\

- **Delete logged data at startup** - This check box deletes the logged data files from the log folder whenever you create a new measurement routine.

CMS Sensors



Setup Options dialog box - Laser Sensor tab example for CMS sensors

Hardware/Software Area

This area displays the current CMS toolkit version.

Initialization Area

You can use the **IP address** and **Port #** boxes to define the IP address and the port number of the CMS controller.

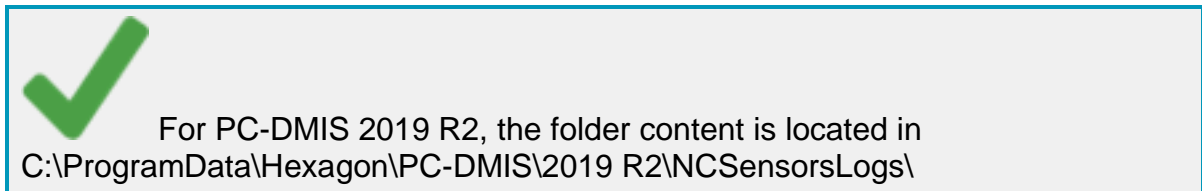
General Area

You can use the **Beep level** box to set the volume for beep sounds that come from the CMS controller. It can accept any value from 0 to 100. A value of 0 turns off the volume completely.

Logging Area

Logging area - You can use this area to generate text-based log files that contain communication results between PC-DMIS and the laser sensor when the measurement routine executes. The information it sends to the log files includes scans, nominals of calculated features, and so on. Hexagon Technical Support can use these files to resolve issues that involve your laser sensor.

- **Log Enabled** - This check box enables or disables data sent to the log files.
- **Open Log Folder** - This button opens up the folder that contains your log files.

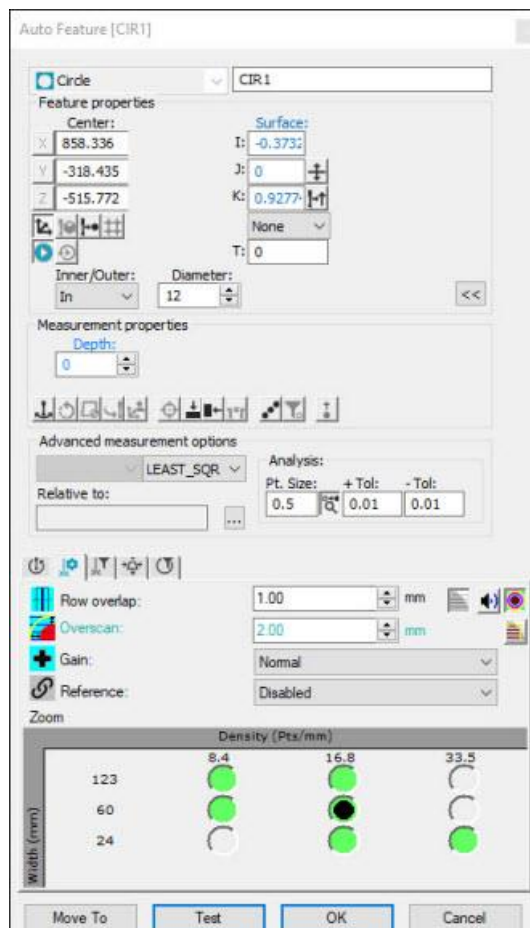


- **Delete logged data at startup** - This check box deletes the logged data files from the log folder whenever you create a new measurement routine.

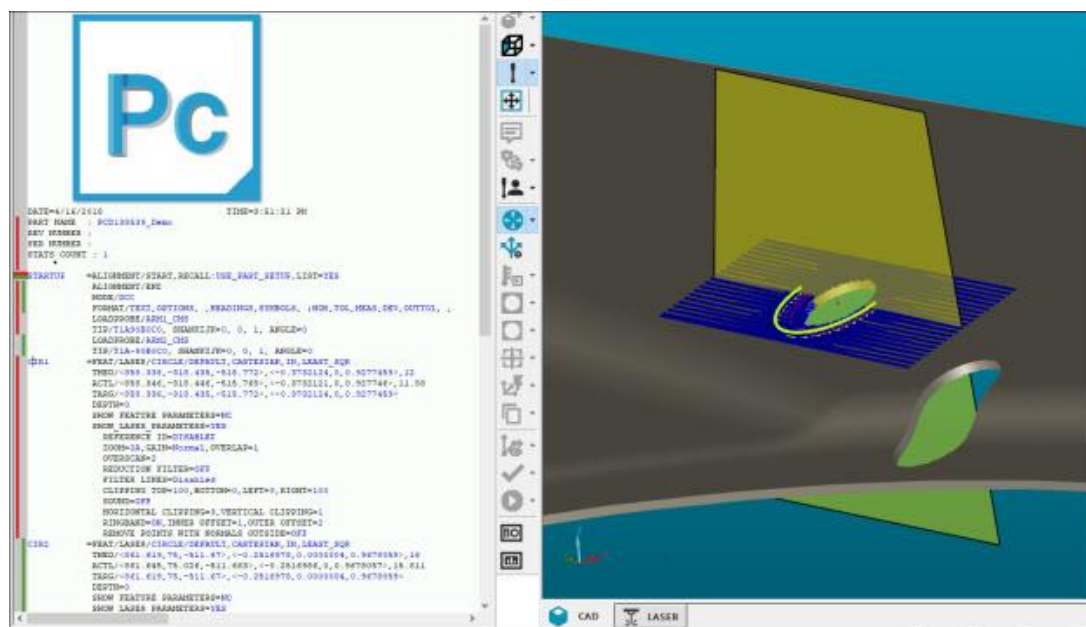
Draw Laser Working Area Check Box

If you select the **Draw Laser Working Area** check box, CMS probe parameters draw the trapezoid with the correct dimensions. This functionality helps with simulation in Offline mode. This feature is available for Laser Auto Features and laser scans.

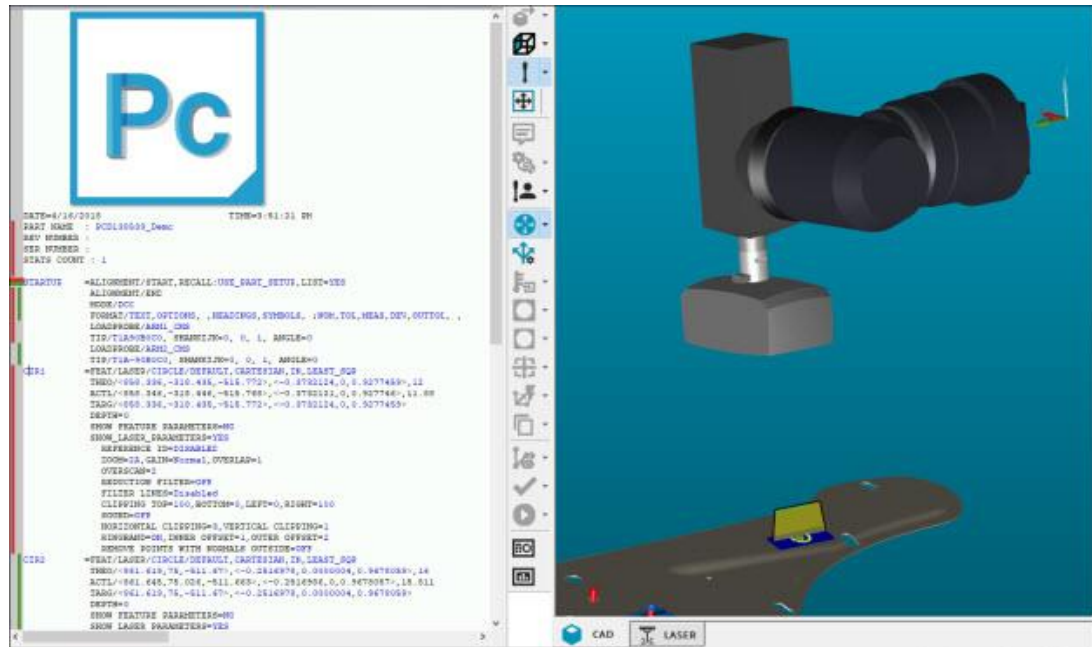
- For a Laser Auto Feature, the trapezoid that represents the working area of the laser is shown in the center of the feature. The trapezoid moves according to the simulation of the laser stripes. For an example, see the images below:



Example of Circle Auto Feature dialog box



Example of Circle Auto Feature



Example of Circle Auto Feature

- For a laser scan, the trapezoid that represents the working area of the laser is shown as the start point. The trapezoid moves according to the simulation of the laser stripes. For an example, see the images below:

Linear Open Scan

Scan type:

ID:

Scan parameters
 Increment:

CAD controls
☐ Select

To Points conversion

Theoretical scan points

#	X	Y	Z
37	760.052	-5.566	-5.0
38	760.050	-5.469	-5.0

☐ Manual points

Boundary points and vectors

#	X	Y
D	759.437	-59.584
2	760.050	-5.469

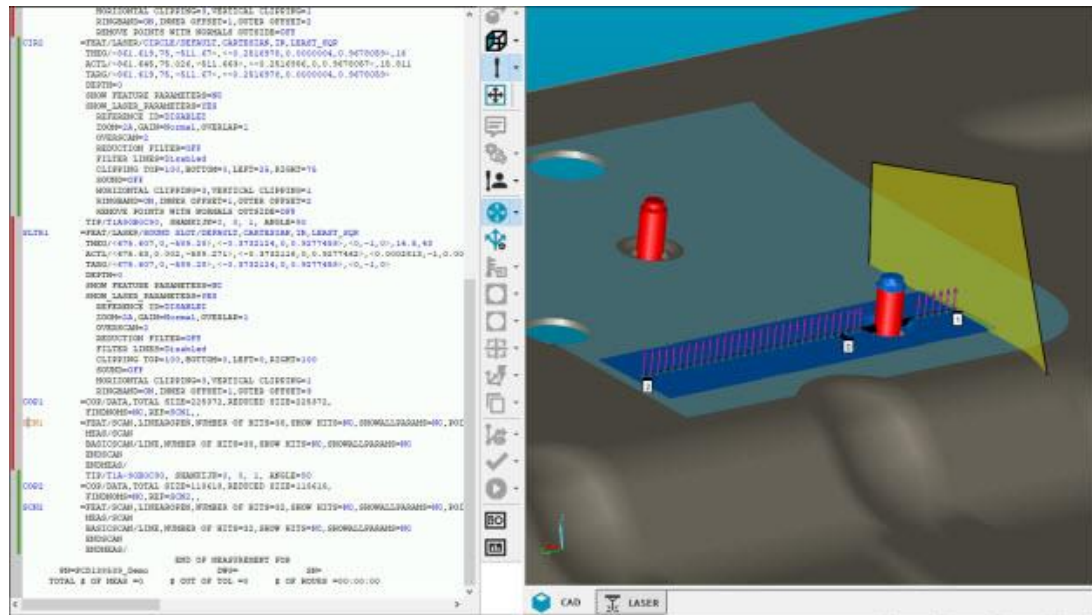
V...	I	J
C...	-0.968	-0.015
E...	-0.252	0.000

Point cloud reference feature:

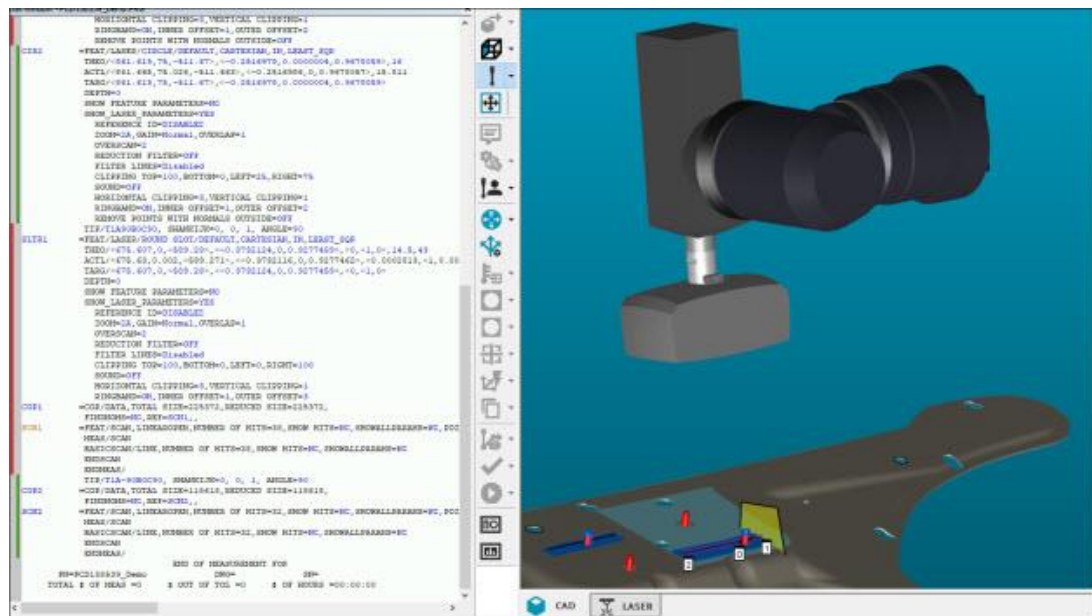
ARM1_CMS

X	Y	Z	W
761.304	-91.162	-537.759	0

Example of Linear Open Scan dialog box



Example of Linear Open Scan



Example of Linear Open Scan

If you change the zoom settings (located on the **Laser Scan Properties** tab) and sensor-based clipping settings (located on the **Laser Clipping Region Properties** tab), PC-DMIS updates the trapezoid.

Using Zeiss Eagle Eye 2 with Zeiss I++ DME Server

The steps below describe how to use the Zeiss Eagle Eye 2 with the Zeiss I++ DME server.

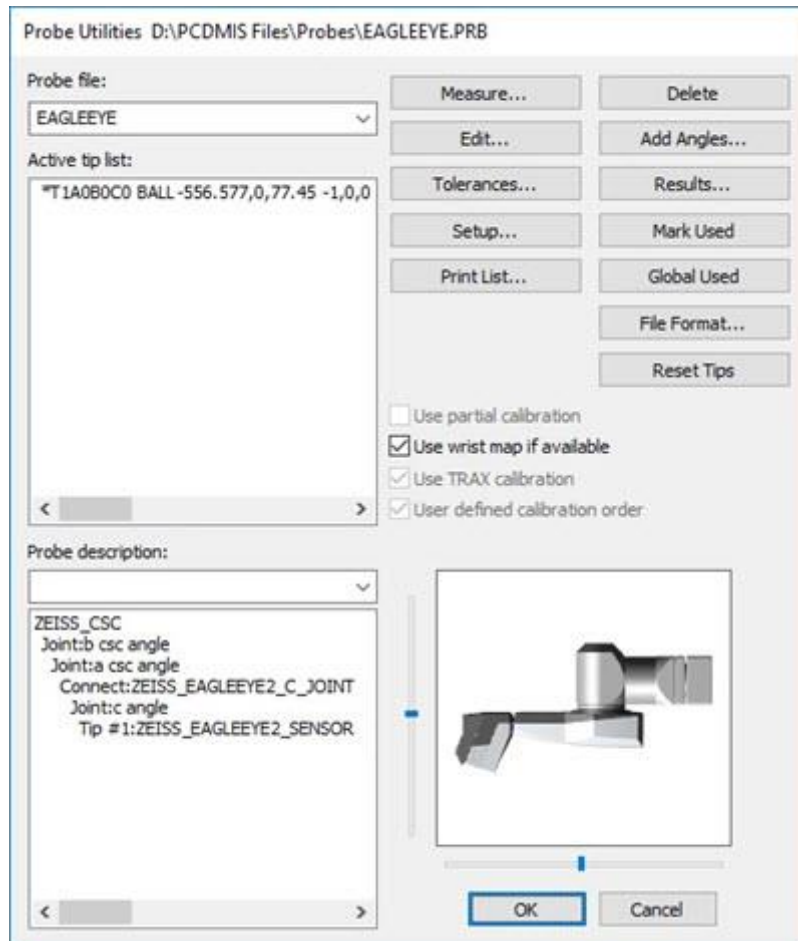
1. Set up the PC-DMIS I++ client. For details, see "I++ DME Client Interface" in the MIIM documentation.



The qualification of the sensor is done inside the I++ DME server.

You can access the MIIM help file in the language subfolder where PC-DMIS is installed. For English, this is the **en** subfolder.

2. Use the `ZeissWrist` registry entry to enable the wrist in PC-DMIS. For details, see "ZeissWrist" in the "Option" section of the PC-DMIS Settings Editor documentation.
3. Define the probe assembly.



Probe Utilities dialog box

4. Select the **Use wrist map if available** check box.
5. Select the tip from the **Active tip list** and then click **Edit** to open the **Edit Probe Data** dialog box.

Edit Probe Data

Tip ID: T1A0B0C0

DMIS label:

X center: -556.577

Y center: 0

Z center: 77.45

Shank I: -1

Shank J: 0

Shank K: 0

Thickness: 0

Diameter: 0

PrbRdv: 0

ScanRdv: 0

With Averaging

Diameter: 0

PrbRdv: 0

Fastprobe Mode

X center: -556.577

Y center: 0

Z center: 77.45

Diameter: 0

PrbRdv: 0

With Averaging

Diameter: 0

PrbRdv: 0

Calibration date: Unknown

Calibration time: Unknown

Gage Scan Filter: None

Nickname: EE2_2

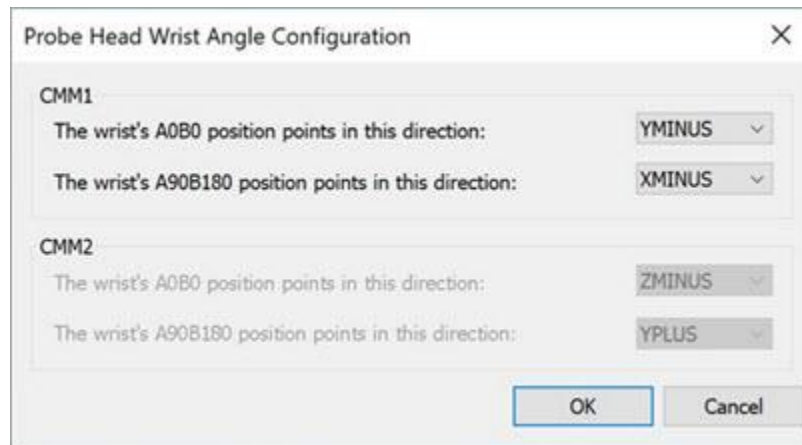
OK

Cancel

Edit Probe Data dialog box

6. Enter a name in the **Nickname** box for the A0B0C0 tip that corresponds to the probe name given in the I++ DME Server for the EagleEye probe.
7. Set up the probe orientation:
 - a. Open the **Setup Options** dialog box (**Edit | Preferences | Setup**).
 - b. Select the **Part/Machine** tab.
 - c. Click the **Probe Head Orientation** button to open the **Probe Head Wrist Angle Configuration** dialog box.
 - d. In the **CMM1** area, set these two options:
 - Select the **YMINUS** option from the **wrist's A0B0 position points in this direction** list.

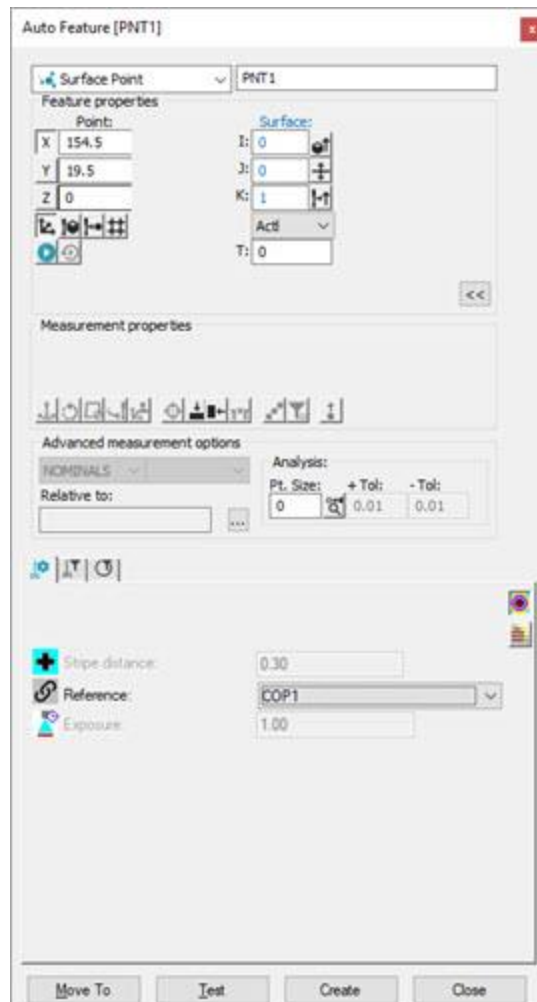
- Select the **XMINUS** option from the **The wrist's A90B180 position points in this direction** list.



Probe Head Wrist Angle Configuration dialog box

Differences between the Zeiss Eagle Eye 2 and HP-L-10.6 (Formerly CMS)

- PC-DMIS does not use the **Laser Sensor** tab in the **Setup Options** dialog box.
- Changes to the **Laser Scan Properties** toolbox tab in the **Auto Feature** dialog box are:
 - For the Eagle Eye 2 measurement, the software hides the **Zoom** and **Gain** properties and adds the **Exposure** and **Stripe distance** properties.
 - **Stripe distance** is the distance between the laser stripes along the path line. Typically, you should use a value between 0.3 and 0.5, inclusive.
 - The default value for the **Exposure** setting is 1.0. The valid values are 0.01 to 20, inclusive.



- Changes to the **Laser Scan Properties** toolbox tab on the **Scan Feature** dialog box are:
 - For the Eagle Eye 2 measurement, the software hides the **Zoom** and **Gain** properties and adds the **Exposure** and **Stripe distance** properties. The **Scan Feature** dialog box settings are the same as the settings described above for the **Auto Feature** dialog box.

Comparison of HP-L-5.8 and HP-L-10.6 Sensors

This topic describes the similarities and differences between the HP-L-5.8 sensor for CMM and the HP-L-10.6 (CMS106) sensor for DCC.

Similarities

- The values in the **Measure Laser Probe** dialog box (**Insert | Hardware Definition | Probe | Measure** button) are the same:

Measure Laser Probe

Motion

☐ Man

☒ DCC

☐ Man+DCC

Measure

Cancel

Type of calibration operation

☒ Tips

☐ Sensor

☐ Offset

☐ Continuous Wrist

Operation Settings

Move speed (mm/sec): 100

Parameter sets

Name:

Save

Delete

Calibration tool

☐ Tool mounted on rotary table

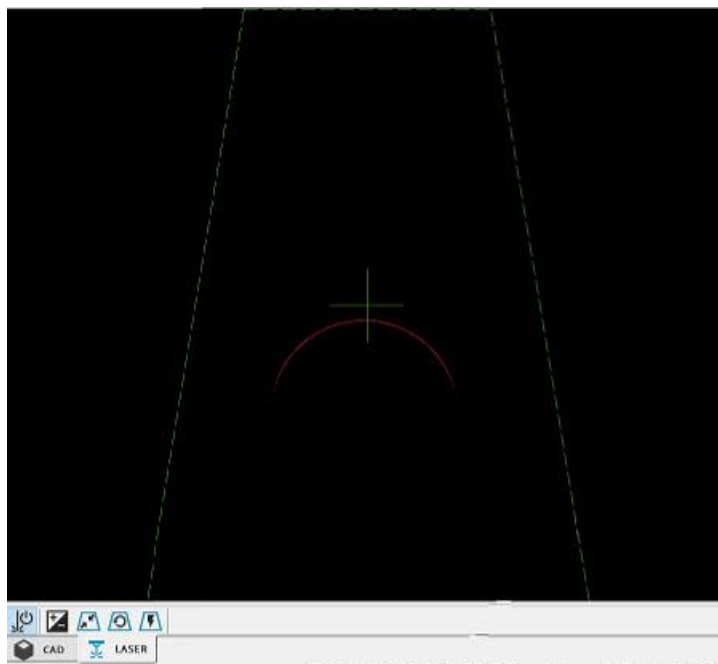
List of available tools:

WHITE SPHERE 0,0,1 25.441 0

Add Tool... Delete Tool Edit Tool...

Measure Laser Probe dialog box

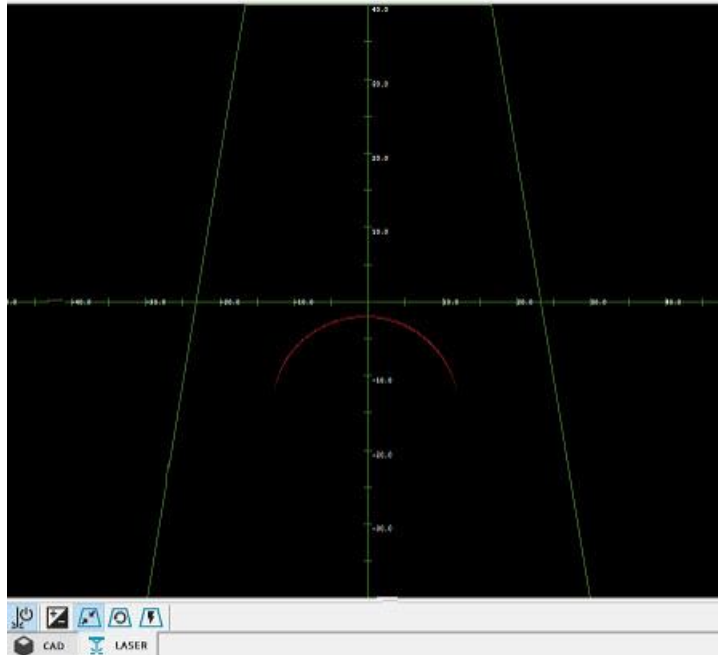
- The X, Y, and Z values on the **Position Probe** tab in the Probe Toolbox are the same.
- The **Laser** tab in the Laser View in the Graphic Display window is the same:



Graphic Display window - Laser tab

Differences

- The shape of the sensor is different.
- The related components in probe.dat are different.
- The standoff distance, and the sensor's field of view (that is, the sensor's geometry) are different:



Graphic Display window - Laser tab

- For the HP-L-5.8, the **AutoGain** button appears in the Laser View in the Graphic Display window. When the HP-L-5.8 sensor is in range on a part, you can select the button to learn the best gain setting, and update the Probe Toolbox accordingly. You can also use this functionality while you set up Laser Auto Features and laser scan properties. For more information about how to set up these properties, see "Creating Auto Features with a Laser Sensor" and "Scanning Your Part Using a Laser Sensor".
- The default value for the **Increment 2** option (the increment distance between scan lines) in the **Scan parameters** area for a Patch Advanced Scan is 45 mm for the HP-L-5.8 (the HP-L-10.6 has a different default value).
- The differences on the **Laser Scan Properties** tab in the Probe Toolbox in the **Auto Feature** dialog box are as follows:
 - The HP-L-5.8 has only one scan zoom state, the dimension of the field of view is fixed. (There are no green option buttons on the **Laser Scan Properties** tab as there are for the HP-L-10.6 and HP-L-20.8.)
 - For the HP-L-5.8, there are five sensitivity modes (**1**, **2**, **3**, **4**, and **5**) in the **Gain** list on the **Laser Scan Properties** tab. When you select a mode, the image in the Laser View updates in real-time. You can also select the **Quality Filter** icon next to the **Gain** list to enable or disable the Quality Filter mode.

Step 4: Calibrate the Laser Sensor

The calibration process described in this step may vary depending on the "Measure Laser Probe Options" and the type of installed interface. For detailed information on laser sensor calibration options, see the "Measure Laser Probe Options" topic.

Calibrating Perceptron Sensors



During calibration, PC-DMIS temporarily overrides your current exposure and gray sum values with the default exposure and gray sum values covered in the "Exposure and Gray Sum Settings During Calibration" topic. Once calibration finishes, the software restores your original values.

Use this procedure to calibrate your laser sensor for the first time:

1. Select **Insert | Hardware Definition | Probe** to open the **Probe Utilities** dialog box.
2. From the **Active Tip List** box, select the tip that you defined in step 2.
3. Click **Measure** to open the **Measure Laser Probe** dialog box (for information on this dialog box, see "Measure Laser Probe Options").
4. From **Type of Calibration Operation**, select one of the options. Then for Perceptron sensors, select **Offset**.
5. Define other calibration options as needed: **Motion** type, **Move Speed**, **Parameter Sets**, and **Calibration Tool**.



If you use a multi-sensor CMM with both a contact probe and a laser probe, make sure a calibrated contact probe first locates the sphere location for the laser calibration tool. This correlates the laser sensor measurement data with the contact probe calibration.

6. Click **Measure** to begin the calibration procedure. Follow any on-screen instructions. The first several prompts that you see are identical to the setup procedure for touch-trigger probes.



If you use the **MAN** or **MAN + DCC** motion options, or if you answer **Yes** to the message “Has the sphere moved”, you need to bisect the qualification sphere. For information, see "Bisecting the Calibration Sphere". Once you do an Offset calibration, the software no longer asks you to bisect the sphere unless you answer **Yes** to the message “Has the sphere moved”.



Certain sensor tip angles may cause the laser beam to fall on a portion of the calibration tool's stem. In some cases, the standard deviation for the sensor calibration of those tips exceeds the expected amount. In those cases, PC-DMIS displays a message to ask if you want to repeat the calibration of those tips. If you click **Yes**, the system uses the offsets and orientation determined by the first measurement rather than the theoretical values. This results in a clipping around the target that is more accurate during this re-calibration.

7. Once execution finishes, PC-DMIS returns to Learn mode and shows the **Probe Utilities** dialog box.
8. Once the sensor calibration finishes, PC-DMIS shows the **Probe Utilities** dialog box.
9. If needed, click **Add Angles** to define any other tip angles that you need to calibrate.
10. From the **Active Tip List** box, select the tips that you want to calibrate. The initial tip calibration only finds offset information for the sensor configuration.
11. Click **Measure** to open the **Measure Laser Probe** dialog box. If you don't select any angles, the software asks if you want to calibrate all the tips.
12. From the **Measure Laser Probe** dialog box, select the **Tips** option.
13. For **Calibration Tool**, select the same tool that you used earlier.
14. Click **Measure** to begin the tip calibration. Once calibration finishes, PC-DMIS shows the **Probe Utilities** dialog box.



PC-DMIS stores Offsets of each axis for Perceptron sensors in the registry as `HotSpotErrorEstimateX`, `HotSpotErrorEstimateY`, and `HotSpotErrorEstimateZ`. For details, see "`HotSpotErrorEstimateXYZ`" in the PC-DMIS Settings Editor documentation.

Once you perform either **Offsets** or **Sensor** calibration, depending on the sensor type, you need to perform only steps 8 through 15 on any new probe file that utilizes the same sensor and CMM.

Calibrating Portable CMS Laser Sensors

Use this procedure to calibrate a portable laser CMS sensor using a planar artifact:

1. From the **Probe Utilities** dialog box, click **Measure** to open the **Measure Laser Probe** dialog box. For information on this dialog box, see "Measure Laser Probe Options".
2. Select the appropriate sensor mode. The default is **Zoom2A**.
3. Place the planar artifact in a convenient location for the arm to measure.
4. Click **Measure** to begin the calibration procedure. Follow any on-screen instructions.
5. The calibration procedure requires that you take 17 laser stripes on the planar artifact in different positions and orientations with respect to the planar artifact. To help you visualize where to take the stripe, the system draws a yellow target line on the **Laser** tab in the Graphic Display window.

Calibrating DCC CMS Laser Sensors

The calibration process described in this step may vary depending on the laser sensor options and the type of installed interface. For detailed information on calibration options, refer to the "Measure Laser Probe Options" topic.

Use this procedure to calibrate your laser sensor for the first time:

1. Select **Insert | Hardware Definition | Probe** to open the **Probe Utilities** dialog box.
2. From the **Active Tip List** box, select the tip that you defined in Step 2.
3. Click **Measure** to open the **Measure Laser Probe** dialog box (for information on this dialog box, see "Measure Laser Probe Options").
4. Select the appropriate sensor mode. The default is **Zoom2A**.

5. Define other calibration options as needed: **Motion** type, **Move Speed**, **Parameter Sets**, and **Calibration Tool**.



If you use a multi-sensor CMM with both a contact probe and a laser probe, make sure a calibrated contact probe first locates the sphere location for the laser calibration tool. This correlates the laser sensor measurement data with the contact probe calibration.

6. Click **Measure** to begin the calibration procedure. Follow any on-screen instructions. The first several prompts that you see are identical to the setup procedure for touch-trigger probes.



If you use the **MAN** or **MAN + DCC** motion options, or if you answer **Yes** to the message "Has the sphere moved", you need to bisect the qualification sphere. For information, see "Bisecting the Calibration Sphere". Once you do an Offset calibration, the software no longer asks you to bisect the sphere unless you answer **Yes** to the message "Has the sphere moved".

7. Once execution finishes, PC-DMIS returns to learn mode and opens the **Probe Utilities** dialog box.
8. If needed, click **Add Angles** to define any other tip angles that you need to calibrate.
9. From the **Active Tip List** box, select the tips that you want to calibrate. The initial tip calibration only finds offset information for the sensor configuration.
10. Click **Measure** to open the **Measure Laser Probe** dialog box. If you don't select any angles, the software asks if you want to calibrate all the tips.
11. From the **Measure Laser Probe** dialog box, select the appropriate sensor mode. The default is **Zoom2A**.
12. Select the **Tips** option.
13. For **Calibration Tool**, select the same tool that you used earlier.
14. Click **Measure** to begin the tip calibration. Once calibration finishes, PC-DMIS shows the **Probe Utilities** dialog box.



Certain sensor tip angles may cause the laser beam to fall on a portion of the calibration tool's stem. In some cases, the standard deviation for the sensor calibration of those tips exceeds the expected amount. In those cases, PC-DMIS displays a message to ask if you want to repeat the calibration of those tips. If you click **Yes**, the system uses the offsets and orientation determined by the first measurement rather than the theoretical values. This results in a clipping around the target that is more accurate during this re-calibration.

Calibrating a CWS/WLS Sensor

You can calibrate the CWS tip offset on a sphere. Sphere tools with a less reflective surface work better than those with a highly reflective surface. Calibration is supported on fixed mount multi-sensor machines and on indexing wrists with a TKJ connector.

The calibration executes using the current temperature compensation.

The measurement range of most CWS probe heads is small. This can mean the manual point taken when the tool has moved or when using the Manual+DCC motion must be very close to the sphere pole or nearest point for the calibration to execute successfully.

During calibration execution, the machine automatically moves to the center of the CWS measurement range, or to the needed measurement range position for each point.

PC-DMIS does not support multiple wrist angle tip calibration in a single calibration operations. You must calibrate each tip separately.

When you calibrate a wrist angle tip for the first time and the tool has not moved, select Man+DCC. For all subsequent measurements of this tip, select DCC.

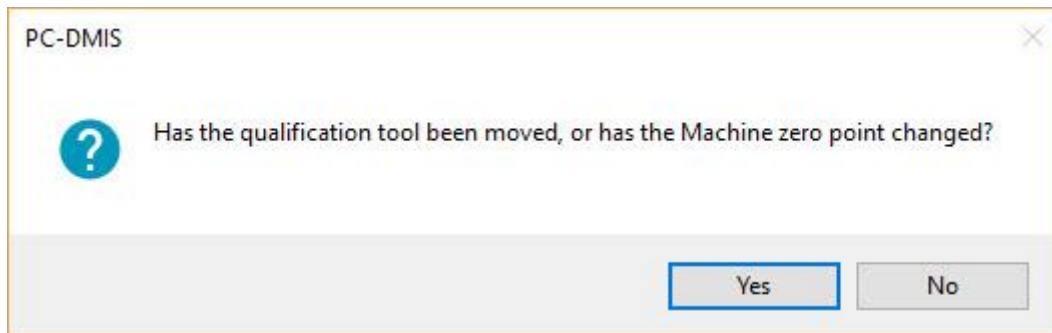


There are no automatic clearance moves before or after the calibration measurement sequence. Ensure clearance for any wrist rotation needed to position the wrist for the specified tip before starting the calibration. Ensure probe clearance for the move to the measurement start position.

The following steps outline the procedure to calibrate your laser sensor for the first time:

1. Select the **Insert | Hardware Definition | Probe** menu item.
2. In the **Probe Utilities** dialog box, define the CWS probe and tip.
3. Select **Measure** to open the **Calibrate Probe Offset** dialog box (for details, see "Measure CWS/WLS Laser Probe").

4. Configure the settings and select **Calibrate**.
5. Indicate if the qualification tool has moved.



If you select **Yes**, PC-DMIS displays the **Execution** dialog box and prompts you to take a manual point. The point should be at the top or nearest point of the sphere from the perspective of the probe and probe vector. If you select **No**, PC-DMIS displays the **Execution** dialog box and begins DCC measurement.

6. After the calibration measurement completes, click **Results** in the **Probe Utilities** dialog box to see detailed results.

Mapping Infinite Wrist DCC CMS Laser Sensors

A hardware configuration of a CMS laser sensor and an infinitely indexable wrist, such as the CW43L, has the ability to qualify infinite tip orientations. You can define the tip orientations by wrist angles A, B, and C through a Laser Wrist Map (LWM). You can create a LWM if you qualify a grid of tip orientations that cover the specified range of angles A, B, and C.

Once you create the LWM for a specific sensor, you can add new tips to the sensor and if those tips are within the angle range that you specified during map creation, they are automatically qualified and you can begin measuring.



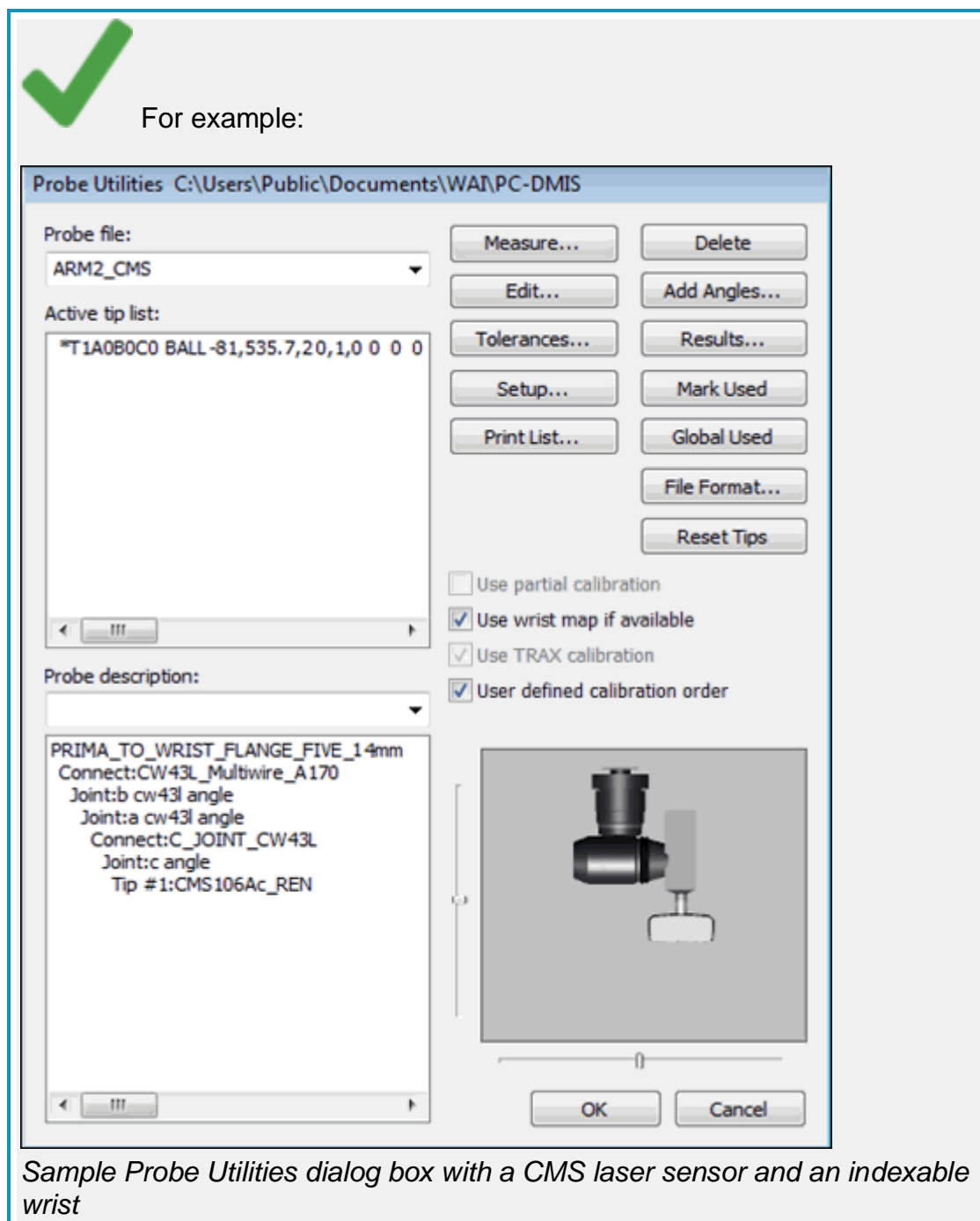
You need to recreate the LWM each time a component of the wrist changes (for example, when the CJoint changes). Also, refer to your hardware and vendor information for the appropriate times to map a wrist since this can change based on the device construction and manufacturer recommendations.

The following steps outline the procedure for mapping infinite wrist DCC CMS laser sensors:

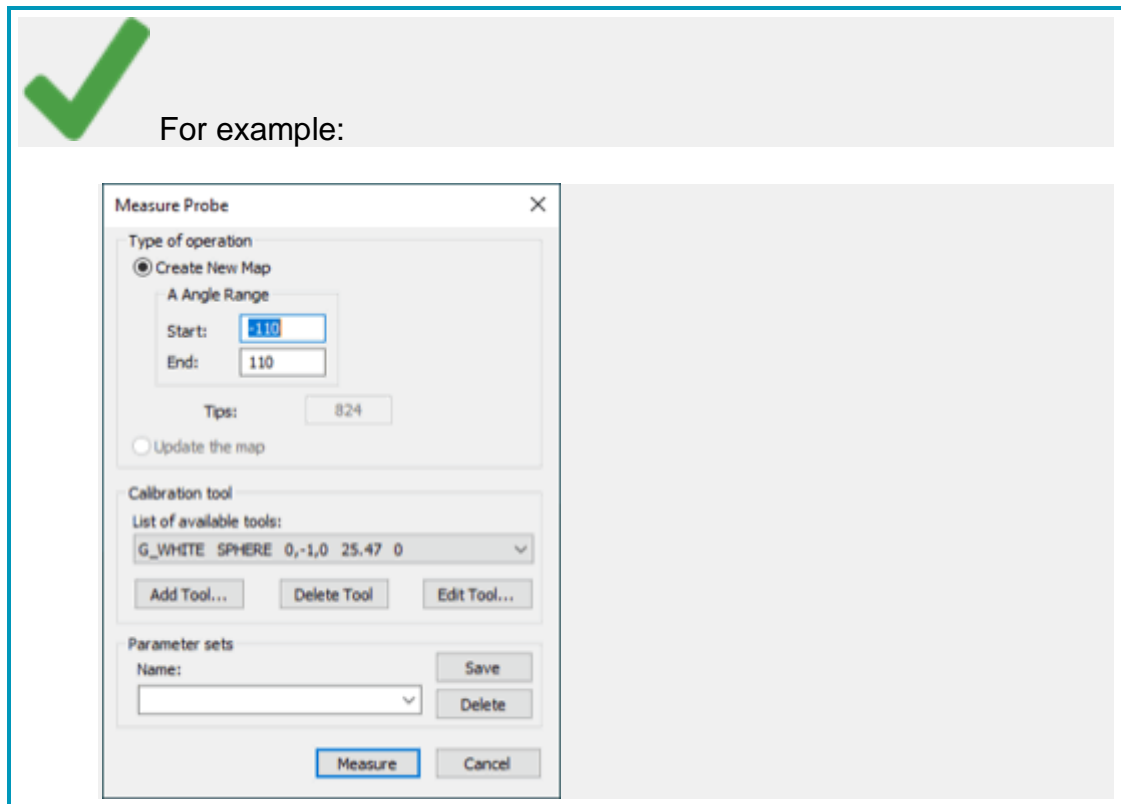
1. Define the sensor:

a. In the **Probe Utilities** dialog box, create a sensor as indicated below:

- Infinitely indexable wrist, such as the CW43L
- CJoint
- CMS laser sensor

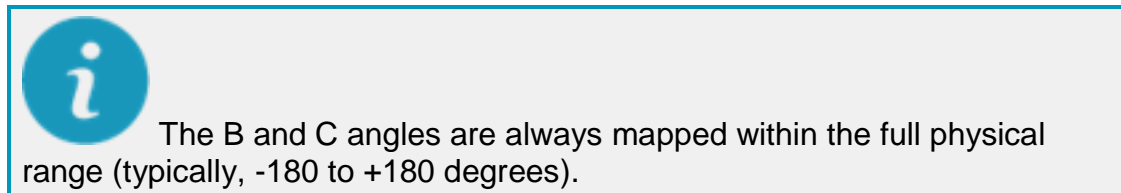


- b. Select the **Use wrist map if available** check box.
- c. Click **Measure** to display the **Measure Probe** dialog box.



2. Create the map:

- a. From the **Measure Probe** dialog box, select the **Create New Map** option.
- b. For **A Angle Range**, type the desired **Start** and **End** values. These values define a range of angles that form a virtual cone. The map qualifies any tip orientations that fit in this virtual cone.



The **Tips** box displays the total number of tips measured to create the map.

- c. Click **Measure**.
 - PC-DMIS measures five sensor orientations around the sphere tool.
 - PC-DMIS measures all of the tips in the mapping grid.

Updating an Existing Map

Once you create the map, you can recover the correct qualification for all of the tips whenever a geometrical or thermal parameter of the Sensor - Wrist system changes. For example, after the sensor experiences a physical collision, or when the room temperature changes.

To recover the correct qualification:

1. From the **Measure Probe** dialog box, select the **Update the map** option.
2. Click **Measure**. PC-DMIS starts to re-measure the same five sensor orientations around the sphere tool as it measured during the map creation process.

Resuming Map Creation

If the process of creating a map is interrupted (because the machine powered down, you were interrupted, or some math calibration errors occurred, for example), a **Resume** option appears in the **Measure Probe** dialog box. You can use this option to continue creating the map.

To resume the process of creating a map:

1. Select the **Resume** option in the **Measure Probe** dialog box. PC-DMIS automatically calculates which tips are still missing in the current map and creates a list of the missing tips to be measured.



You are not able to use the **Resume** option again until PC-DMIS successfully completes the mapping.

2. Click **Measure**. PC-DMIS starts to measure the tips necessary to complete the map.

Defining Parameter Sets for Map Creation

You can define a parameter set to create a map. You can also use the [AUTOCALIBRATE](#) command within a measurement routine to update a map.

To define a parameter set:

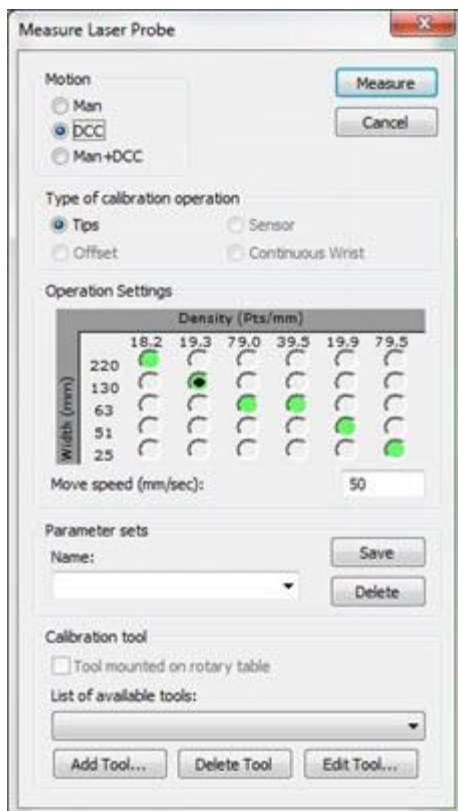
1. In the **Measure Probe** dialog box, select and type the desired values.
2. In the **Name** box, type a name for the parameter set.

3. Click **Save**.
4. To close the dialog box, click **Cancel**.

For more information about parameter sets and how to use the [AUTOCALIBRATE](#) command, see "Dual Arms with Wrists Calibration Example" in the PC-DMIS Core documentation.

Measure Laser Probe Options

The options on the **Measure Laser Probe** dialog box determine the procedure that the software uses for the laser sensor calibration. To access this dialog box, open the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**), and then click **Measure**.



Measure Laser Probe dialog box

Change the following options as needed or as directed in "Step 4: Calibrate the Laser Sensor".

Motion

- **Man** - This option requires that you manually position the arm in several different locations that bisect the calibration tool. This varies based on the sensor manufacturer. This option is the only available motion option for arm machines.
- **DCC** - Use this mode when the laser sensor has accurate offsets provided by the sensor manufacturer or if you have already run the calibrate "offset" routine. This moves the machine through a series of positions as recommended by the sensor manufacturer. You are not required to position the sensor manually for each tip that is calibrated.
- **Man+DCC** - This mode is similar to DCC mode, except that you must position the sensor over the sphere to begin the calibration sequence for each of the tips to calibrate. The software prompts you to position the sphere at the beginning of the calibration process.

Types of Calibration Operations



The options in this section are available based on the laser sensor. **Tips** works for all probes, and **Offset** is only for Perceptron sensors.

- **Tips** - Use this option to do a standard calibration for all marked tips for your laser sensor.
- **Offset** - Use this option to estimate the laser sensor offset for Perceptron laser sensor types. You only need offset calibrations to position the machine correctly to calibrate tips. If you skip this step, the probe may miss the sphere during tip calibration.



When you calibrate Perceptron sensors for the first time:

1. With the **Offset** option, calibrate a single tip.
2. With the **Tips** option, calibrate the first tip angle and any other tip angles.

For more details, see "Step 4: Calibrate the Laser Sensor".

Operation Settings

The items that appear in this area vary based on the laser sensor type.

- **Sensor states** - As described in the "Scan Zoom States (for CMS Sensors)" topic, these options appear only for CMS sensors. You can use these options to select a predefined sensor state. Each state has a specific combination of sensor frequency, data density, and Field of View (FOV) width.
- **Move Speed [%]** - Determines the percentage of the maximum machine speed that the software uses during the calibration process.

Parameter sets

Parameter sets allow you to create, save, and use saved sets for your laser sensor. This information is saved with the probe file, and it includes the settings for your laser sensor.

To create your own named parameter sets:

1. Modify any parameters on the **Measure Laser Probe** dialog box.
2. From the **Parameter Sets** area, in the **Name** box, type a name for the new parameter set, and click **Save**. To delete a saved parameter set, select it, and click **Delete**.

Calibration Tool

Select the appropriate calibration tool. If this is your first calibration, you need to click **Add Tool** to first define the calibration tool. For specific information on defining a qualification tool, see the "Defining Hardware" chapter of the PC-DMIS Core documentation.



Make sure you use the spherical qualification tool that comes with your laser sensor. The surface characteristics of this tool are designed for optimal scanning results. If you use a tool made by another manufacturer, it may produce inaccurate data.

Measure CWS/WLS Laser Probe

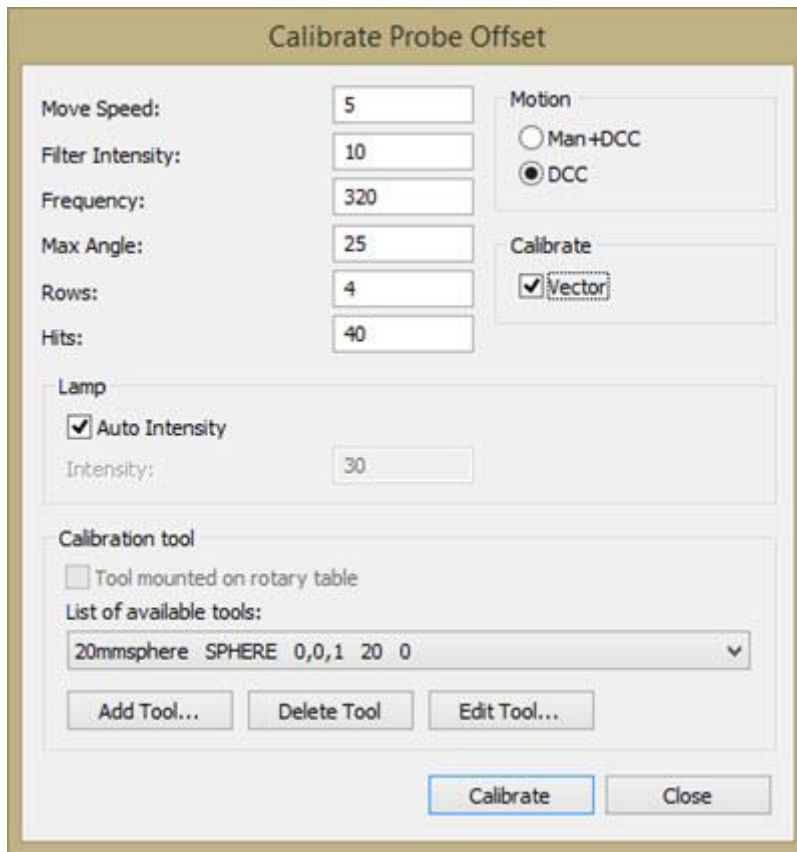
The options on the **Calibrate Probe Offset** dialog box determine the procedure that the software uses for the calibration. To access this dialog box, define your probe in the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**) and then click **Measure**.

Requirements Prior to Calibration

To begin the calibrating process, you must define a qualification tool. The only tool type supported is a sphere. From the **List of available tools**, select a currently-defined qualification tool.

- To define a new qualification tool to add to the list of available tools, click **Add Tool**.
- To change the configuration of the currently-defined qualification tool, click **Edit Tool**.
- To delete the currently-defined qualification tool, click **Delete Tool**.

Click the **Measure** button to display the **Calibrate Probe Offset** dialog box.



The image shows the 'Calibrate Probe Offset' dialog box. It contains several input fields and checkboxes. The 'Move Speed' is set to 5, 'Filter Intensity' to 10, 'Frequency' to 320, 'Max Angle' to 25, 'Rows' to 4, and 'Hits' to 40. Under 'Motion', 'DCC' is selected. Under 'Calibrate', 'Vector' is checked. In the 'Lamp' section, 'Auto Intensity' is checked and 'Intensity' is set to 30. In the 'Calibration tool' section, 'Tool mounted on rotary table' is unchecked, and the 'List of available tools' dropdown shows '20mmsphere SPHERE 0,0,1 20 0'. At the bottom, there are buttons for 'Add Tool...', 'Delete Tool', 'Edit Tool...', 'Calibrate', and 'Close'.

The settings on this dialog box are:

Move Speed: Sets the percentage of the maximum machine speed that the software uses during the calibration process.

Filter Intensity: Sets the CWS filter intensity. For details, see "CWS Parameters" in the PC-DMIS Vision documentation.

Frequency: Sets the CWS frequency. For details, see "CWS Parameters" in the PC-DMIS Vision documentation.

Max Angle: Sets the maximum angle away from the sphere pole or zero angle point for the pattern of points. The best angle depends on the CWS probe being used. Different probe heads have different maximum measurement angles.

Rows: The number of rows in the pattern of measurement points.

Hits: The number of hits in the pattern of measurement points.

Lamp Auto Intensity: Sets the lamp intensity to automatic. For details, see "CWS Parameters" in the PC-DMIS Vision documentation.

Lamp Intensity: Sets the lamp intensity when not in automatic mode. For details, see "CWS Parameters" in the PC-DMIS Vision documentation.

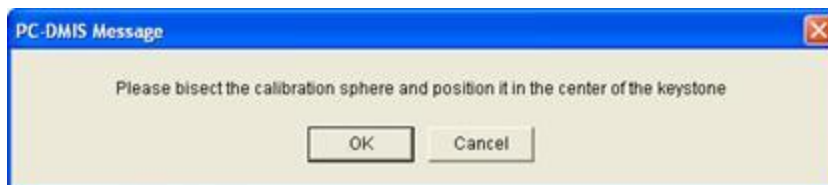
Motion Man+DCC: Requires a manual point at the beginning of the calibration. PC-DMIS executes all subsequent points in DCC mode.

Motion DCC: Measures the sphere automatically in DCC mode. Ensure you position the probe with the appropriate clearance for any wrist rotation and movement to the sphere measurement points.

Calibrate Vector: Enables the vector calibration measurements. The software measures the sphere two additional times after the tip offset calibration in order to calculate the vector of the CWS probe.

Manually Bisecting the Calibration Sphere

When you use either the MAN (Manual) or MAN + DCC Motion option, you will be required to manually bisect the qualification sphere. This is also necessary if you have moved the sphere or do not know the location of the sphere. The calibration procedure will prompt you when you need to move the machine.

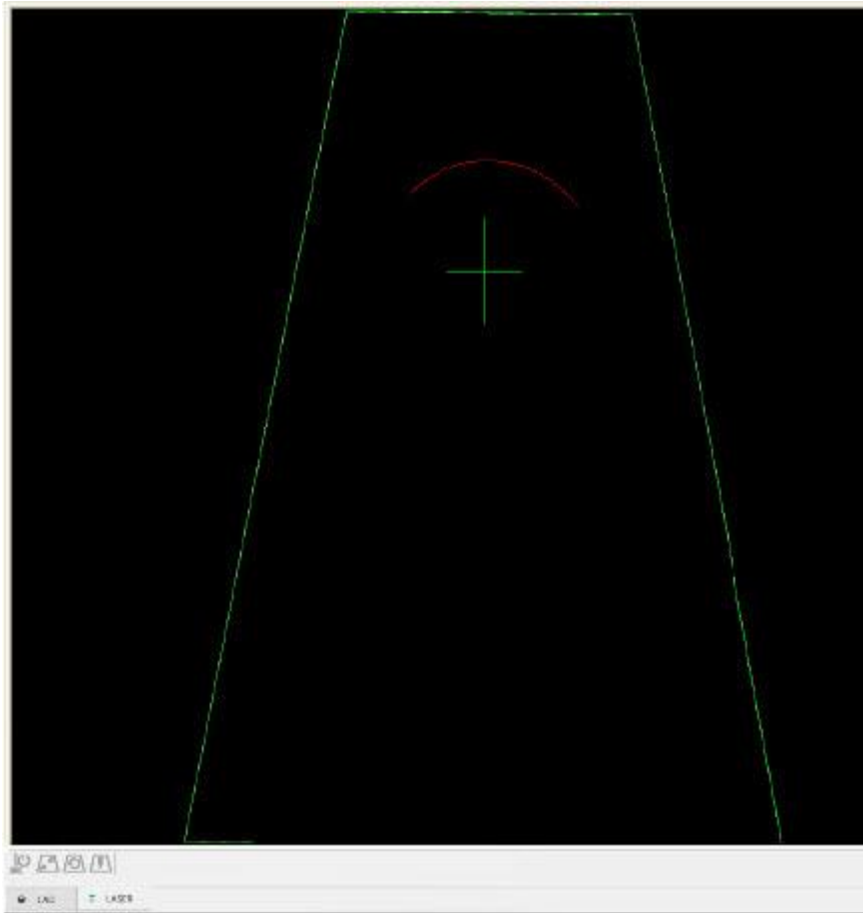


PC-DMIS Message

To manually bisect the sphere:

1. Leave the PC-DMIS Message open.
2. Switch to the **Laser** tab on the main Graphic Display window.

3. Click the **Start/Stop** button. This turns on the laser. A flashing red arc appears in the graphic area of the **Laser** tab and a green crosshair. The red arc is where the laser hits the calibration sphere.
4. Center the cross hair inside the circular region formed by the arc by moving the machine with the jog box. The red arc moves as you move the machine. If you imagine that the flashing arc indicates the edge of a circle, the center point of this imaginary circle should optically align with the center of the cross hair.



Aligning the arc

5. Once you have aligned the arc, click the **On/Off** button again. This turns off the laser.
6. Click **OK** in the PC-DMIS Message to accept the change you made of aligning the arc. PC-DMIS stays in Execute mode, and the laser sensor moves through a series of defined positions used to calibrate your tip.
7. At each position, the laser beam hits the sphere in a stripe and the laser sensor collects the data from that stripe. The collected data and the corresponding machine position determine the sensor's mounting orientation on the machine.

8. Once execution finishes, PC-DMIS returns to learn mode and shows the **Probe Utilities** dialog box.

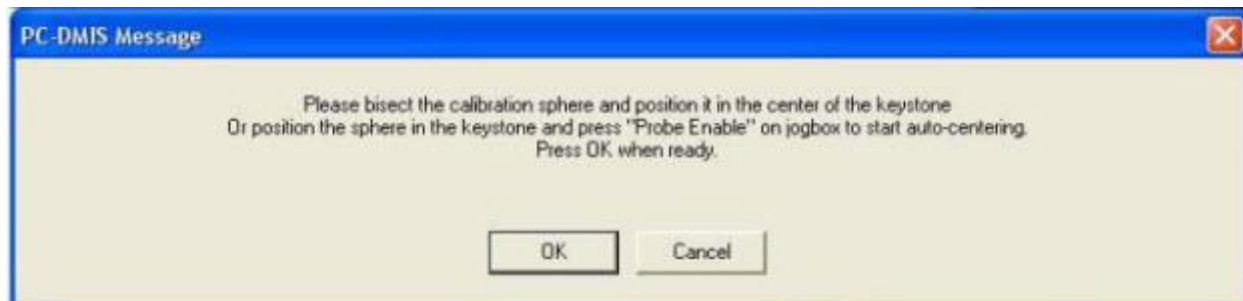
CMS Automatic Self-Centering of Tool Sphere

The CMS laser sensor provides automatic self-centering (bisecting) of the calibration tool sphere during calibration if you answer **Yes** to the question, "Has the sphere moved?". From the Graphic Display window, click the **Laser** tab. You can drive the laser sensor to the center of the sphere.

You have two possibilities at this point:

- You can manually bisect the sphere and bring it in the center of the Keystone and then press **OK** to start the laser calibration.
- Display a portion of the calibration sphere in the Laser View and then press the **Probe Enable** button to automatically center the sphere. When completed, press the **OK** button to complete the laser calibration.

The PC-DMIS Message dialog box appears as soon as PC-DMIS determines that the calibration sphere has been moved.



Follow the instructions as described in the message box.

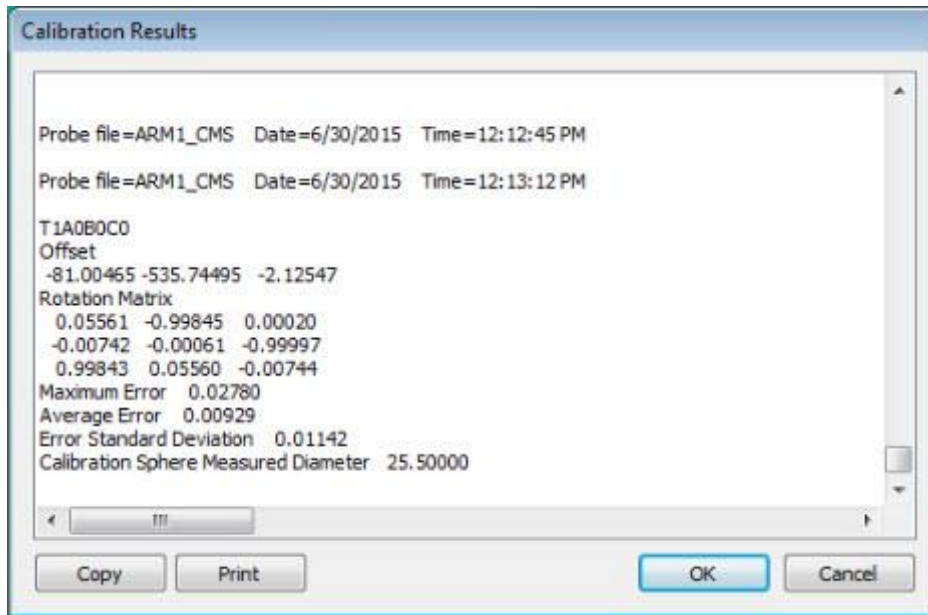
Press the **OK** button when done.



For convenience, during the auto-centering procedure, the laser sensor alignment stripe appears in yellow.

Step 5: Check the Calibration Results

From the **Probe Utilities** dialog box, click the **Results** button to show the **Calibration Results** dialog box.



Calibration Results dialog box

PC-DMIS records several results from the calibration in this dialog box. Look at the maximum, average, and standard deviation values.

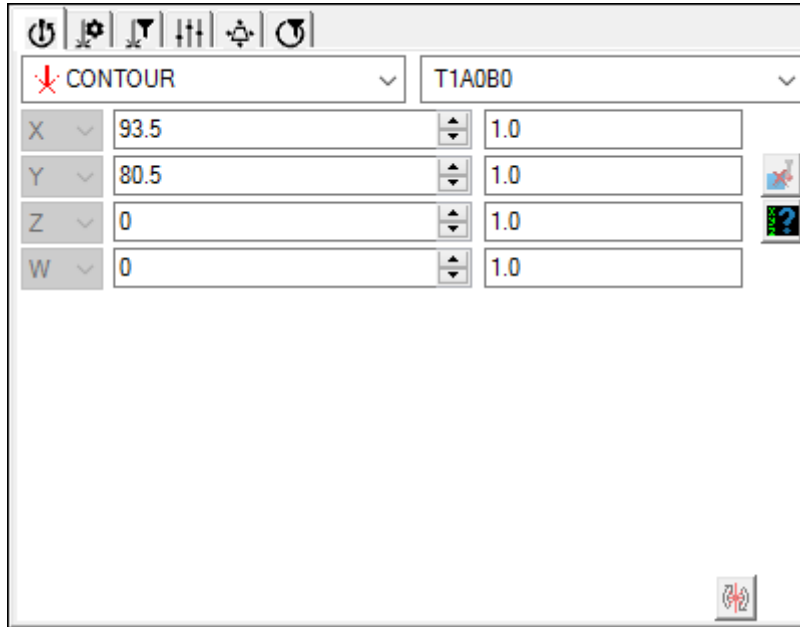
The maximum should be somewhere between 20 to 100 microns. The average and the standard deviation should be around 20 microns.

If the values look correct, click the **OK** button to close the **Calibration Results** dialog box. You have these options:

- To paste the report into a different application (such as Microsoft Word, Notepad, or other application), click **Copy**, open the desired application, and press Ctrl + V to paste it.
- To send the report to a printer, click **Print**.

This finishes the setup and calibration process for your laser sensor. You can now use all the laser-related functionality.

Using the Probe Toolbox in PC-DMIS Laser



Probe Toolbox with the laser sensor-related tabs

The **View | Probe Toolbox** menu option displays the Probe Toolbox. The Probe Toolbox contains various laser sensor parameters that you can use to acquire the data points that a measurement routine needs.



Your LMS license or portlock must contain the Laser option, and you must be working with a supported laser sensor to access the laser-related tabs in the Probe Toolbox.

The Probe Toolbox contains the laser parameters on these tabs:

For Portable Configurations



Laser Scan Properties *^+!



Laser Filtering Properties *+!



Laser Pixel Locator Properties *



Feature Extraction ^!

For CMM Configurations



Position Probe



Laser Scan Properties



Laser Filtering Properties



Laser Pixel Locator CG Properties



Laser Clipping Region Properties



Feature Extraction



Laser AF Multiple Creation



The list above shows all possible Probe Toolbox tabs. The tabs that are available depend on the sensor that you have on your system. If the capabilities for a tab do not apply to your specific sensor, then that tab is unavailable.

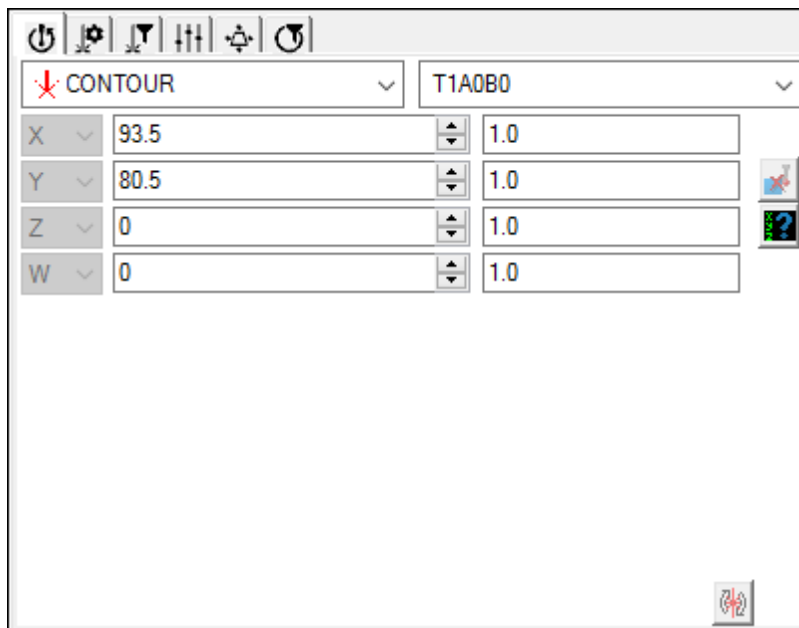
* For Perceptron probes, these tabs are visible when you close the **Auto Feature** dialog box.

^ For Perceptron probes, these tabs are visible when you open the **Auto Feature** dialog box.

+ For CMS probes, these tabs are visible when you close the **Auto Feature** dialog box.

! For CMS probes, these tabs are visible when you open the **Auto Feature** dialog box.

Laser Probe Toolbox: Position Probe tab



Probe Toolbox - Position Probe tab

The **Position Probe** tab of the Probe Toolbox (**View | Other Windows | Probe Toolbox**) allows you to select the current probe file and tip and define the current probe location in the active alignment coordinates. You can double click the X, Y, and Z values to edit them.



Warning: When you edit the current probe location, the machine moves to the new coordinate without notice. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

If you don't see any information in the **Probes** and **Probe Tips** lists of the Probe Toolbox, you need to first define a probe. For information on how to define a probe, see the "Defining Hardware" chapter in the PC-DMIS Core documentation.

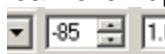


While you can use this tab with all probe types (contact, laser, and optical), this document only covers PC-DMIS Laser related items. For information about the toolbox as it relates to probes in general, see "Using the Probe Toolbox" in the "Using Other Windows, Editors, and Tools" chapter in the PC-DMIS Core documentation.

To Position Your Laser Sensor

You can use the **Position Probe** tab of the Probe Toolbox (**View | Other Windows | Probe Toolbox**) to position your laser sensor. This tab contains sets of values in two columns.

Left Column: The X, Y, Z values. They show the laser sensor's current position. You can click up and down arrows to change the value in the **XYZ Probe Position**

box  for an axis. This moves your laser sensor in real-time by the increment value on the right.

Right Column: The increment values. These specify how much to increase or decrease the XYZ Probe Position box for each axis when you click the up and down arrows on the left column.

Alternately, you can type the XYZ values in the left column, and press Enter to move your laser sensor to a predefined position.

Controls for Position Probe Tab

The toggle buttons on **Position Probe** tab of the Probe Toolbox (**View | Other Windows | Probe Toolbox**) are:



Probe Readouts - This toggle button shows or hides the Probe Readouts window. You can easily resize or relocate this window. Most of the information on

the Probe Readouts window is the same for all probe types. For details, see "Using the Probe Readouts Window" in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.

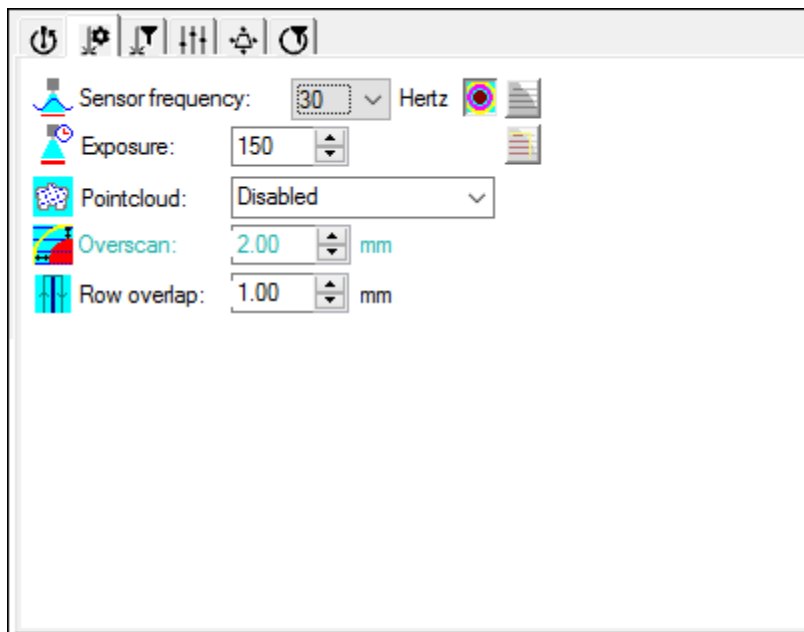


Laser On/Off - This toggle button turns the laser on or off. It is only available for laser probes.



Initialize Probe - This button starts or initializes the laser. You can't do anything with the laser until it is initialized. This takes about 15 seconds. (This button appears on this tab for DCC configurations.)

Laser Probe Toolbox: Laser Scan Properties tab

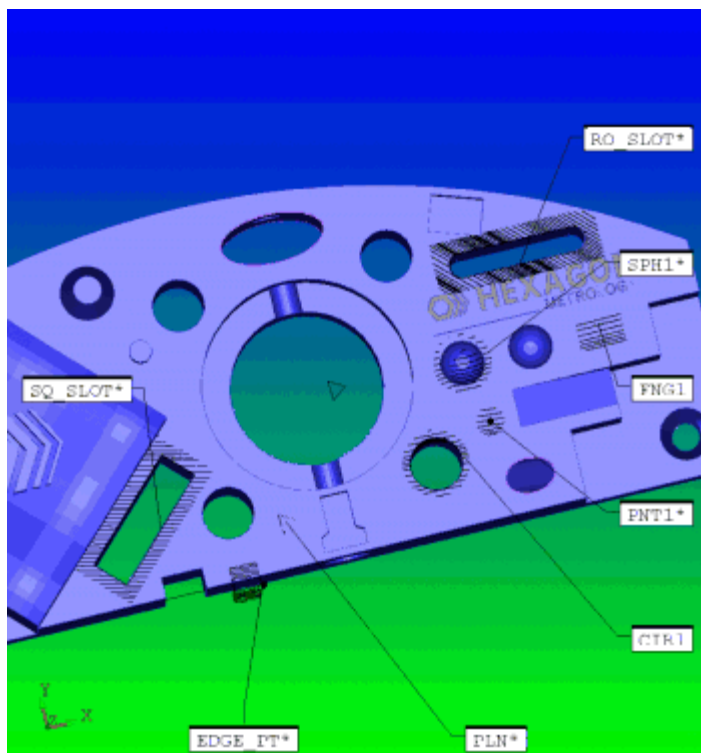


Probe Toolbox - Laser Scan Properties tab


The **Laser Scan Properties** tab defines how PC-DMIS acquires data from the scan, and whether or not scan lines and feature visualizations appear in the Graphic Display window.





Show/Hide Stripes - This button toggles the display of the laser stripes on the part model. Click this button to allow the laser scan stripes to appear in real time in the Graphic Display window. PC-DMIS limits how the stripes appear in the Graphic Display window to the distance of the feature nominal plus the **Overscan** value. The software uses the **Overscan** value to control the amount of clipping and the visibility of the stripe. The graphic below gives an example of how these stripes appear.

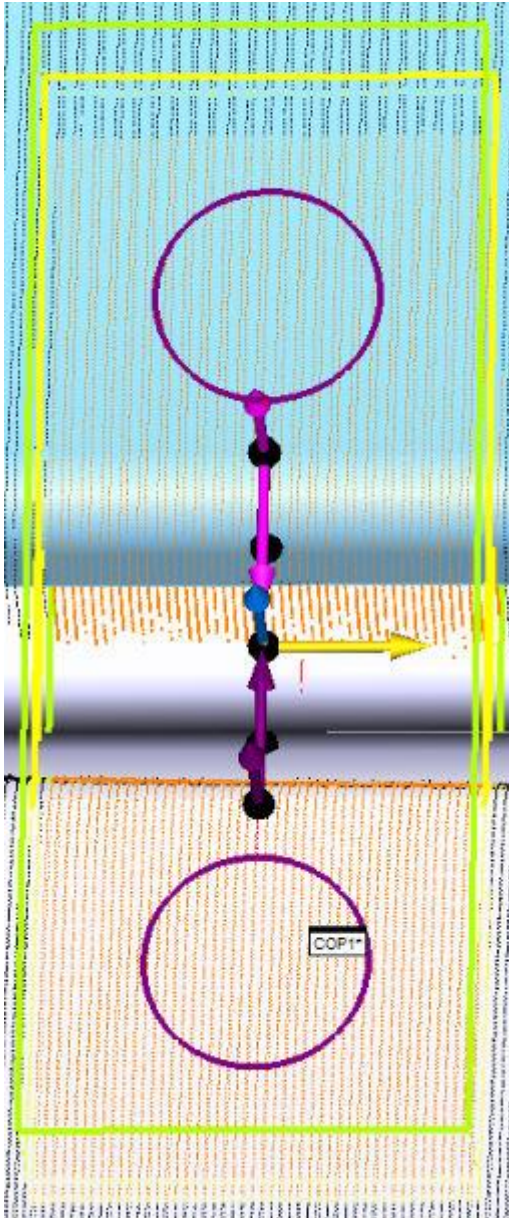


Scan features showing stripes


 **Sound ON/OFF** - This button turns the sound on or off. See "Using Sound Events".


 **Visualization Tools ON/OFF** - This button toggles the display of the colored visualization tools. For more information, see "Understanding the Visualization Tools".

 **Show/Hide Segregated Points** - This button toggles the *display of those points* that the software passes to the feature extractor engine based on the current settings.

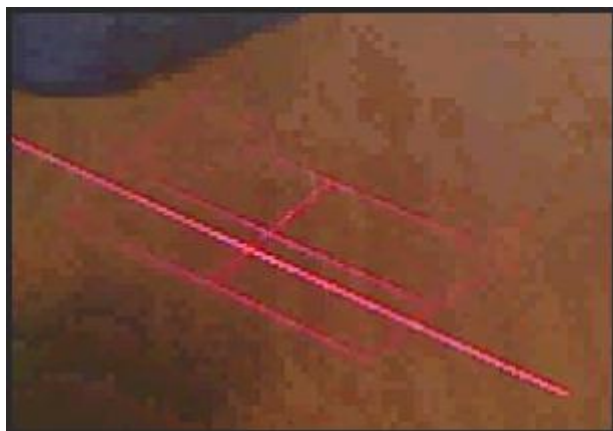


Showing segregated points inside a sample Flush and Gap feature


 **Initialize Probe** - This button starts or initializes the laser. You can't do anything with the laser until you initialize it. This takes about 15 seconds. This button appears on this tab for Portable configurations.

 **Projector:** This button is only available for V5 Perceptron probes on manual arms. Click this button to turn on a projected *grid of red light* that shines on the part. This acts like the cross hairs on a target. As you move the probe toward or away from the part, the probe's laser scan line moves through this target. For optimal results, the laser's scan line should line up with the center line of this target. This essentially serves the same purpose as the scan line indicator, which

helps you to keep the probe at the optimal height when you measure the part. Since this only functions in manual applications, PC-DMIS disables this icon if you use the Probe Toolbox within the **Auto Feature** dialog box.

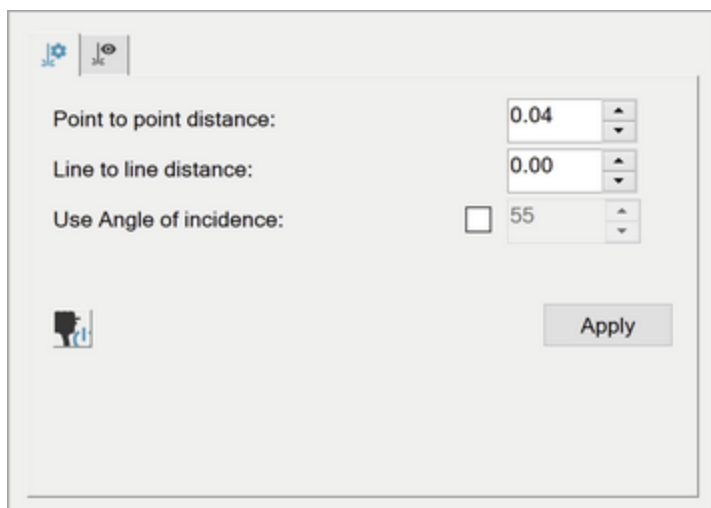


This actual picture of the Projector shows the rectangular grid-like projection of light. The brighter horizontal line is the laser's scan line.

 **AutoZoom ON/OFF** - This button turns the laser AutoZoom functionality on or off. Whenever you start the scan, AutoZoom dynamically pans, zooms, rotates, and sizes the view containing the laser data in the Graphic Display window to show the incoming data.

Laser Scan Properties for a Leica LAS, LAS-XL and T-Scan

For a portable Leica LAS, LAS-XL and T-Scan probe, the **Laser Scan Properties** tab contains these options:




Probe Toolbox - Tracker Laser Scan Properties tab

Point to point distance - This option specifies the distance between two consecutive points in a scan line. The allowed values are between 0.035 mm and 10 mm when you use the up and down arrows. This option is only available for T-Scan scanners.

Line to line distance - This option specifies the distance between two consecutive scan lines. The allowed values are between 0 mm and 50 mm when you use the up and down arrows. This option is only available for T-Scan scanners.

Use Angle of incidence - This option specifies which maximum-allowed angle the software uses for scanning. This value helps avoid bad conditions while scanning (surface reflections, geometry, and so on). This angle is the angle between a ray and the surface normal vector. Allowed values are between 0 and 80 degrees when you use the up and down arrows. This option is only available for T-Scan scanners.

- If you select the check box to the left of the box, PC-DMIS sends the angle value in the field.
- If you clear the check box, PC-DMIS sends a 90-degree angle to the dispatch interface. A value of 90 degrees is the same as clearing the check box.

Connect to Scanner -  This button toggles the tracker scanner connection to the scanning application on and off. For the LAS and LAS-XL scanners, the scanning application is RDS; for the T-Scan scanner, the scanning application is T-Collect.



When the **Connect to Scanner** button is on, PC-DMIS disables all other buttons on the **Tracker Operation** toolbar.

When you re-execute a Tracker program with the scanner, you should not use the **Connect to Scanner** button. In this case, PC-DMIS automatically connects to the scanner application.

For details, see "Tracker Toolbars" in the PC-DMIS Portable documentation.

Apply - This button applies the values defined in this tab without stopping the scanner.



You can override the limitations with the up and down arrows, or enter a value into any of the boxes. However, your machine rejects invalid values and forces them to a valid number.

Other Properties

Sensor Frequency

This parameter controls the internal sensor frequency of the probe. The value that appears is the sensor pulses per second. For sensors with variable frequency capabilities, the higher the frequency, the more data you get. It is important to understand that more data is not always better. With variable frequency scanners, you should use a frequency in the middle of the supported range. This is a good balance between speed and accuracy.

Row Overlap

If the feature or patch scan is larger than the width of the scan line, multiple passes of the probe are taken. In that case, this parameter controls how far each pass overlaps the previous pass. The default value is 1.0 mm.

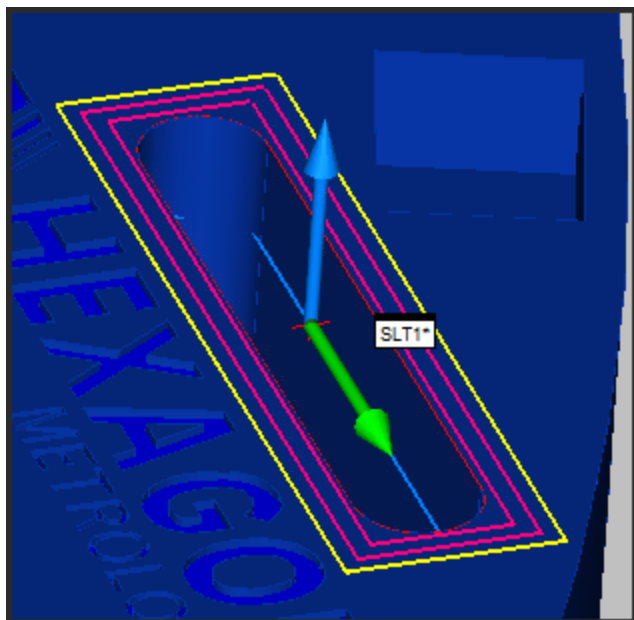
Overscan

For DCC systems, this parameter controls how far beyond the nominal feature's dimensions the probe scans along both the major and minor axis of the feature. The default value is 2.0 mm. If you measure features whose actual location may vary significantly from their theoretical values, you need to increase this value to ensure that PC-DMIS measures the entire feature.

In version 2010 and higher, the **Overscan** value no longer does any sort of clipping of the data. The new **Feature Based Clipping** area in the **Feature Extraction** tab now handles clipping. See the "Feature Based Clipping Parameters" topic.

For a DCC laser Cylinder or Cone feature, the **Overscan** value should be a negative value.

For a laser Stud feature (see the Laser Cylinder for stud information), the **Overscan** value should be a positive number.



Sample Slot feature that shows the overscan in yellow

Exposure

This parameter controls the exposure of the sensor. The default value of 150 works well for most parts, but for parts that absorb a lot of light (such as a black anodized surface), you may need to increase the value. If you're using a sensor that supports the Gray Sum pixel locator type, PC-DMIS sets the exposure value to a material-specific value when you choose a material type. You can find this in the **Material** list on the **Laser Pixel CG Locator Properties** tab of the Probe Toolbox.

The following table shows the available minimum and maximum exposure values for the supported Perceptron probes:

	Perceptron Laser Probes		
Normalized Exposure	V4i (Portable)	V4ix (DCC)	V5
Minimum Value:	32	1	1
Maximum Value:	627	627	1716
Default Value:	150	150	

If you set this to an inappropriate value, it may result in less-accurate measurements.



For Perceptron sensors, you can use the **AutoExposure** button on the **Laser** tab to calculate the best exposure value for you. In addition, if you set the `AutoExposeWithLiveView` registry entry to TRUE, PC-DMIS automatically sets the exposure value in the Probe Toolbox to the best value every time you start the Laser View.

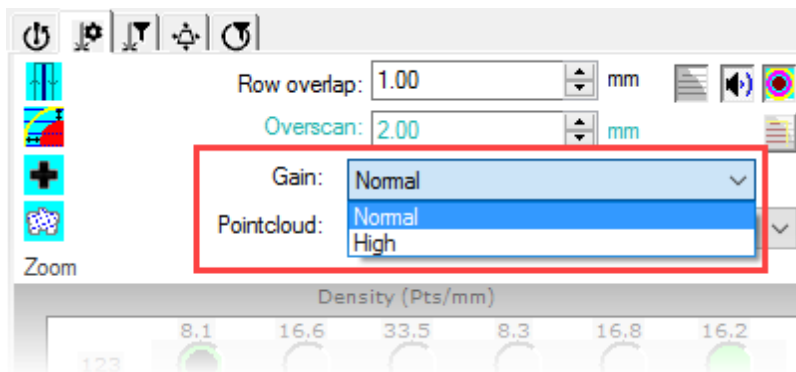
Point Cloud

This parameter defines the COP command from which the auto feature will be extracted. If "disabled" is selected, then the data from the scan will be stored internally by PC-DMIS. You can delete internal data if needed by using the **Operation | Laser Autofeatures** submenu. See "Clearing Auto Feature Scan Data".



The "disabled" option is only used with DCC laser scans.

Gain (for CMS Sensors)



Gain list

CMS sensors provide you with an additional list called **Gain** in the **Laser Scan Properties** tab of the Probe Toolbox.

- CMS106 and CMS108 support **NORMAL** and **HIGH**.
- HP-L-20.8 supports **NORMAL**, **HIGH**, and **XHIGH**.
- HP-L-5.8 supports **1**, **2**, **3**, **4**, and **5**.

This list lets you choose among these sensitivity modes:


Sensitivity Modes

NORMAL sensitivity - You should use this default sensor mode on most normal parts. This mode sets the **QUALITY FILTER** toggle field in the Edit window in Command mode to **ON**, so that the Edit window shows the associated fields. This sensitivity mode also hides the **Quality Filter** icon.

HIGH sensitivity - The **HIGH** sensitivity mode becomes available for selection if you run PC-DMIS in Online mode. You should only use the **HIGH** sensitivity mode if you are scanning a part comprised of a troublesome material where the **NORMAL** sensitivity mode returns poor data. For example, a part that absorbs too much light because it has glossy, dark, or black surfaces may require this type of mode. However, note that scanning a normal part in **HIGH** sensitivity mode may yield noisy data.

XHIGH (extra high) sensitivity - **XHIGH** is similar to **HIGH**. It provides an option for scanning materials that may be even more troublesome than those that can be handled using the **HIGH** option. If you don't get good results using **HIGH**, you can try using the **XHIGH** option. However, as with the **HIGH** option, if you scan a normal part in **XHIGH** mode, it may return even more noisy data.

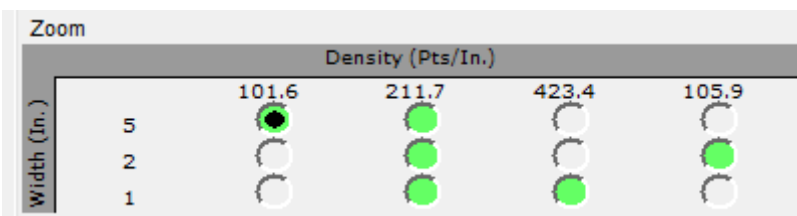
In the **HIGH** and **XHIGH** modes, a **Quality Filter** icon appears next to the **Gain** list:

Quality Filter  - If you enable this mode, PC-DMIS filters low quality points including double reflections, poor quality data on edges, and outliers. If enabled, it sets the **QUALITY FILTER** toggle field to **ON** in the Edit window in Command mode to show the associated fields in the Edit window.

1, 2, 3, 4, and 5 sensitivities - These sensitivities are available for the HP-L-5.8 sensor.

Scan Zoom States (for CMS Sensors)

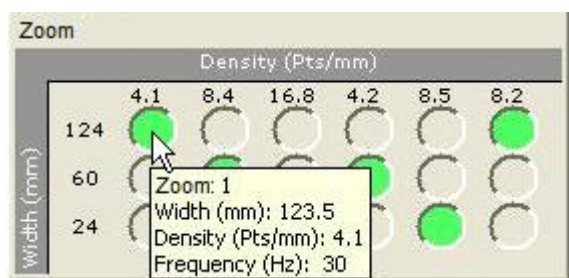
CMS sensors provide you with an additional area called **Zoom**, located at the bottom of the **Laser Scan Properties** tab of the Probe Toolbox. This area tells the sensor to work in predefined zoom states, with each state comprised of a specific combination of sensor frequency, data density, and Field of View (FOV) width.



Sample Zoom area

This area displays a grid of option buttons arranged in columns and rows. Across the top, the "columns" show the data density. Along the side, the "rows" list the FOV width. You can only select proper combinations which are the option buttons with a green background. The software grays out the improper combinations.

You can hover your mouse pointer over any valid option button to displays a yellow tooltip that details its scan mode information.

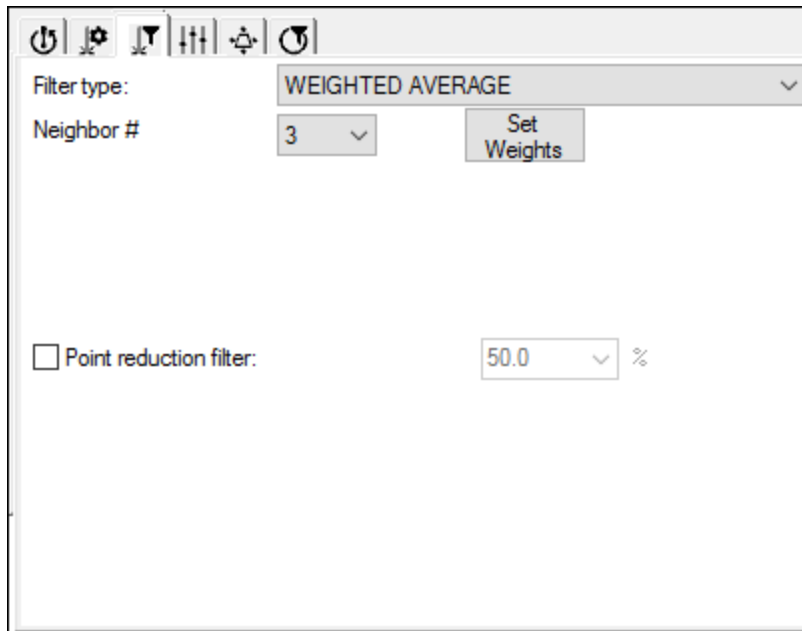


Sample tooltip under the mouse

Available Scan Zoom States for HP-L-20.8

		Density (Pts/mm)					
Width (mm)		18.2	19.2	78.9	39.5	19.8	79.5
	220	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
	130	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
	63	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
	51	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
	25	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>

Laser Probe Toolbox: Laser Filtering Properties tab



Probe Toolbox - Laser Filtering Properties tab

The **Filtering** tab is useful when you want to filter the data as PC-DMIS collects it.



Scanning methods with a portable device using a Perceptron laser differ from DCC machines. If you open the **Auto Feature** dialog box and are using a portable device with a Perceptron laser, the **Laser Pixel CG Locator Properties** tab is hidden.

The following filtering options are available from the list.

Filter Type: Only Available for Perceptron Sensors

- **None** - Filtering does not take place if you select **None**. This is the default setting.
- Long Line
- Median
- Weighted Average

Filter Type: Only Available for CMS Sensors

- Stripe

Density Type: Only Available for Perceptron Sensors

- **None** - Density filtering does not take place if you select **None**. This is the default setting.
- Intelligent Density Management (Contour V5 only)



In PC-DMIS 2010 MR3 and later, the **Point** filter type for CMS and **Column Sampling Rate** for Perceptron have been combined in a generic **Point Reduction Filter** check box, which is visible on all filter types regardless of the laser sensor used.

Filter Type: None

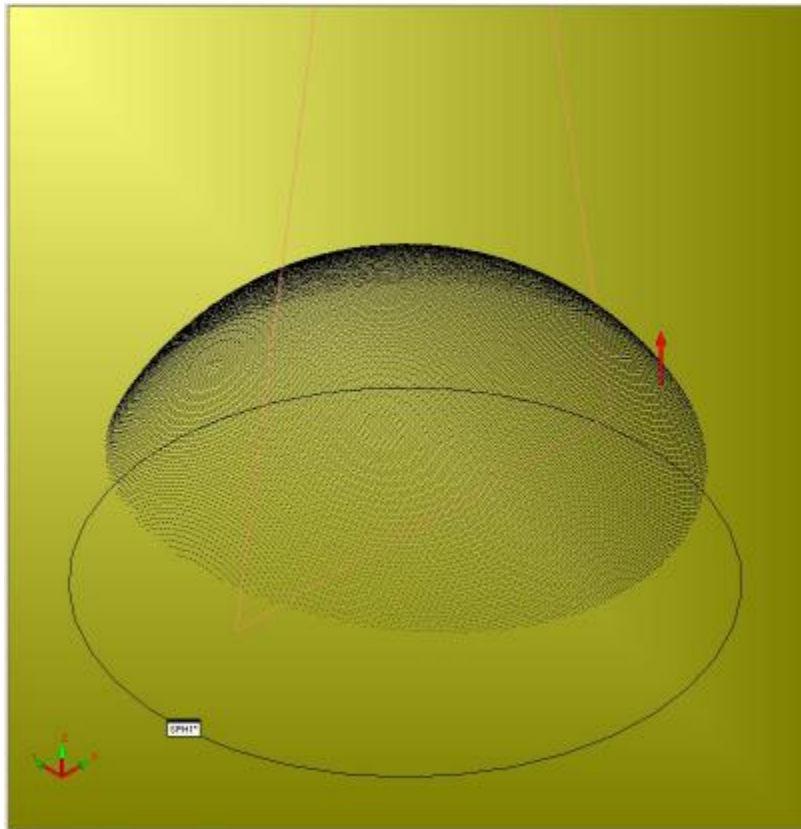
The screenshot shows a software interface for configuring filter settings. At the top, there is a toolbar with icons for power, settings, filter, and other functions. Below the toolbar, the 'Filter type' dropdown menu is set to 'NONE'. The 'Density type' dropdown menu is also set to 'NONE'. At the bottom, there is a checkbox labeled 'Point reduction filter' which is checked. To the right of the checkbox is a dropdown menu showing '75.0' and a percentage sign.

None filter type

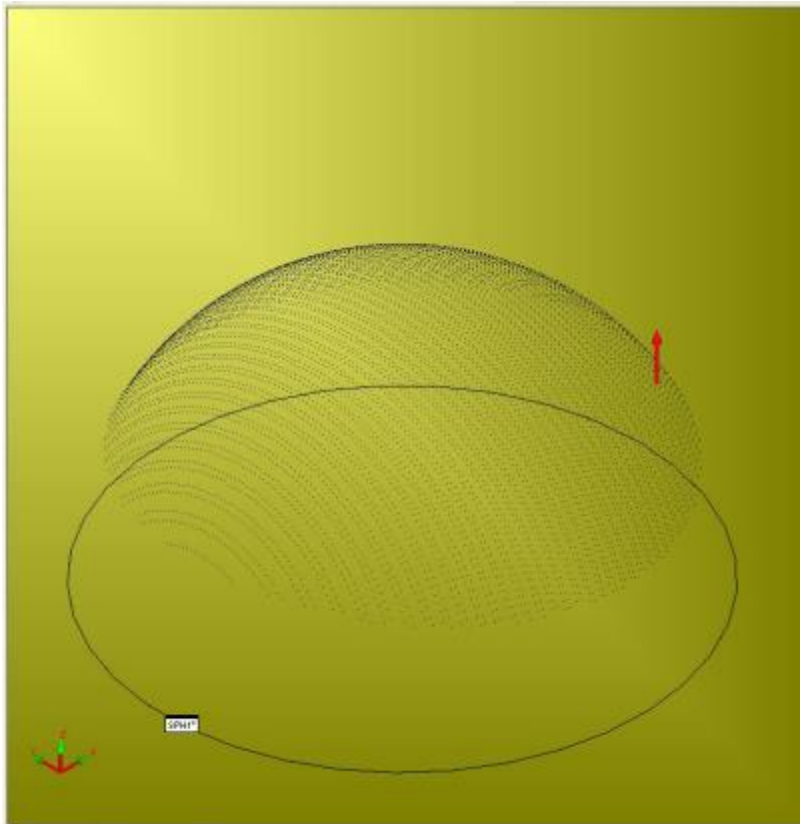
No initial filter is made. However, you do have the option to filter by point reduction.

Point Reduction Filter: This check box determines whether PC-DMIS filters points along the scan line. If the check box is marked, you can select the desired percentage of total points to filter. If the check box is cleared, the complete data set is acquired without any filtering.

Example of Point Filtering Disabled



Example of Point Filtering of 50%



Filter Type: Long Line



This filter type is only available for Perceptron sensors. It is usually used only for measuring spheres and some cylinders.

Filter type: LONG LINE

Above: 5000 Right: 5000

Below: 5000

Density type: NONE

☒ Point reduction filter: 75.0 %

Long Line Filter Type

The **Long Line** filter finds the longest continuous line or stripe of data in the image and rejects the rest of the data. PC-DMIS enforces the use of the Long Line filter during calibration. The laser stripe may be broken up due to the geometry of the measured part. This filter finds the longest unbroken line, and is often used with sphere measurements. PC-DMIS considers a section of the stripe continuous based on these parameters:

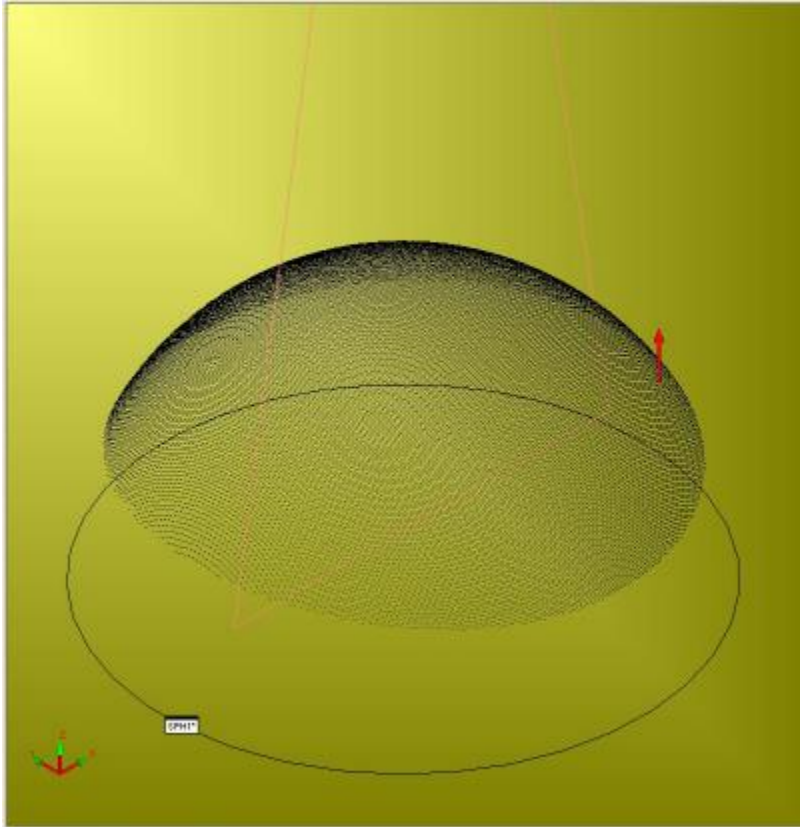
Above: This value determines the number of pixels in the image that the next pixel can rise above and still allow as part of a continuous line. The value indicates the number of milli-pixels above the current pixel that the filter will use.

Below: This value determines the number of pixels in the image that the next pixel can fall below and still allow as part of a continuous line. The value indicates the number of milli-pixels below the current pixel that the filter will use.

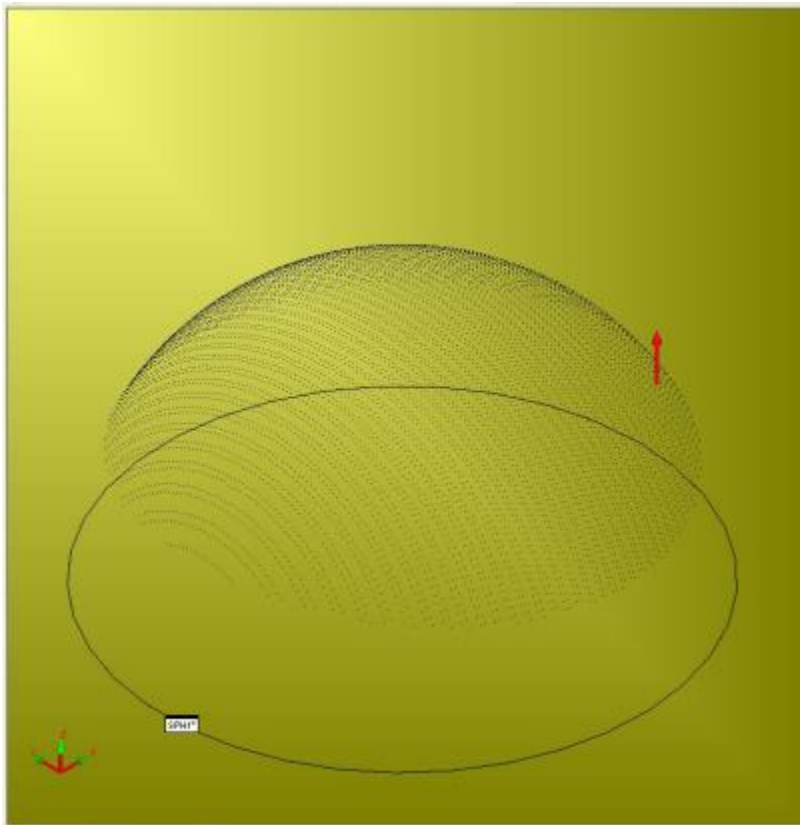
Right: This value determines the number of allowed, missing milli-pixels to the right of the current pixel and still allow as part of a continuous line.

Point Reduction Filter: This check box determines whether or not PC-DMIS filters points along the scan line. If you select the check box, you can select the desired percentage of total points to filter. If you do not select the check box, PC-DMIS acquires the complete data set without any filtering.

Example of Point Filtering Disabled



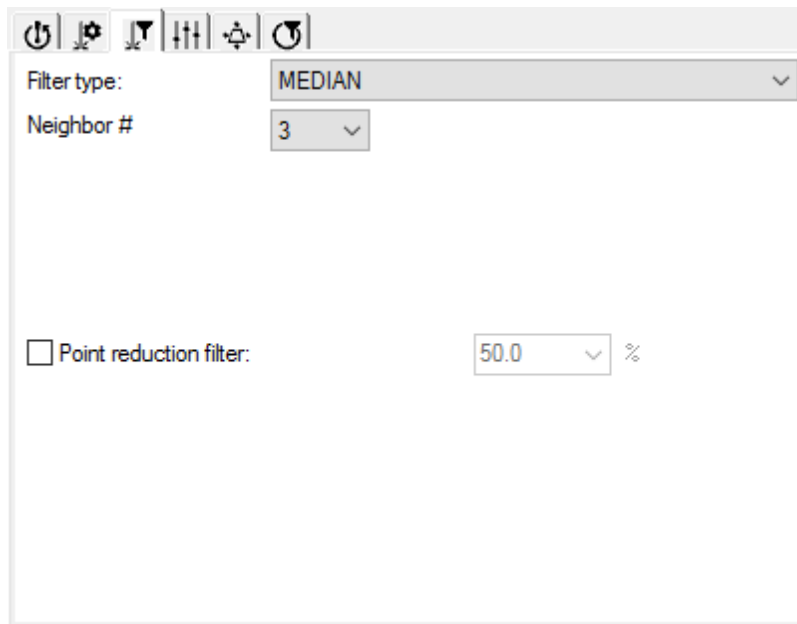
Example of Point Filtering of 50%



Filter Type: Median



This type is only available for Perceptron sensors.



Median filter type

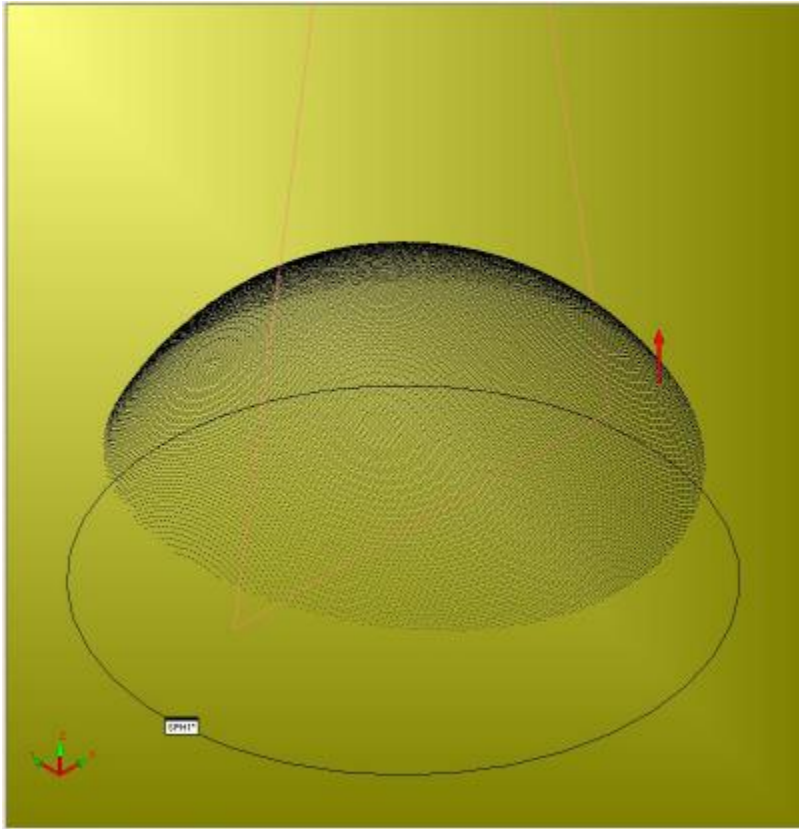
The **Median** filter smooths laser stripe data by computing a new location for each pixel. For each pixel in the stripe, the median filter takes the nearest neighboring pixels, computes the median, and uses the median for the new location of the pixel.

Neighbor #: This value determines the number of total neighboring pixels that the software considers when PC-DMIS calculates a new location of any given pixel in a single stripe.

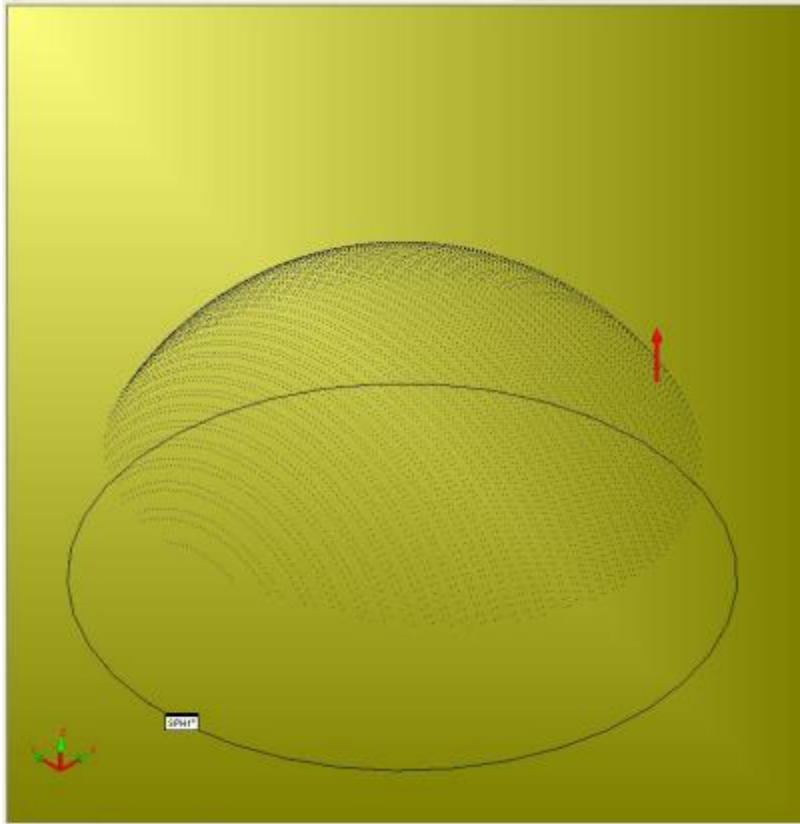
For example, if the number of neighbors is 9, then for each pixel in the stripe, the filter takes four data points to the left and four data points to the right (for a total of 9 pixels, including the current one). It then computes the median and uses it for the location of the current pixel.

Point Reduction Filter: This check box determines whether PC-DMIS filters points along the scan line. If the check box is marked, you can select the desired percentage of total points to filter. If the check box is cleared, the complete data set is acquired without any filtering.

Example of Point Filtering Disabled



Example of Point Filtering of 50%



Filter Type: Weighted Average



This type is only available for Perceptron sensors.

Filter type: **WEIGHTED AVERAGE**

Neighbor #: **3** **Set Weights**
9, 10, 9

Density type: **NONE**

☒ Point reduction filter: **75.0** %

Weighted Average Filter Type

The **Weighted Average** filter smooths stripe data by computing a new location for each pixel. For each pixel in the stripe, this filter will use a weighted average of its neighboring pixels to compute a new location. This is the default filter.

Neighbor #: This value determines the number of total pixels considered when calculating a new location of any given pixel in a single stripe.

Set Weights: This button sets the relative importance of a given pixel's neighbor.

Filter Weights

Center

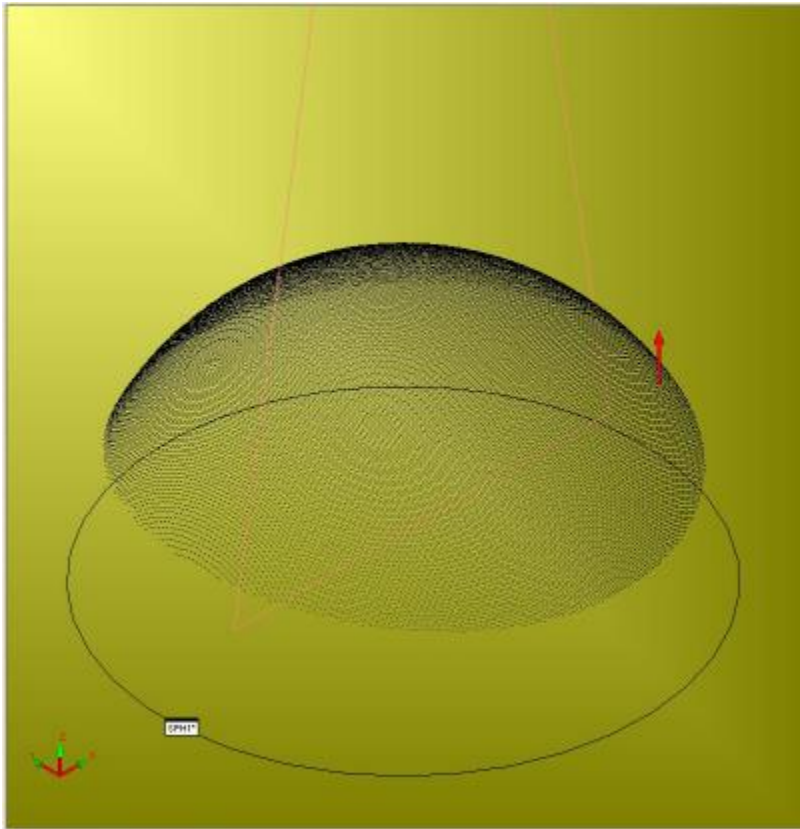
5 7 8 9 10 9 8 7 5

Cancel OK

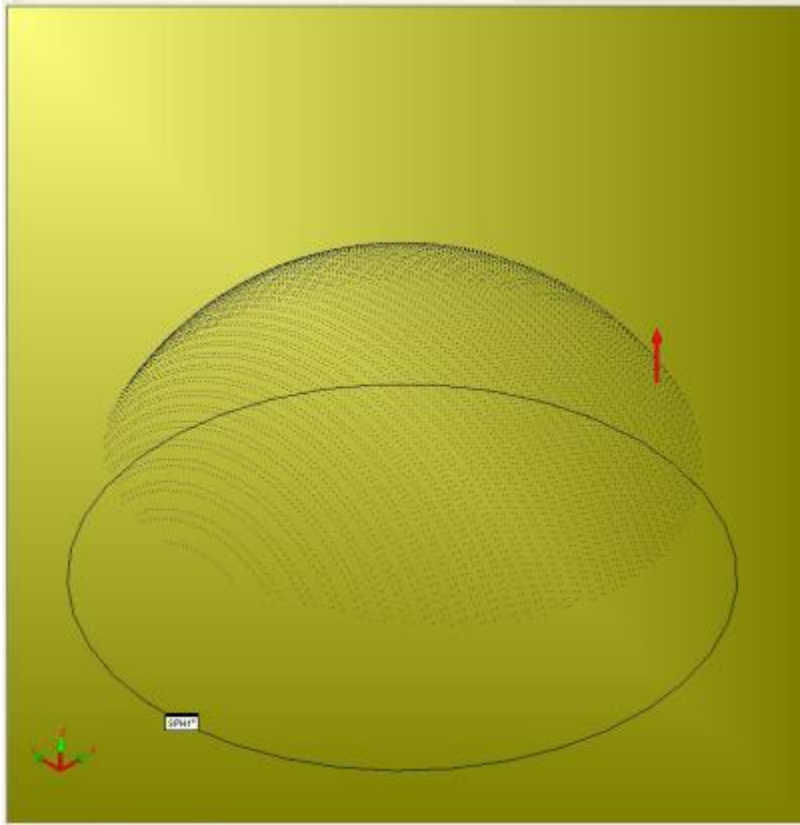
Use the up and down arrows for each of the pixel locations. Click **OK** to save your changes or **Cancel** to close without saving.

Point Reduction Filter: This check box determines whether or not PC-DMIS filters points along the scan line. If marked, you can select the desired percentage of total points to filter. If cleared, the complete data set is acquired without any filtering.

Example of Point Filtering Disabled



Example of Point Filtering of 50%



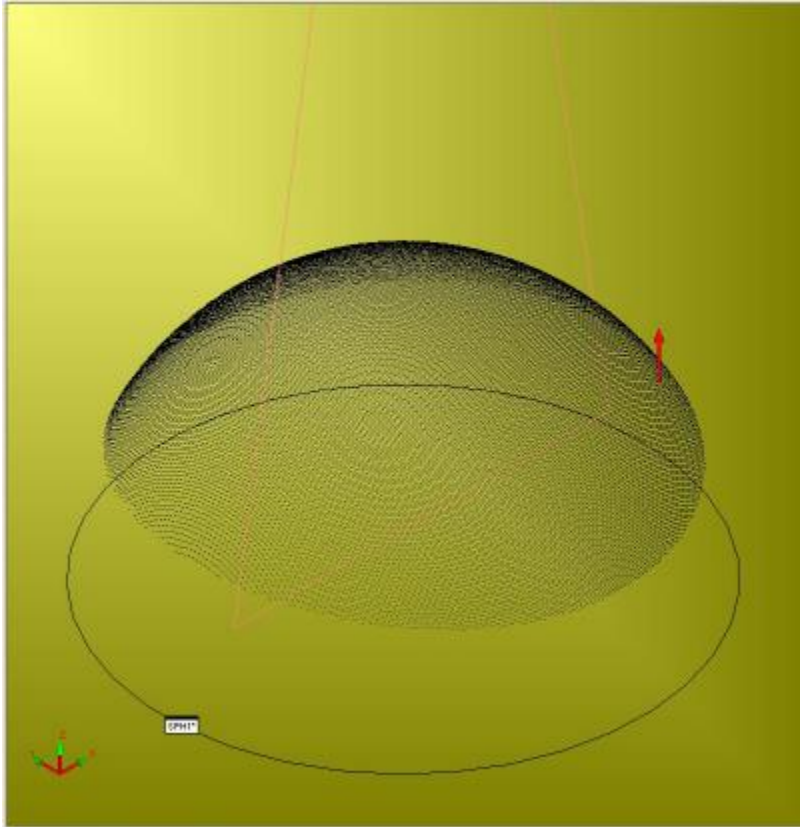
Filter Type: Stripe



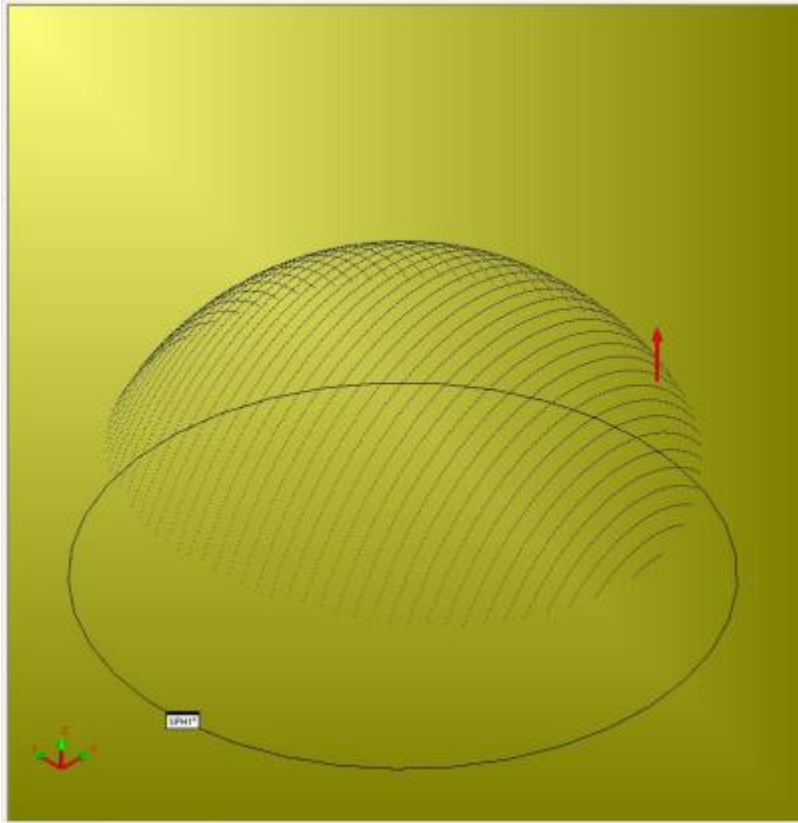
This type is only available for CMS sensors.

The **Stripe Filter** list lets you filter scan lines along the scanning direction. You can select a number from a scale of 1 to 10 (1 represents minimum filtering while 10 represents maximum filtering). If disabled, this acquires the complete data set without any filtering.

Example of Stripe Filtering Disabled



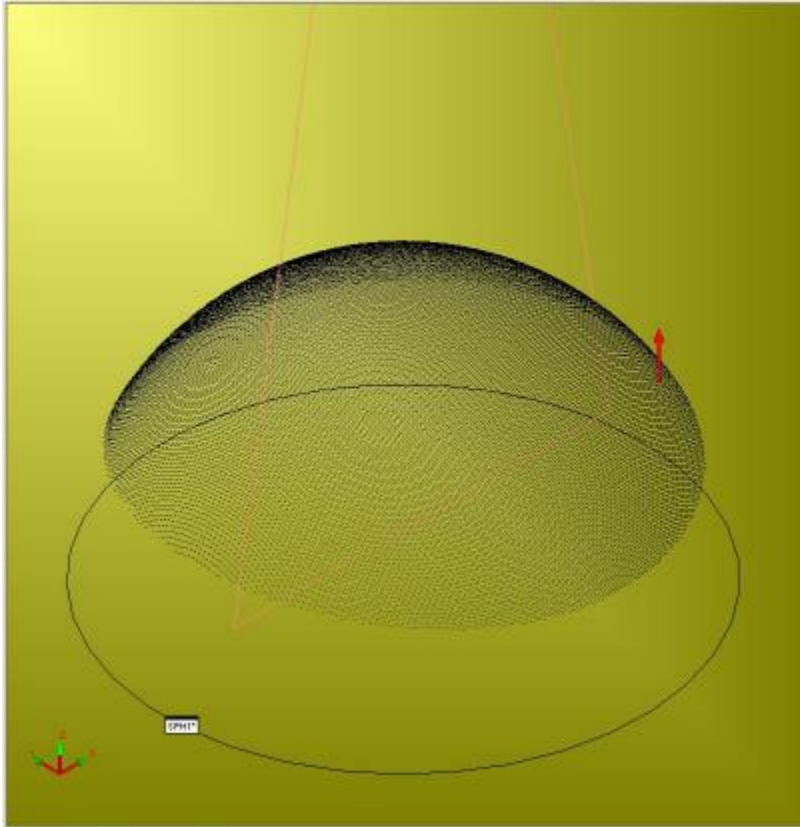
Example of Stripe Filtering of 5



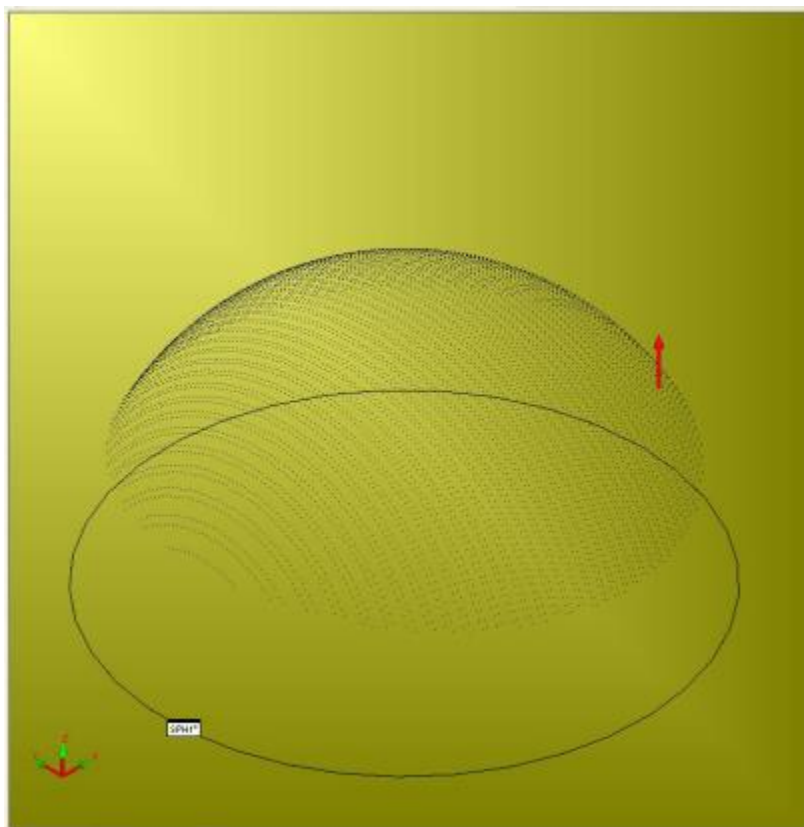
If you're using a CMS sensor with the Perceptron Toolkit as the Feature Extractor, the Auto Feature Square Slot feature only allows odd-numbered stripe filters (1,3,5,7,9).

Point Reduction Filter: This check box determines whether or not PC-DMIS filters points along the scan line. If marked, you can select the desired percentage of total points to filter. If cleared, the complete data set is acquired without any filtering.

Example of Point Filtering Disabled



Example of Point Filtering of 50%



Density Type: Intelligent Density Management



This type is only available for the Perceptron Contour V5 sensor.

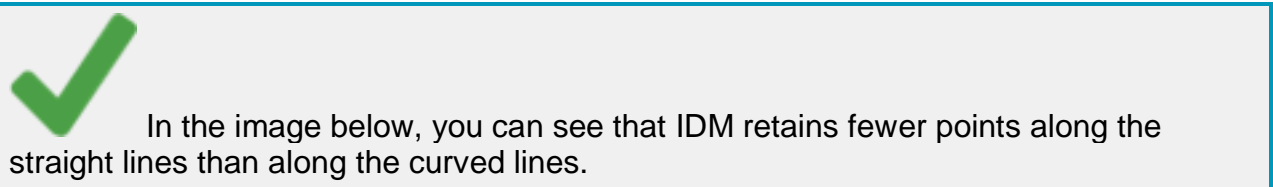
Filter type:	NONE
Density type:	INTELLIGENT DENSITY MANAGEMENT
Flatness tolerance:	70
Maximum span:	1000
<input type="checkbox"/> Point reduction filter:	50.0 %

Intelligent Density Management with Filter Type - None

Intelligent Density Management (IDM) is *only* available for Perceptron V5 laser sensors. You can only scan at high speeds with IDM. You can use features scanned with IDM for auto feature extraction because edge points are found with IDM.

You can use **Filter Type** and **Density Type** together. For example, you might want a Long Line filter with IDM density. However, if you only want to apply the IDM density, set the **Filter Type** to **None**.

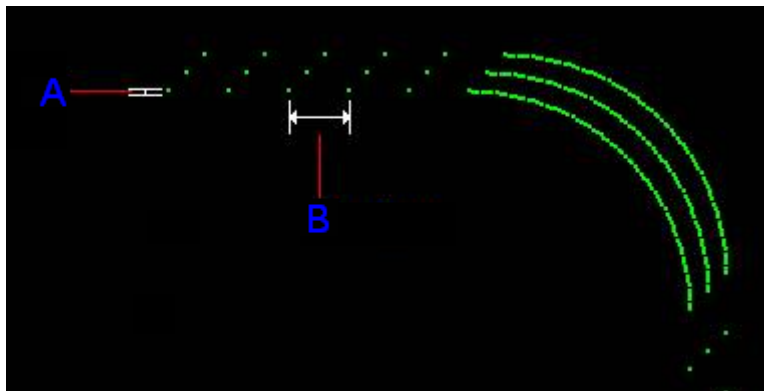
The two IDM settings work together to determine which points to reduce (remove) based on the position of the neighboring points. When data points are considered to be on the same plane, only a few points are needed. IDM retains points if they are outside the **Flatness Tol** or if the **Max Span** distance has been reached.



IDM uses the following settings:

Flatness Tol (A): Provides a tolerance distance in microns. If neighboring points exceed this distance, IDM considers those points to not reside in the same plane. Points that deviate from this range are included in the subset of points. This value should be between 1-60.

Max Span (B): Provides the maximum distance (in microns) that included points can be from each other. Once the **Max Span** has been reached for points that are within the **Flatness Tol** a new point will be included in the subset of points. This value should be between 150-2500.



IDM Sample - **Flatness Tol** (A) and **Max Span** (B)

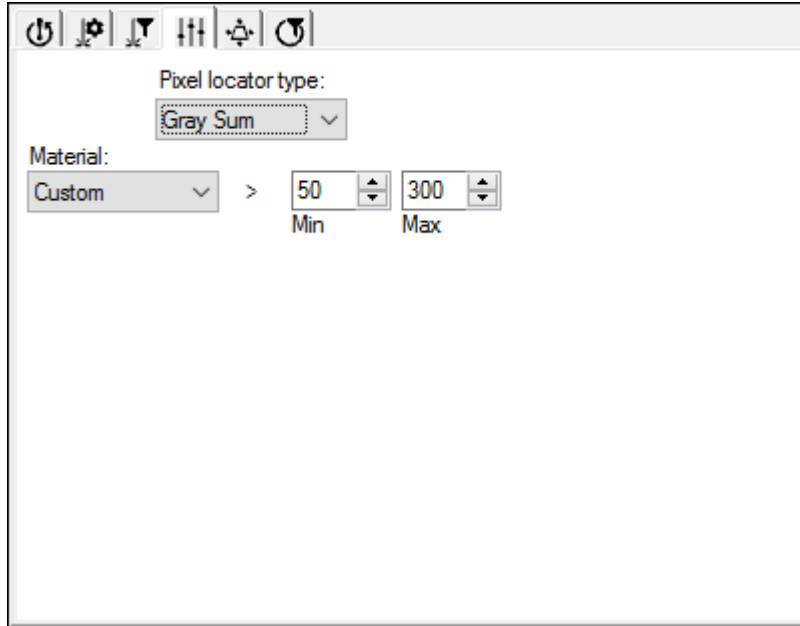
Examples of IDM Settings

Flatness Tol.	Max Span	Result
15	1000	Provides data at nominal 1 mm point-spacing. This allows you to achieve significant data reduction without sacrificing surface details. This is the "optimal data compression" because it provides a good balance of CPU load, memory usage, and graphic card load.
150	2500	This is the maximum data-reducing IDM setting. This setting places a heavy load on the CPU but it reduces memory usage and graphics card load.
1	60	Emulates V4 probe performance with a V5 probe. This setting is easy on the CPU but it requires more memory and puts an increased load on the graphics card.
1	120	This essentially turns off IDM.

Laser Probe Toolbox: Laser Pixel CG Locator Properties tab



Only advanced users in specific situations should use the **Laser Pixel CG Locator Properties** tab.



Probe Toolbox - Laser Pixel Locator Properties tab



Scanning methods with a portable device using a Perceptron laser differ from DCC machines. If you open the **Auto Feature** dialog box and are using a portable device with a Perceptron laser, the **Laser Pixel CG Locator Properties** tab is hidden.

The **Laser Pixel CG Locator Properties** tab only appears if you have a Perceptron laser sensor. This tab uses various mathematical algorithms to change how the software accurately determines the pixels comprising the stripe.

The algorithms operate on an image that consists of rows and columns of pixels. The laser stripe within that image illuminates a band of pixels. The pixel locator then computes the location of the true pixel in the image.

In the following pixel locator algorithms, PC-DMIS computes a surface point based on the illumination of a column of pixels in the image:

Gray Sum: If you select this locator type, PC-DMIS limits the data collection to the parts of the line that fall between the specified **Min** and **Max** values. These minimum and maximum limits are a percentage of the average intensity for each laser line. These limits can be used to improve the data quality for specific part geometry situations. See "Feature and Material Settings".

Material: This list allows you to select a predefined material type (**Custom**, **Sheetmetal**, **White**, **Blue**, **Black** and **Aluminum**) with its corresponding Min/Max values. When you select a material type, the software loads the saved Min/Max values for that material type. Using the default option of **Custom** you can define a generic set of Min/Max values. If you modify the Min/Max values, the **Material** type automatically switches to Custom.

Min: If any part of the laser line's intensity *falls below* this value, the software won't use that part. In situations where the *edges* are important, you can reduce this value so more of the edge data is preserved as the laser wraps around the edges. For a *shiny part* with internal corners that cause reflections and noise in the data, you can increase this value to eliminate the "noise" generated by internal reflections.

Max: If any part of the laser line intensity *exceeds* this value, the software won't use that part. In some situations where a part has many contours that you cannot easily follow, the laser reflects strongly. This causes localized over-exposures. Reducing this value may help to ensure that the overexposed areas do not provide bad data.



The software always selects Gray Sum for portable devices using the Perceptron V5 laser sensor.

Fixed Threshold: If you select this locator type, PC-DMIS discards all the data below the threshold and it computes the actual pixel location as the center of gravity of the remaining pixels within the column.

Gradient: If you select this locator type, PC-DMIS computes the actual pixel location. It looks at a column of pixels and finds where the slope changes direction. For each direction change, PC-DMIS creates a pixel.

Exposure and Gray Sum Settings by Feature and Material

Based on the feature type and the part material type the Exposure value found on the **Laser Scan Properties** tab and the **Min** and **Max** Gray Sum values found on the **Laser Pixel CG Locator Properties** tab should be adjusted according to the following table:

Exposure and Gray Sum Settings				
Feature Based				
Feature	Material	Exposure	Min Grey Sum	Max Grey Sum
Sphere	Tungsten Calibration Sphere	120	10	300
	Ceramic	80	10	300
Gap/Flush	Sheet Metal	150	30	300
	White	100	30	300
	Blue	120	30	300
	Black	450	10	300
Circle	Sheet Metal	100	50	300
	White	100	50	300
	Blue	120	50	300
	Black	450	30	300
	Aluminum	80	50	300
Slot	Sheet Metal	100	50	300
	White	100	50	300
	Blue	120	50	300
	Black	450	30	300
	Aluminum	80	50	300
Edge Point	Sheet Metal	100	50	300
	White	100	50	300
	Blue	120	50	300
	Black	450	30	300
	Aluminum	80	50	300
Plane	Sheet Metal	100	30	300

	White	100	30	300
	Blue	120	30	300
	Black	450	10	300
	Aluminum	80	30	300
Surface Point	Sheet Metal	100	30	300
	White	100	30	300
	Blue	120	30	300
	Black	450	10	300
	Aluminum	80	30	300

Exposure and Gray Sum Settings

Exposure and Gray Sum Settings During Calibration

Prior to the start of calibration, PC-DMIS sets the exposure and gray sum values to the following:

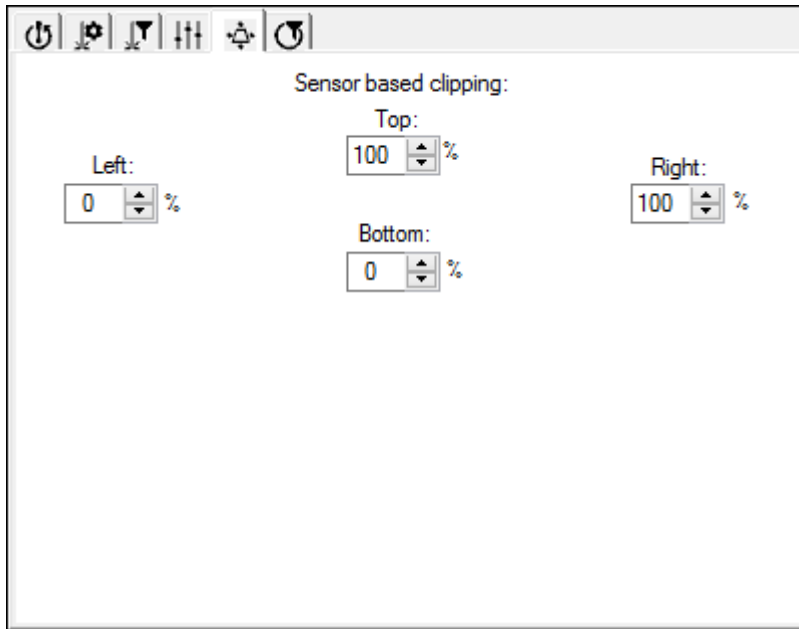
- **Exposure:** 300
- **Gray Sum Min:** 10
- **Gray Sum Max:** 300

These are the settings that work best for most calibration scenarios. Once the calibration finishes, PC-DMIS restores your original exposure and gray sum values (from before calibration). While gray sum values of 10, 300 are often appropriate for calibration, values of 30, 300 are typical for normal scanning.

Also, the default exposure value of 300 is often not sufficient in rare lighting conditions (such using a V4i with sodium lighting). If PC-DMIS has difficulty accepting the arcs during the calibration process, you may need to raise the default calibration exposure value to 400 or so. In these cases, modify the

`PerceptronDefaultCalibrationExposure` registry entry located in the **NCSensorSettings** section of the PC-DMIS Settings Editor. For details, see the PC-DMIS Settings Editor documentation.

Laser Probe Toolbox: Laser Clipping Region Properties tab



Laser Clipping Region Properties tab

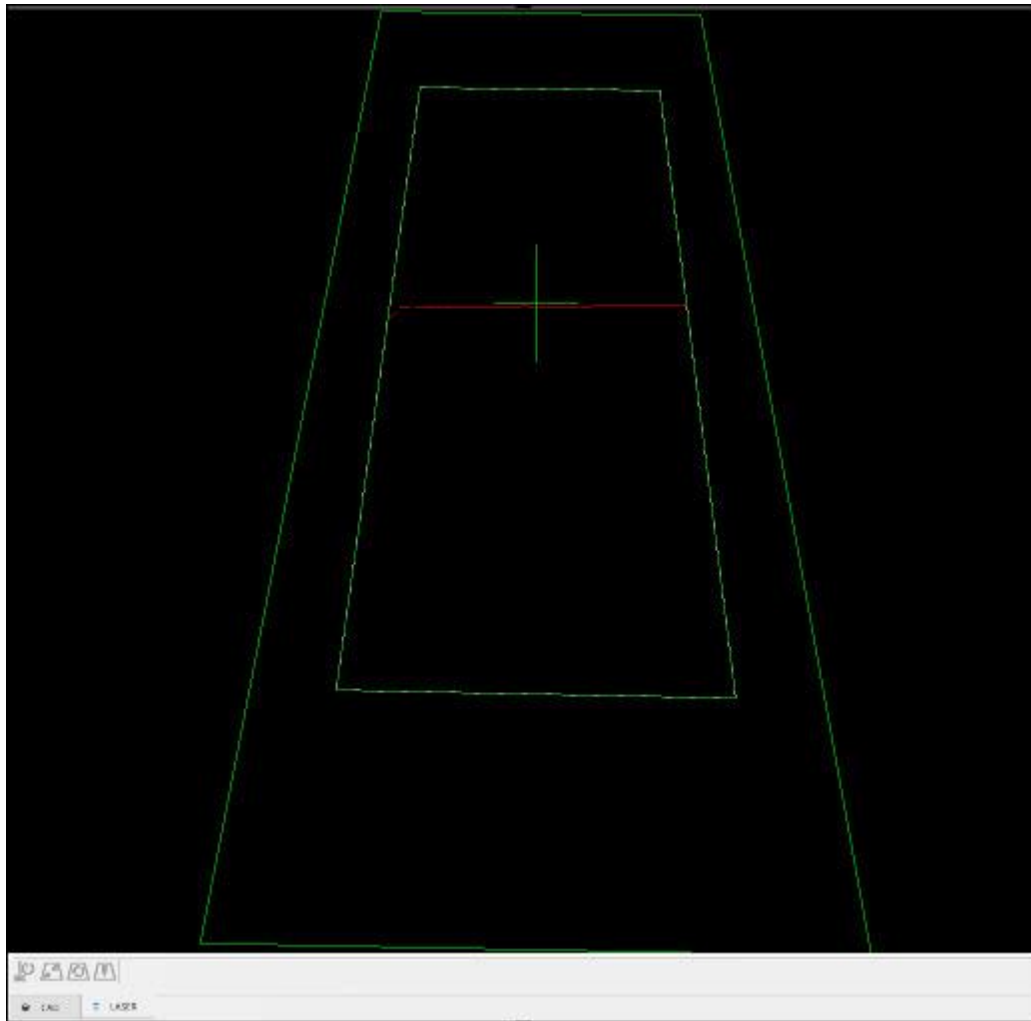
The **Laser Clipping Region Properties** tab allows you to set parameters to discard data outside a specified region, within the sensor's field of view. This feature lets you keep only pertinent data.

Keystone: The large green trapezoid in the Laser View (see below) that represents the sensor's maximum field of view. The clipping region is within this field of view.

Sensor Based Clipping region: The smaller green trapezoid within the sensor's field of view.

The **Top**, **Left**, **Right**, and **Bottom** boxes can be set with values from 0 to 100 percent that allow control over the clipping region. This lets you discard data that is not needed.

When the **Bottom** and **Left** values are at 0% and the **Top** and **Right** values are at 100%, the sensor keeps all of the data collected because the clipping region is the same as the maximum field of view.



Example of clipping data using Top 85, Bottom 85, Left 15, and Right 15

You can use the clipping region, for example, when measuring a hole. Since you wouldn't want data from a neighboring hole to interfere with the feature computation, you can control the area that is clipped, thereby discarding the unwanted data.

Laser Probe Toolbox: Feature Extraction tab

The screenshot shows the 'Feature Extraction' tab in the Laser Probe Toolbox. It contains three main sections:

- Feature Based Clipping:**
 - Horizontal (mm): 2
 - Vertical (mm): 10
 - ☐ CAD offset: 2
- Ring Band:**
 - ☐ Ring Band
 - Inner offset (mm): 0.5
 - Outer offset (mm): 2
- Filters:**
 - ☐ Remove outliers
 - Standard deviation multiple: 1
 - ☐ Remove points with normals outside:
 - Max incidence angle: 75

Feature Extraction tab

You can use the **Feature Extraction** tab to specify ring band and feature-based clipping parameters as well as remove outliers on supported features.



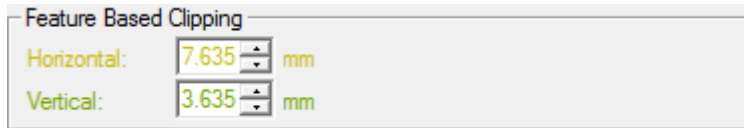
The **Feature Extraction** tab is only available when using a laser sensor.

Depending on the feature type, the following feature extraction parameters are available:

- Feature Based Clipping parameters: This is available for all auto features.
- Ring Band parameters: This is only available for Circle, Cone, Cylinder, Round Slot, and Square Slot auto features.
- Filters:
 - Remove outliers parameter: This is only available for Surface Point, Plane, Cone, Cylinder, Sphere, and Flush and Gap auto features.
 - Remove points with normals outside parameter: This is only available for Surface Point, Plane, Circle, Round Slot, Square Slot, Polygon, Cylinder, Cone, and Sphere auto features.

Also see "Extracting Auto Features from Pointclouds".

Feature Based Clipping Parameters



Feature Based Clipping

Horizontal: 7.635 mm

Vertical: 3.635 mm

Feature Based Clipping area for non-plane auto features

PC-DMIS can clip laser data in both the horizontal and vertical directions when you type a distance in the **Horizontal** box and, when available, the **Vertical** box. This distance clips all of the laser data outside of the defined distance and excludes that data when it extracts the feature.

Alternately, for the Plane auto feature, you can clip data within an offset boundary around all of the CAD elements on a surface. This is also termed CAD segregation. See "CAD Offset" below.

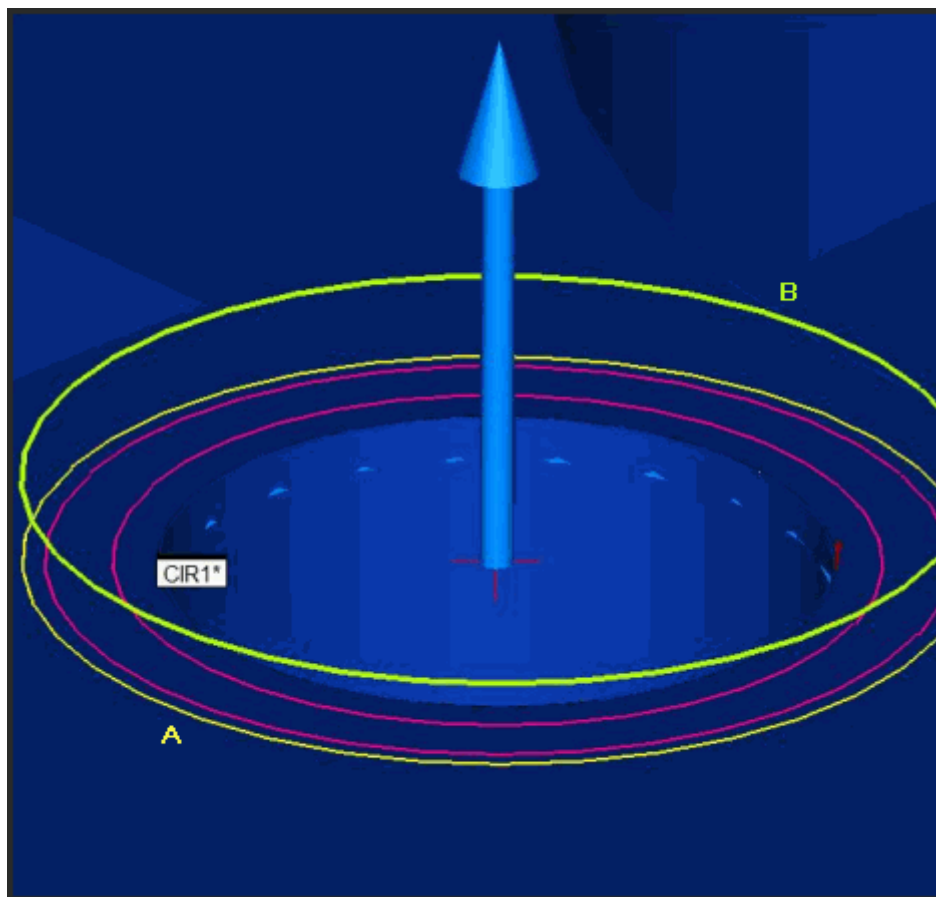
For the Cone auto feature, the value for the **Horizontal** option defines how much larger than the theoretical diameter is the circular boundary within which the feature points lie. The value for the **Vertical** option defines how much longer than the theoretical length is the cylindrical boundary within which the feature points lie.

Horizontal and Vertical Clipping

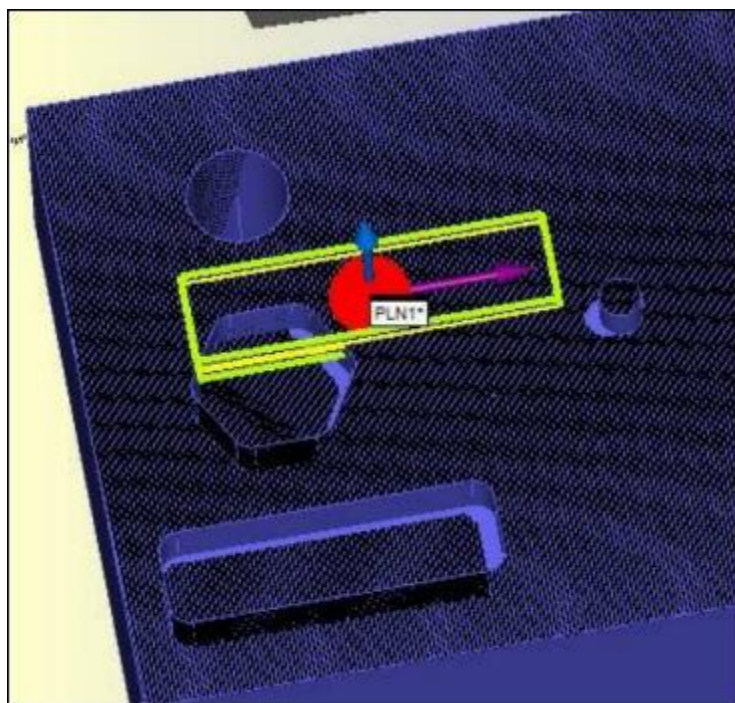
All of the auto features support horizontal clipping. These features support vertical clipping:

- Circle
- Cone
- Cylinder
- Polygon
- Edge Point
- Round Slot
- Square Slot
- Surface Point
- Plane

The clipping distances defined in the feature based clipping rings are shown as colored rings. The horizontal clipping appears as a yellow ring, and the vertical clipping appears as a light green ring.



Sample Circle auto feature with horizontal clipping (A) and vertical clipping ring (B)



Sample Plane auto feature with horizontal and vertical clipping enabled

CAD Offset

Feature Based Clipping	
Horizontal (mm):	3
Vertical (mm):	1
<input checked="" type="checkbox"/> CAD offset:	3

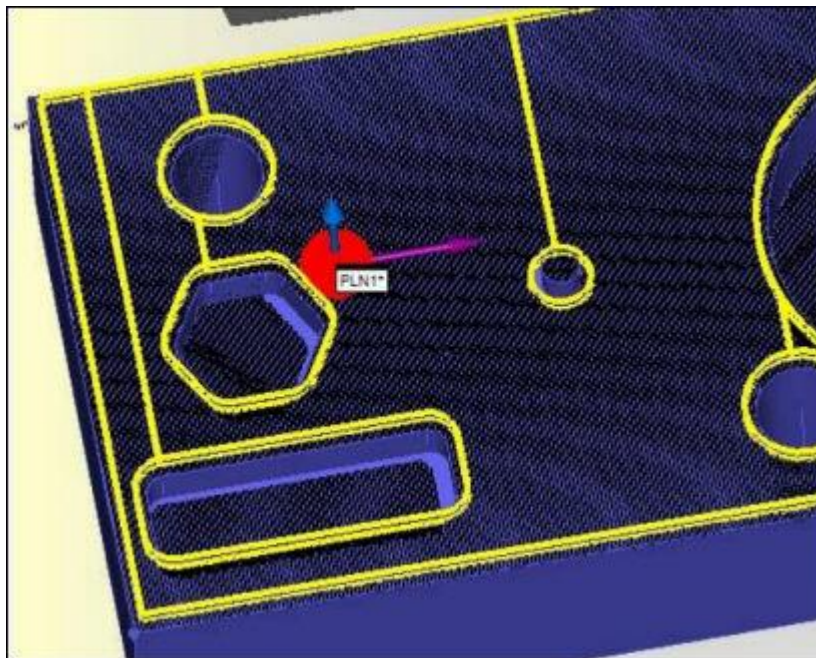
Feature Based Clipping area for Plane auto feature



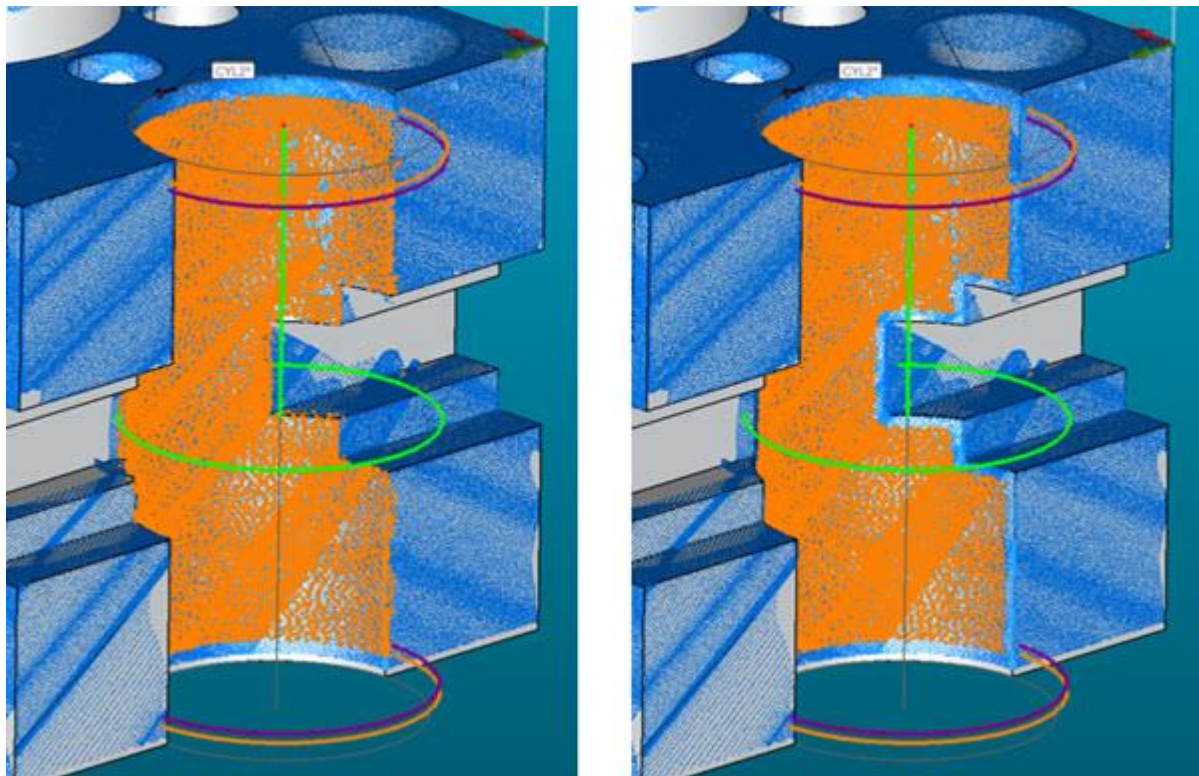
The **CAD offset** check box is available for all 3D Auto features (Plane, Cone, Cylinder and Sphere).

You can enable the CAD offset option for Plane, Cone, Cylinder and Sphere auto features. The **CAD offset** option provides a way for PC-DMIS to shrink away from the selected CAD face and eliminate points which are within the offset distance to the edges of the feature.

For Plane auto features, when you select this check box, PC-DMIS creates a yellow offset boundary around each feature in the CAD model on the surface. For a Cone, Cylinder or Sphere auto feature, PC-DMIS does not display this yellow offset boundary outline.



Example of a Plane auto feature with CAD-based clipping enabled



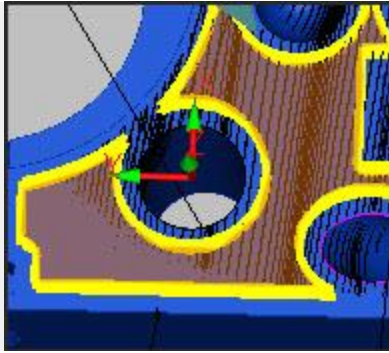
Example of a Cylinder auto feature with no CAD offset (left) and with 2mm CAD offset (right)

You must select a CAD face when using the **CAD offset** option.

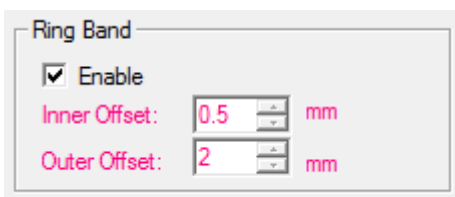


PC-DMIS clips the laser data that falls inside of an offset boundary for all of the features in the CAD model on a surface. The data outside of the offset boundary is used to solve the plane.

For example, consider the image below, which shows a section of a sample part. The translucent orange overlay, added to the image here for clarification only, indicates the data that PC-DMIS would use to create the Plane auto feature:



Ring Band Parameters



Feature Extraction - Ring Band

The **Ring Band** area is used to calculate the feature's projection plane and normal vector. The feature data is projected up into the ring band's plane. The following **Ring Band** controls are used to accomplish feature extraction for circles, round slots and square slots:

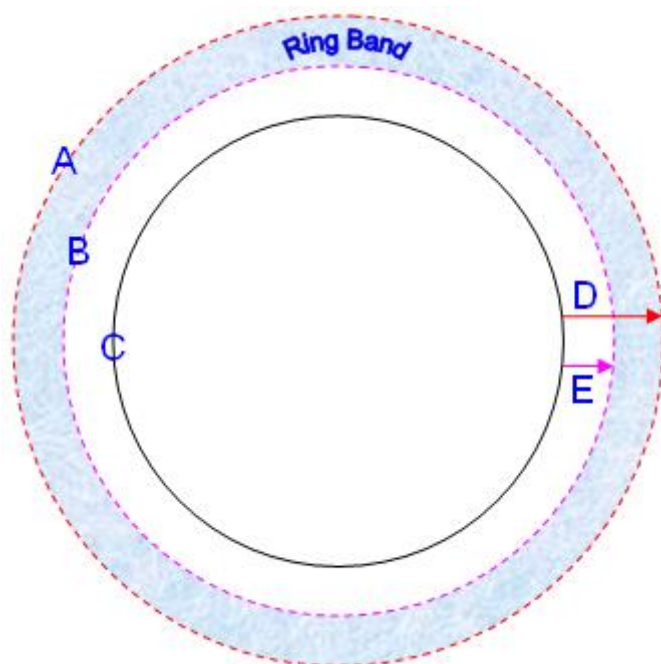
Enable - When this option is selected, the **Ring Band** options take effect.

The following default values are used when Auto Circle, Auto Round Slot, and Auto Square Slot are disabled:

- **Inner Offset** = 0.4x the theoretical diameter value
- **Outer Offset** = **Inner Offset** value + 3mm

Inner Offset - Provides the offset from the theoretical feature radius or form for the *inner* edge of the ring band. This value is expressed in measurement routine units and must be greater than or equal to zero (a value of zero means an inner edge of the ring band coincides with the feature nominal.) See Image below.

Outer Offset - Provides the offset from the theoretical feature radius or form for the *outer* edge of the ring band. This value is expressed in measurement routine units and must be greater than the **Inner Offset** value. See Image below.



(A) Ring band Outer Edge

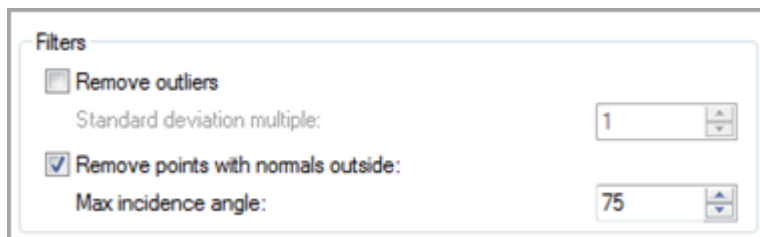
(B) Ring Band Inner Edge

(C) Feature Theoretical value

(D) Outer Offset

(E) Inner Offset

Filters



Feature Extraction - Filters area

Remove outliers: When you select this check box, it excludes outliers from the feature based on the value for the **Standard deviation multiple** option. The **Remove outliers** check box applies only to the Auto Cone, Auto Surface Point, Auto Plane, Auto Cylinder, Auto Sphere, and Auto Flush and Gap features.

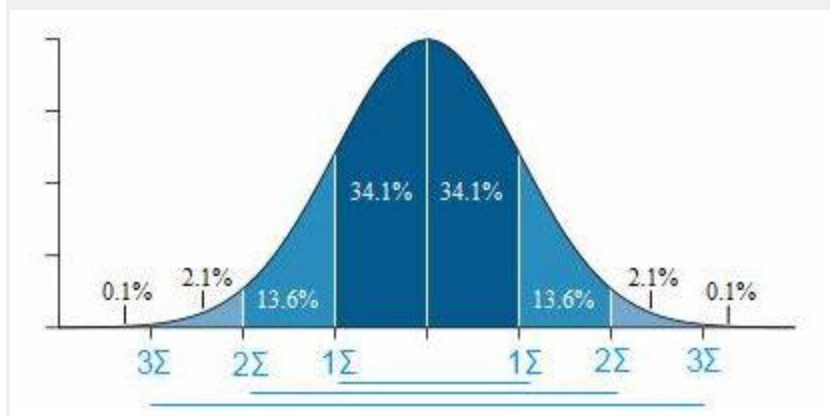
- The feature extractor evaluates the feature internally two or more times on the first attempt to get the standard deviation based on all points.

- In successive attempts, it re-evaluates the feature using only the points that are in the range of the outlier multiplied by the Σ . The sigma is the range, in the Gaussian distribution of the deviations, where the 68.2% of the best points used for fitting the feature lie.

Standard deviation multiple: The value for this option defines the selectivity of the filter. It can be a generic real number that is greater than 0. If **m** is the selected value, it means that all the scan points that deviate from the extracted cone are greater than **m x Actual standard deviation** (that is, the standard deviation of the measured points with respect to the calculated feature) and are cut off from the calculation. Therefore, the lower the value of **m**, the more selective the filter.



In the first evaluation, the standard deviation is evaluated on all points. In a normal distribution, this could be represented as follows:



This means that the best points are in the interval from 0 to 1Σ . For example, if you wanted to get points only in that range, you would need to specify an outlier value from 0 to 1. Worse solutions would be obtained if you used higher outlier values.


Remove points with normals outside:

When enabled, this setting compares the estimated normal of each scanned point within the clipping zone to the feature theoretical normal (or CAD surface for 3D features).



This parameter is only available for laser Circle, Cone, Cylinder, Edge Point, Flush and Gap, Plane, Polygon, Round Slot, Sphere, Square Slot, and Surface Point auto features. The Edge Point and Flush and Gap features use the 2D filter method.

When measuring the laser feature, this filter excludes scanned points which are on the opposite side of the part or on adjacent surfaces. The smaller the **Max incidence angle**, the more points are excluded.

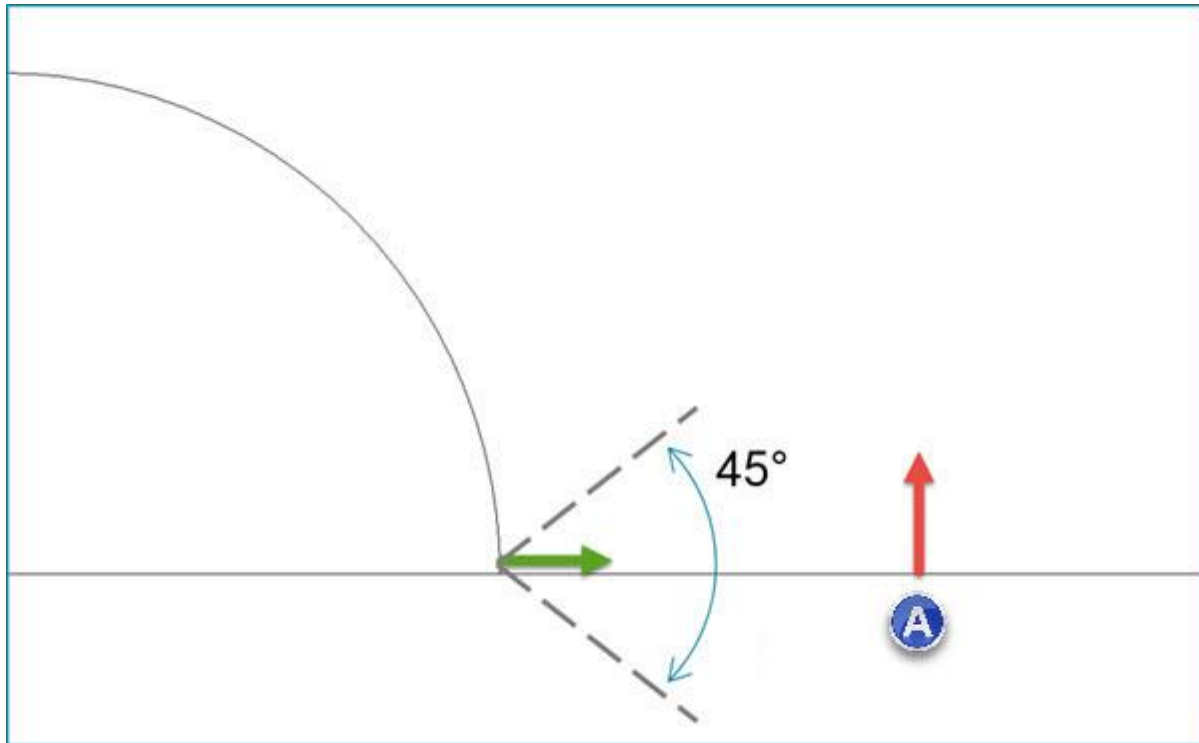
The effect of the **Max incidence angle** filter is enabled when the **Show/Hide segregated points** button () on the **Laser Scan Properties** tab of the **Laser Auto Feature** dialog box is enabled .

3D Features Using the Max Incidence Angle

Laser Auto Features have a Horizontal and Vertical clipping zone. All scanned points within the clipping zone are initially evaluated.

For 3D features (Surface Point, Plane, Cylinder, Cone, and Sphere), this setting compares the estimated normal of each scanned point to the feature theoretical normal or the vector of the CAD surface if a CAD model is used.

Points with a vector that fall outside of this angle are excluded when measuring the feature.



(A) - Plane (adjacent surface)

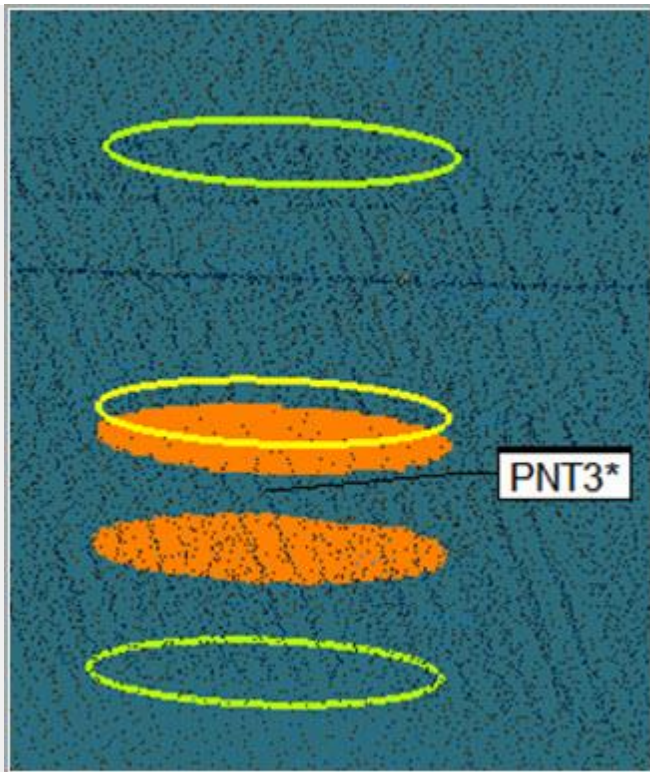
Example of the applied to a 3D Laser Auto Point feature



On a thin sheetmetal part that was scanned from both sides, a Laser Auto Surface Point was created.

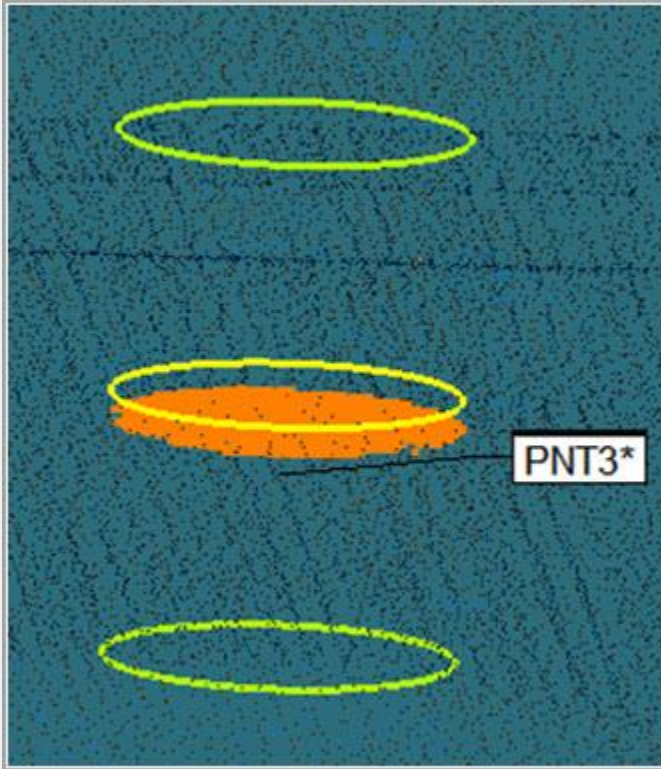
The Feature Extraction - Vertical clipping zone is set so it includes the part deviations, which in this case are larger than the sheetmetal thickness.

In this image, the scan uses no **Max incidence angle**:



Since the normals of the scanned points are not taken into consideration, the extracted point uses data from both sides of the part.

In this image, the scan uses a **Max incidence angle** of 60 degrees:



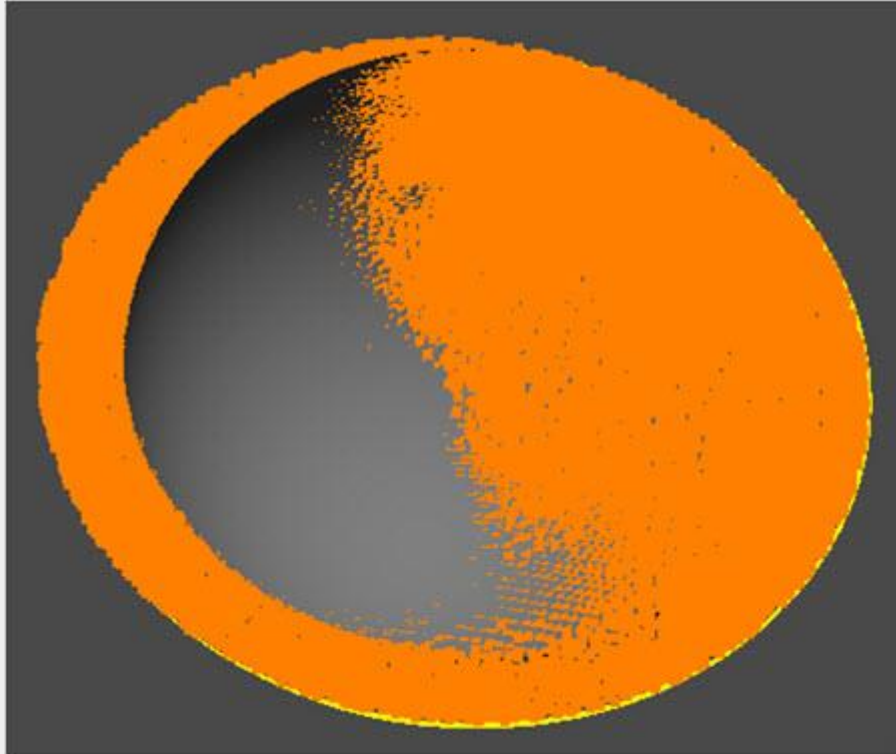
The software compares the estimated normal of each point in the clipping zone to the laser auto surface point theoretical normal. Points that fall outside this angle are not used for the feature calculation.

Example of the applied to a 3D Laser Auto Sphere feature



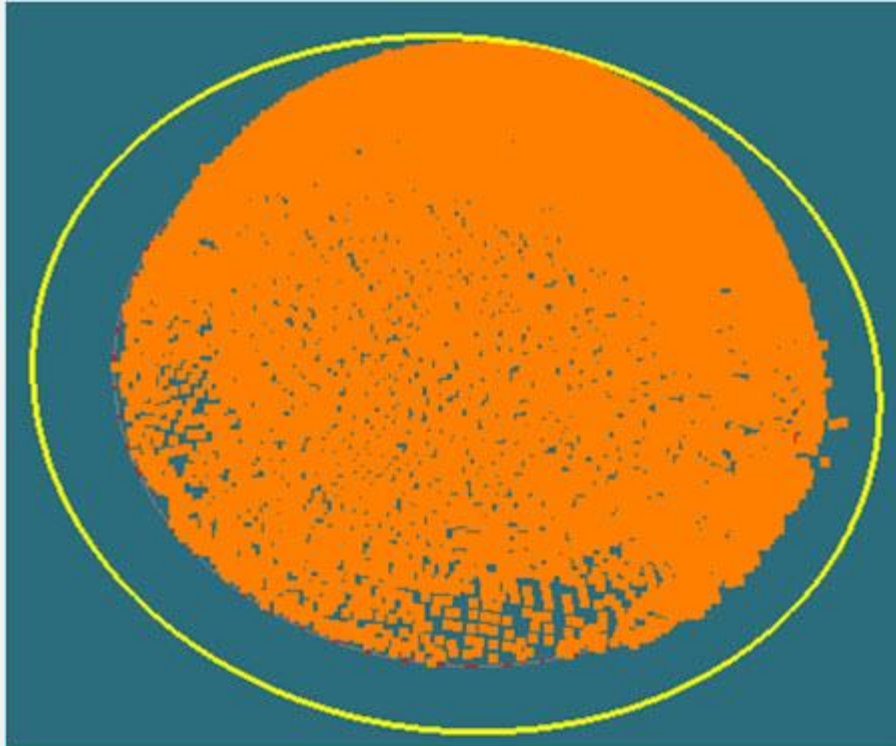
Laser extraction of a sphere previously required additional steps and manual selection to exclude adjacent surfaces.

In this image, no **Max incidence angle** is used:



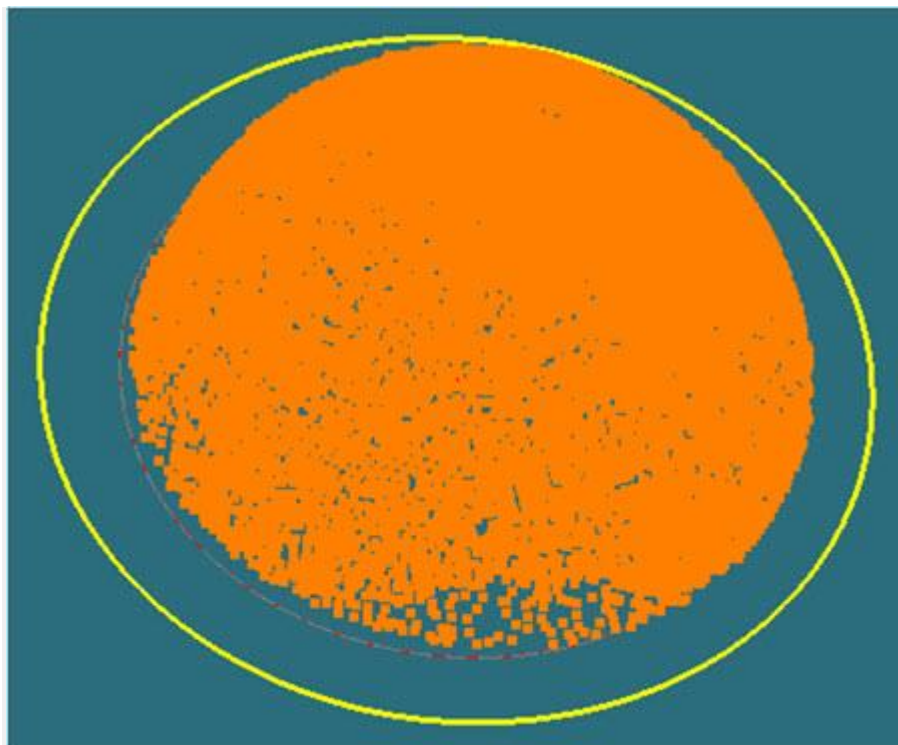
Data from the adjacent plane is used for sphere calculation.

In this image, a **Max incidence angle** of 60 is used:



A few outlying points are included.

In this image, a **Max incidence angle** of 45 degrees is used:

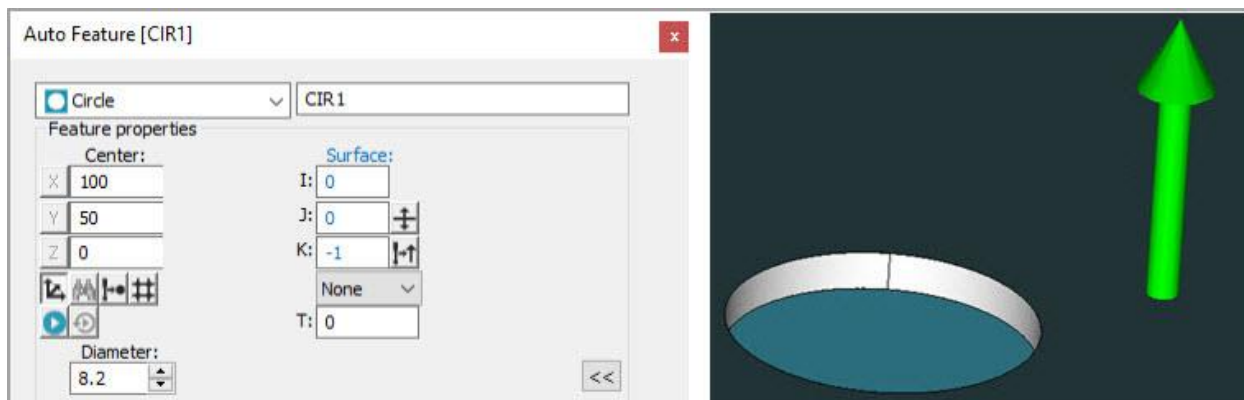


In this final example, the actual sphere data is best represented.

2D Features Using the Max Incidence Angle

Laser Auto Features have a Horizontal and Vertical clipping zone. All scanned points within the clipping zone are initially evaluated.

For 2D features (Circle and Slots), this setting compares the estimated normal of each scanned point to the feature theoretical Surface Normal.



(A) - Surface Vector

Points with a vector that fall outside this angle are excluded when measuring the feature.

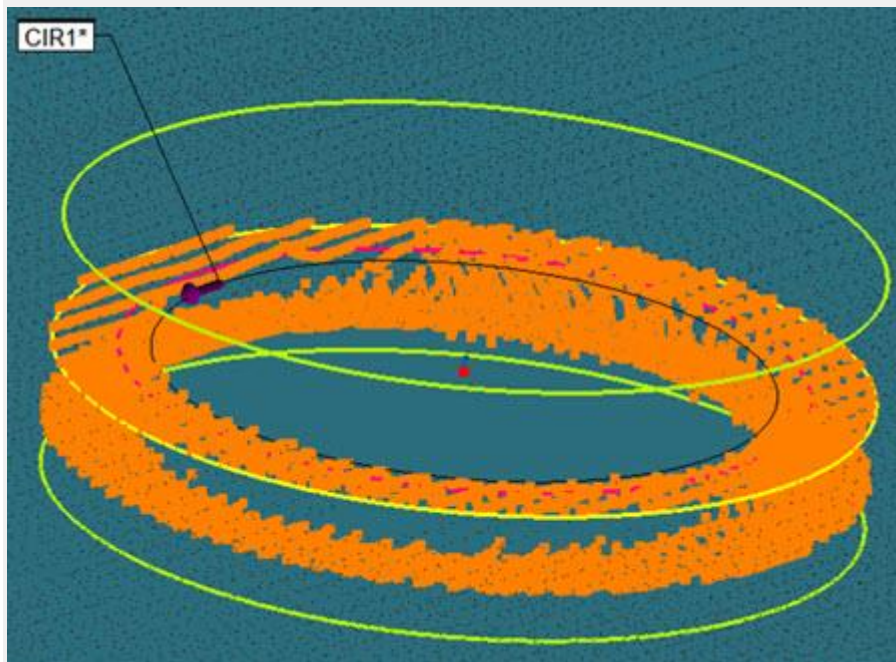
Example of the applied to a 2D Laser Auto Circle feature



On a sheetmetal part that was scanned from both sides, a Laser Auto Circle was created.

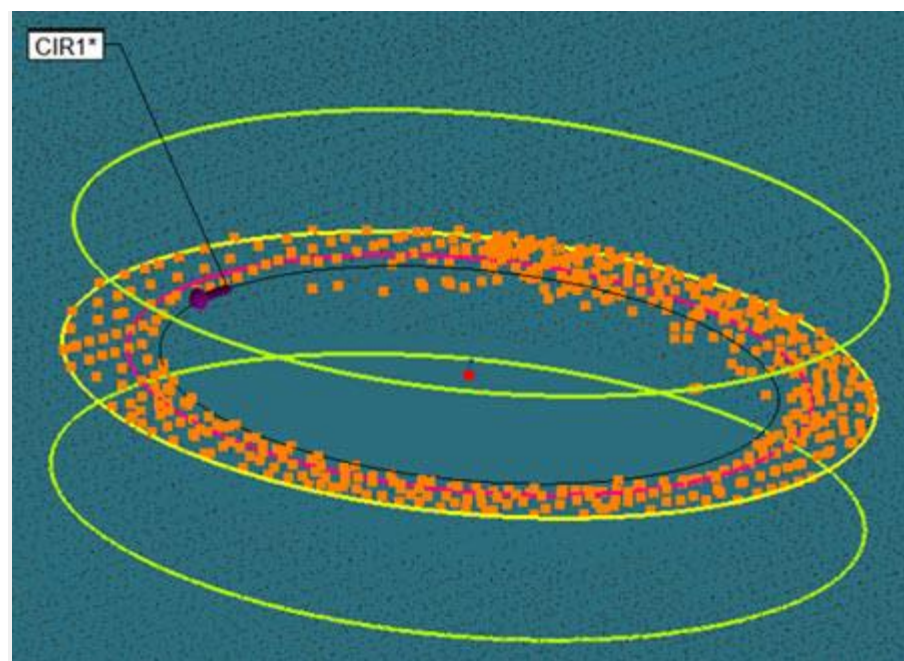
The Feature Extraction - Vertical clipping zone is set so it includes the part deviations, which in this case is larger than the sheetmetal thickness.

In this image, no **Max incidence angle** is used:



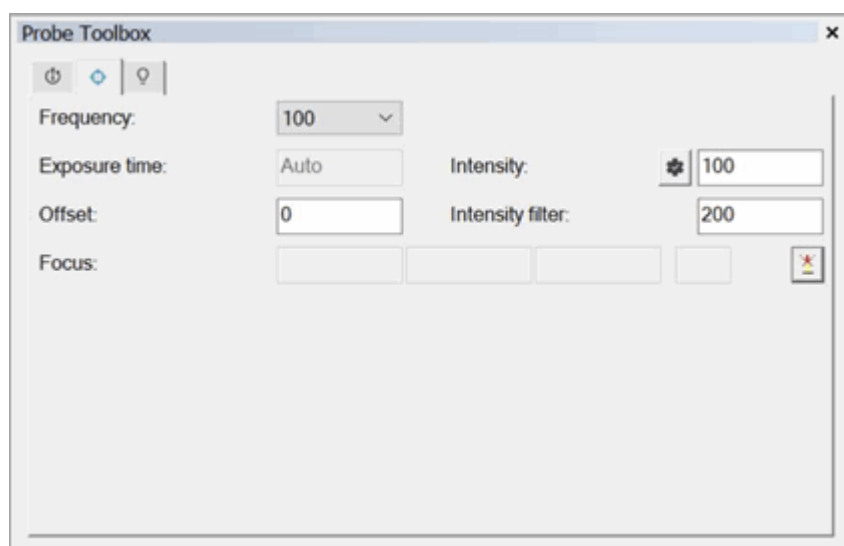
Since the normals of the scanned points are not taken into consideration, the extracted circle uses data from both sides of the part.

In this image, a **Max incidence angle** of 75 degrees is used:



The estimated normal of each point in the clipping zone is compared to the Laser Auto Circle, surface theoretical vector. Points with a vector which fall outside this angle are not used for the feature calculation.

Laser Probe Toolbox: CWS Parameter tab



Probe Toolbox - CWS Parameter tab

The **CWS Parameter** tab in the Probe Toolbox (**View | Other Windows | Probe Toolbox**) is available once the system has been appropriately configured:

- You must configure the CWS as the active laser system. Usually, the factory does this locally during the startup procedure or by a service engineer.
- With the system configured, you must next define a probe with the correct properties. You can construct the probe in the **Probe Utilities** dialog box. You should use the OPTIV_FIXED selection and a lens that includes CWS. You should define this in the USRPROBE.DAT file if not provided by the factory.

The CWS Parameter tab can include the following information:

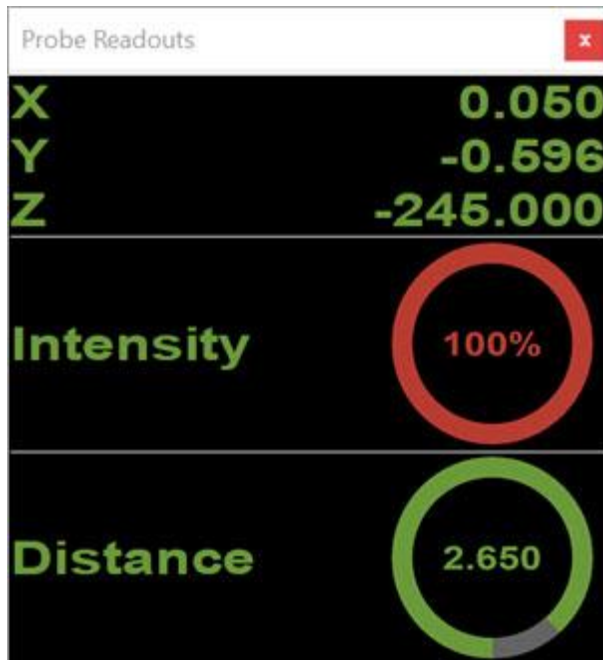
Frequency (Measurement Rate)

The measurement rate sets the number of measured values that the optical sensor records per unit of time. For example, when the measurement rate is set at 2000 Hz, 2000 measurement values are taken per second. The intensity indicator on the display can help you select the correct setting.

Setting Range

As a rule, you should strive to measure at the highest possible measurement rate in order to acquire as many measurement values in as little time as possible. In the case of surfaces with very low reflectivity, it may be necessary to reduce the measurement rate. This has the effect of illuminating the optical sensor's CCD-line longer and thus makes it possible to perform measurements, even if the reflected intensity is very low.

Overmodulation of the CCD-line on highly reflective surfaces and at small measurement rates can lead to measurement errors. If the intensity indicator on the CWS controller box displays a blinking „**Int: 999**“, or the Probe Readouts window displays an intensity value in red at or near 100%, overmodulation is occurring.



Probe Readouts window showing overmodulation

When overmodulation occurs, select the next-highest measurement rate. If the maximum measurement rate (2000 Hz on CHRcodileS, 1000 Hz on CHR150E) is already set, you can reduce the reflected intensity in one of two ways:

- Position the sensing head in the upper or lower threshold of the measurement range.
- Engage the **autoadaptfunction** (where the **Auto Intensity** parameter is set to **ON**). This adapts the lamp intensity continuously, depending on the part reflection. Here, a dark reference is not used. This is the method supported in PC-DMIS.

Exposure time (Brightness Value)

You can select the exposure time (brightness value) here if the **AUTO Intensity** parameter is set to **ON**.

The brightness of the lamp is modulated such that you achieve a defined percentage of the modulation amplitude. The value can lie in the range of 0% to 75%. For most applications, a recommended brightness value is between 20% and 40%.


Auto Intensity

This value defines the relative pulse duration of the LED, and with it the effective brightness of the light source.

If you are measuring a highly-reflective surface on which the highest measurement rate still results in overmodulation, then it makes sense to reduce the exposure time.

The best way to measure a poorly-reflective surface with a high measurement rate is to use a longer pulse duration.

Auto Intensity: OFF

Turn the **Auto Intensity** button () off to use the current light intensity of the LED.

Auto Intensity: ON

With the **Auto Intensity** button set to ON, the independent adjustment of flash time for the LED during an exposure time allows for you to automatically receive the best intensity settings when measuring on variable surfaces. This also allows for an optimal signal-to-noise ratio.

The system modulates the brightness of the lamp to achieve a defined percentage of the modulation amplitude. The value can lie in the range of 0% to 75%. For most applications, the recommended brightness value is between 20% and 40%.

Offset

This is the offset distance that the machine moves to in the measurement direction in addition to the measurement position.

Intensity filter


This value defines the threshold between noise and the measurement signal. The software recognizes that the peaks that fall beneath this threshold are invalid, and show on the display as the measurement value "0".

For a valid measurement, the intensity should fall between 0 and 999 on CHRocodileS or 99 on CHR150E; otherwise, you must change the measurement rate.

If you measure the distance to a surface with low reflectivity, the intensity of the reflected light can be too low, and you must reduce the measurement rate. For a measurement rate under 1 kHz, the threshold value should be 40 on CHRocodileS or 25 on CHR150E. This prevents measurement values of too low an intensity, which rise only slightly above the noise, and would falsify the measurement.

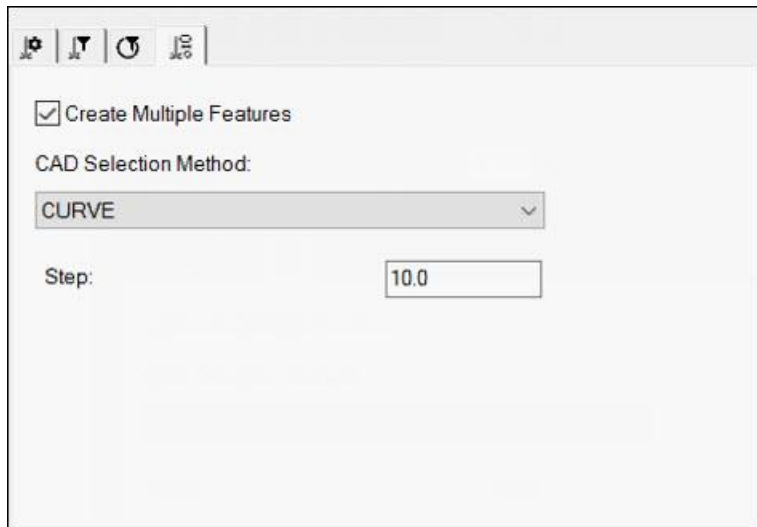
With a measurement rate of 1 kHz and higher (only for CHRocodileS), a threshold of 15 is best to fully exploit the device's dynamics.

Focus

This section has four boxes for the X, Y, Z, and Signal Quality. Click the **Auto Focus** button  on the right to execute a focus or surface measurement to display the X, Y, Z, and Signal Quality values.

For more details, see "CWS Parameters" in the PC-DMIS Vision documentation.

Laser Probe Toolbox: Laser AF Multiple Creation tab



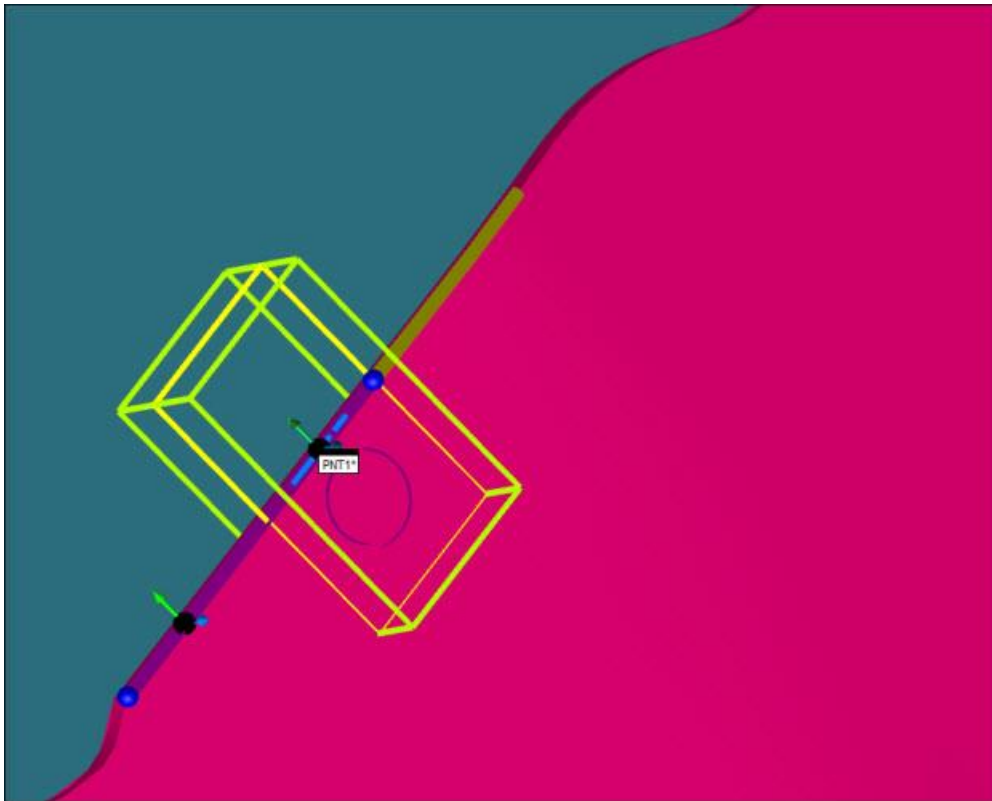
Probe Toolbox - Laser AF Multiple Creation tab

The **Laser AF Multiple Creation** tab is only available for the Laser Edge Point auto feature. This tab appears when the **Pointcloud** option on the **Laser Scan Properties** tab for the Laser Edge Point auto feature is set to a valid COP ID (the option is not set to **Disabled**).

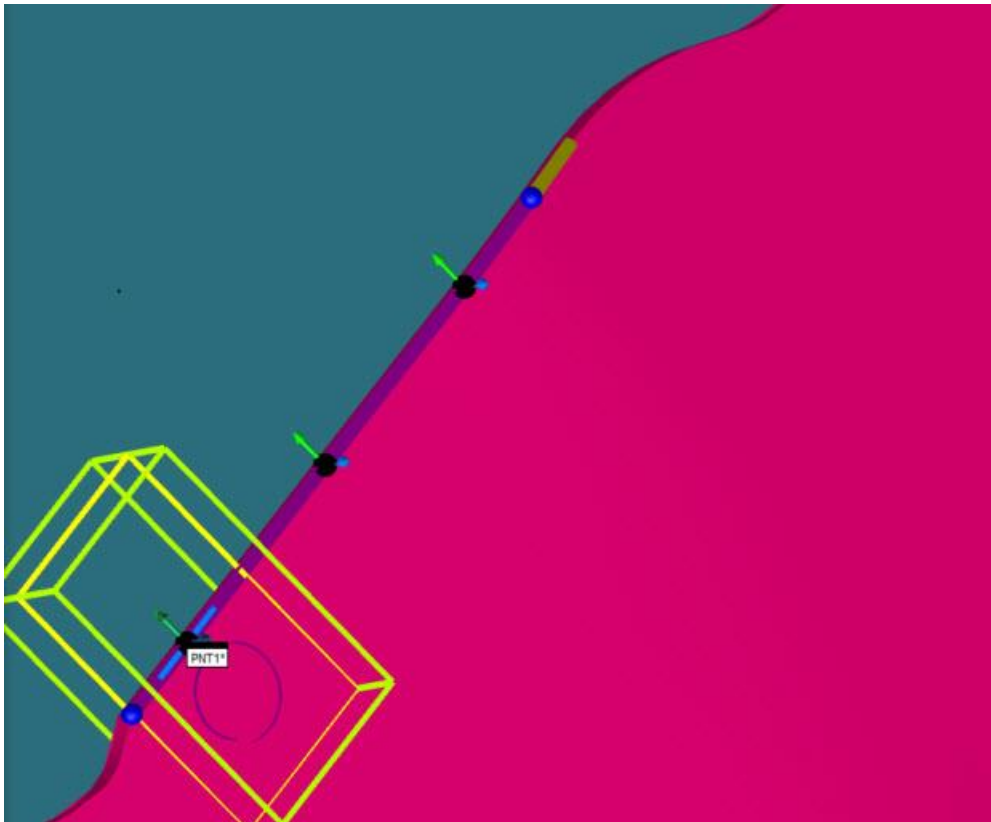
You can use this tab for extracted auto features where the element is extracted from an existing COP object. You cannot use it for features that you directly measure (that is, features where the **Pointcloud** option is set to **Disabled**).

Create Multiple Features - To select curves on the model to create multiple features, select this check box. For Surface Point features, surfaces are selected instead. Note the following:

- The curves must be contiguous. To select or deselect them, press Ctrl. Consider these examples:

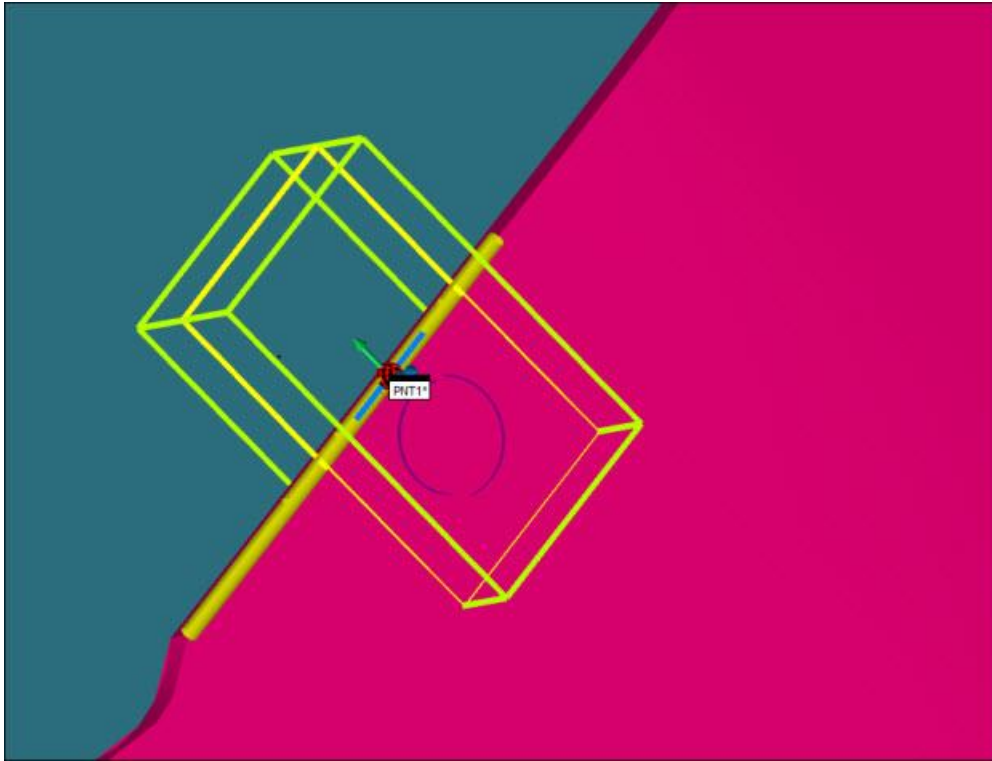


Using Ctrl to select additional contiguous curves



Using Ctrl to select additional contiguous curves

- The first point that is created on the curve is at a distance equal to Horizontal Clipping + Spacer with respect to the start point of the curve itself. This is done on purpose to avoid the extraction of the first point happening off the desired curve. For example:



First curve selection

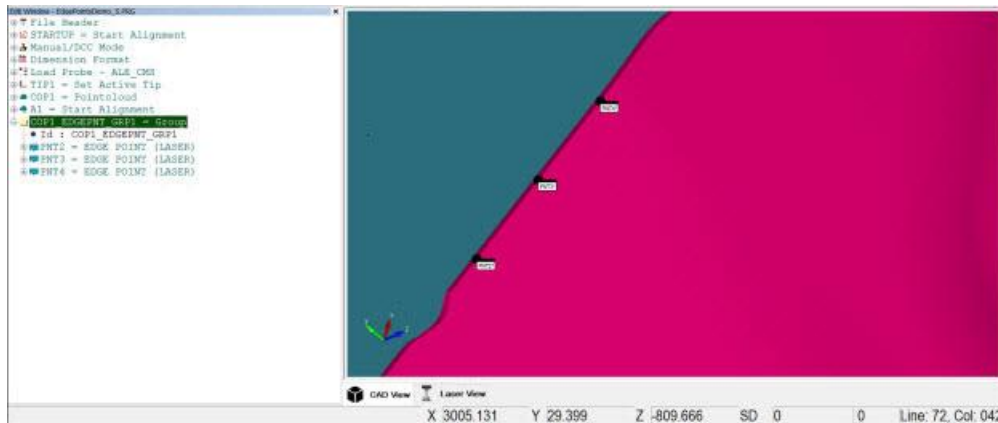
- To enable you to select portions of the CAD curves, use dragging functionality. The features update accordingly.

If you clear the **Create Multiple Features** check box, the edge point has surface and edge vectors set as a starting point to allow you to tune the extraction parameters. This has no effect on the vectors of the features that you create if the **Create Multiple Features** check box is selected. The vectors for those features are created based on the selection of the surface close to the curve. In other words, the surface vector of the resulting features is the one on the surface (close to the curve) that you will click to select the curve itself. Therefore, it is recommended that you do not click exactly on the curve in order to avoid unpredictable vectors (that is, flipped in regard to what is desired).

CAD Selection Method - Select the desired CAD element.

Step - This option enables you to select the spacing along the selected curve or curves between the features you are creating.

A result of a multiple creation appears below:



Execution Modes

With PC-DMIS laser, you can use one of the following execution modes:

- Asynchronous Execution Mode (default mode)
- Sequential Execution Mode

Using Asynchronous Execution Mode

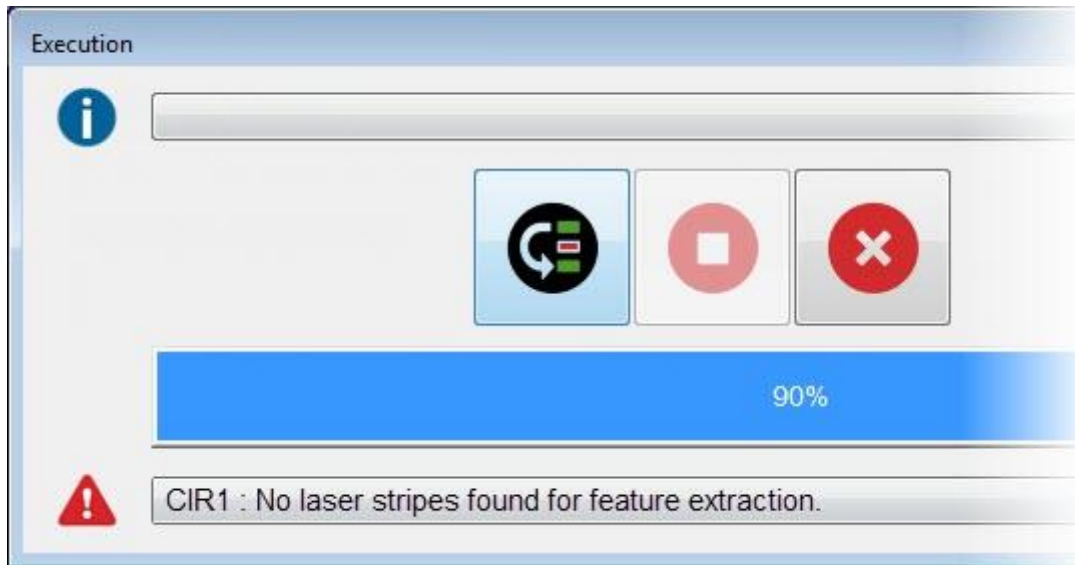
This is the default execution mode. In this mode, in order to speed up execution, the software ignores any feature calculation errors and proceeds to the next feature. If an error occurs during the execution of the measurement routine, the **Execution** dialog box displays these two options:



Cancel - This cancels the execution of the measurement routine.



Skip - This resumes the execution of the measurement routine from the next feature. The skipped feature command turns red in the Edit window.



Execution dialog box

Asynchronous Execution Mode Example

Suppose you have three circles in sequence in your measurement routine. This execution mode behaves as follows:

Scan CIR1.

Begin extraction of CIR1 from its pointcloud.

Scan CIR2.

Begin extraction of CIR2 from its pointcloud.

Scan CIR3.

Begin extraction of CIR3 from its pointcloud.

If CIR2 fails to extract, it generates its error, but because the default execution mode continues execution, the calculation error may appear in the **Execution** dialog box while the machine is already scanning CIR3 or maybe even a later feature. Use Sequential Execution Mode if you want to pause the execution when measurement errors occur.

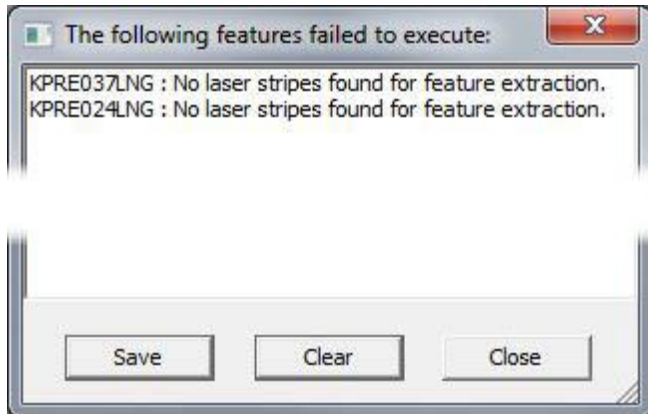
Using the On Error Command with this Mode

In Asynchronous Execution mode, if PC-DMIS encounters an error, and an On Error command has the Skip parameter defined as shown below, it hides the **Execution** dialog box and skips the feature that had the error:

```
ONERROR/LASER_ERROR, SKIP
```


Unless there are critical errors, the Skip parameter lets the measurement routine execute all the way through without anyone tending it.

After the entire measurement routine finishes execution, PC-DMIS displays the features that failed to execute in a dialog box. From that dialog box, you can click on any listed feature to locate the feature command in the Edit window and edit it as needed.



List of failed executed features dialog box

For detailed information on the On Error command, see the "Handling Laser Sensor Errors with On Error Command" topic.

Using Sequential Execution Mode

In Sequential Execution mode, when the measurement routine measures and calculates a feature, it does not proceed with the execution until it finishes calculating the current feature. This execution mode allows you to have concrete information about the problem feature when an error message does appear. In addition, execution stops when a message appears. This may help avoid collisions with the part. Sequential Execution mode is slower than the default (asynchronous execution) mode, but it allows you to monitor errors as they occur.

Generally, you should use this mode when executing a measurement routine for the first time, or when you want to test the machine's movements, laser parameters, or feature calculations.

If an error occurs during Sequential Execution mode, you have the following options in the **Execution** dialog box:



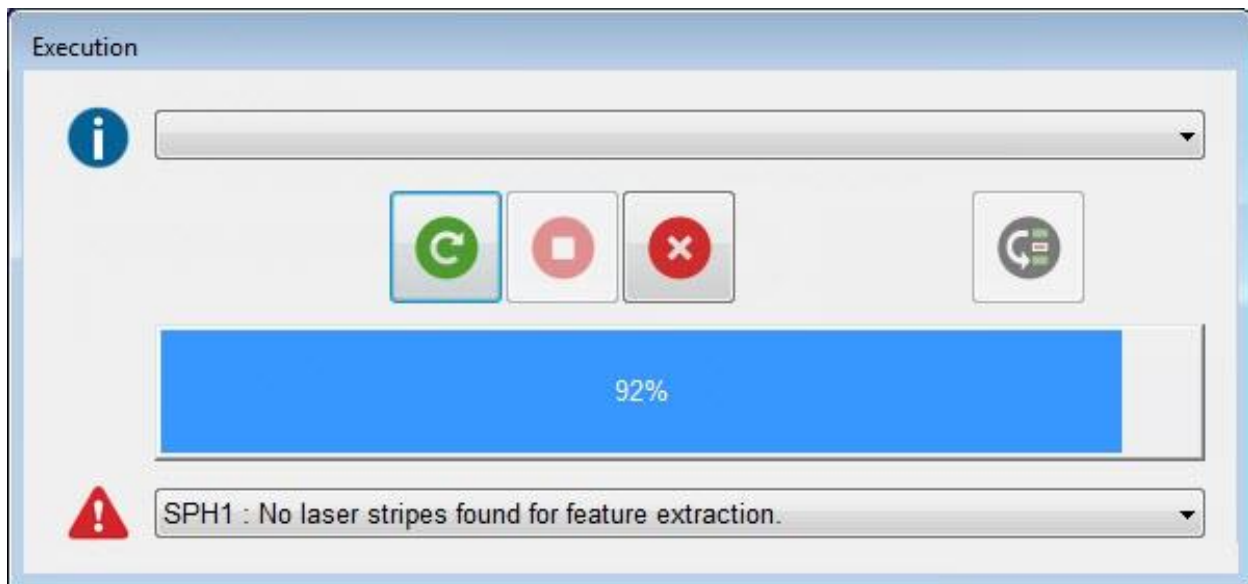
Cancel - This option cancels the execution of the measurement routine.



Skip - This option resumes the execution of the measurement routine from the next feature. The skipped feature command turns red in the Edit window.



Try Again - This option retries the execution. It begins at the failed feature.



Execution dialog box

Enabling Sequential Execution Mode

To enable Sequential Execution mode, select **File | Execute | Sequential Execution** or, from the **Edit Window** toolbar, click the **Sequential Execution** icon.



Sequential Execution icon on the Edit Window toolbar

The software shows this icon in a pressed state when in Sequential Execution mode. PC-DMIS only stays in Sequential Execution mode for the current execution. Afterward, it reverts to the default execution mode.

About the On Error Command

The On Error command does not work with Sequential Execution mode. PC-DMIS ignores any On Error command that it encounters. For detailed information on the On Error command, see the "Handling Laser Sensor Errors with On Error Command" topic.

Using Sound Events

Sound Events provide audible feedback in addition to the visual user interface. This allows you to perform measurement actions if away from the screen. To access the **Sound Events** tab of the **Setup Options** dialog box, select the **Edit | Preferences | Setup** menu item.

When you work with a laser device, these Sound Event options are particularly useful:

Laser Manual Calibration Bottom - This sound plays when calibration measurements for a given field should be taken in the upper region of the sphere.

Laser Manual Calibration Field Counter - This sound plays to indicate in which field measurements should be taken during calibration.

- 1 Beep - Far
- 2 Beeps - Left
- 3 Beeps - Right

Laser Manual Calibration Top - This sound plays when you need to take calibration measurements for a given field in the lower region of the sphere.

Laser Sensor Initialization End - This sound plays at the end of the laser sensor initialization.

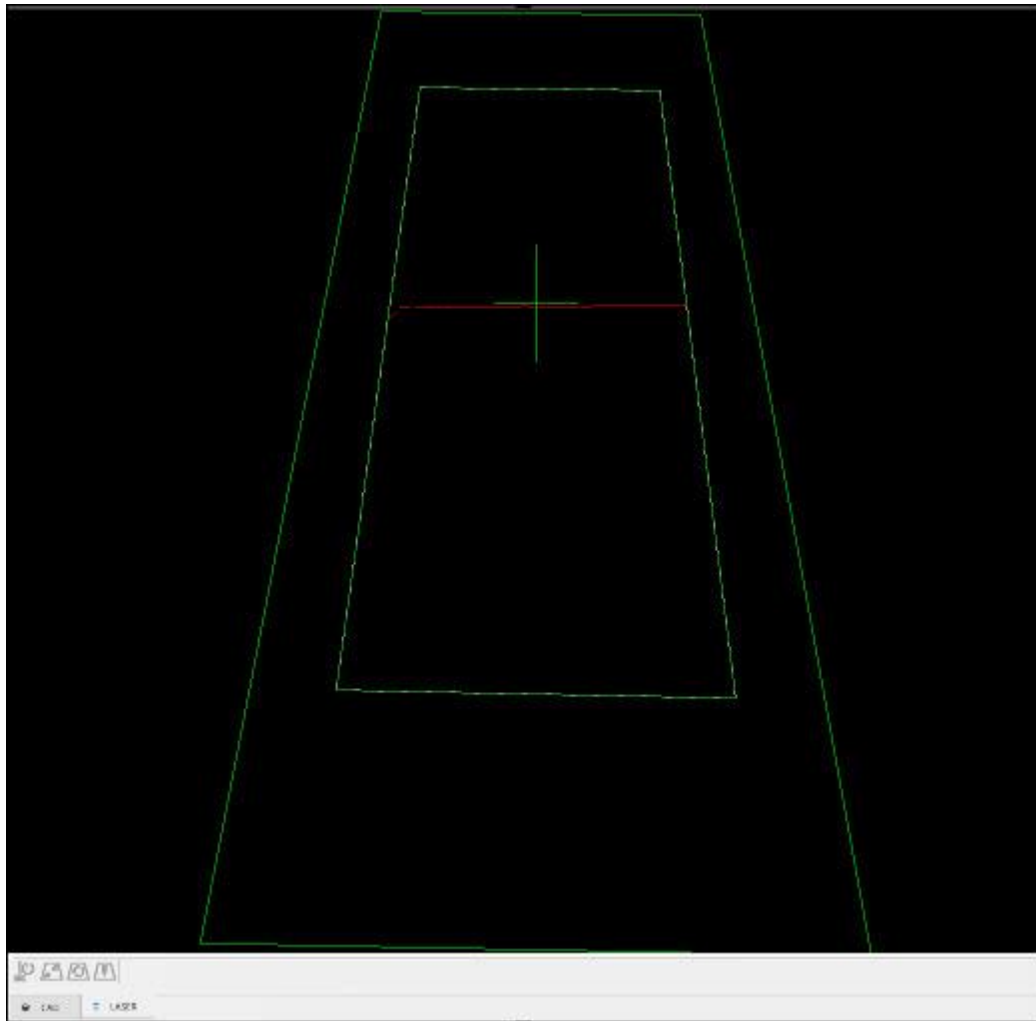
Laser Sensor Initialization Start - This sound plays at the beginning of the laser sensor initialization.

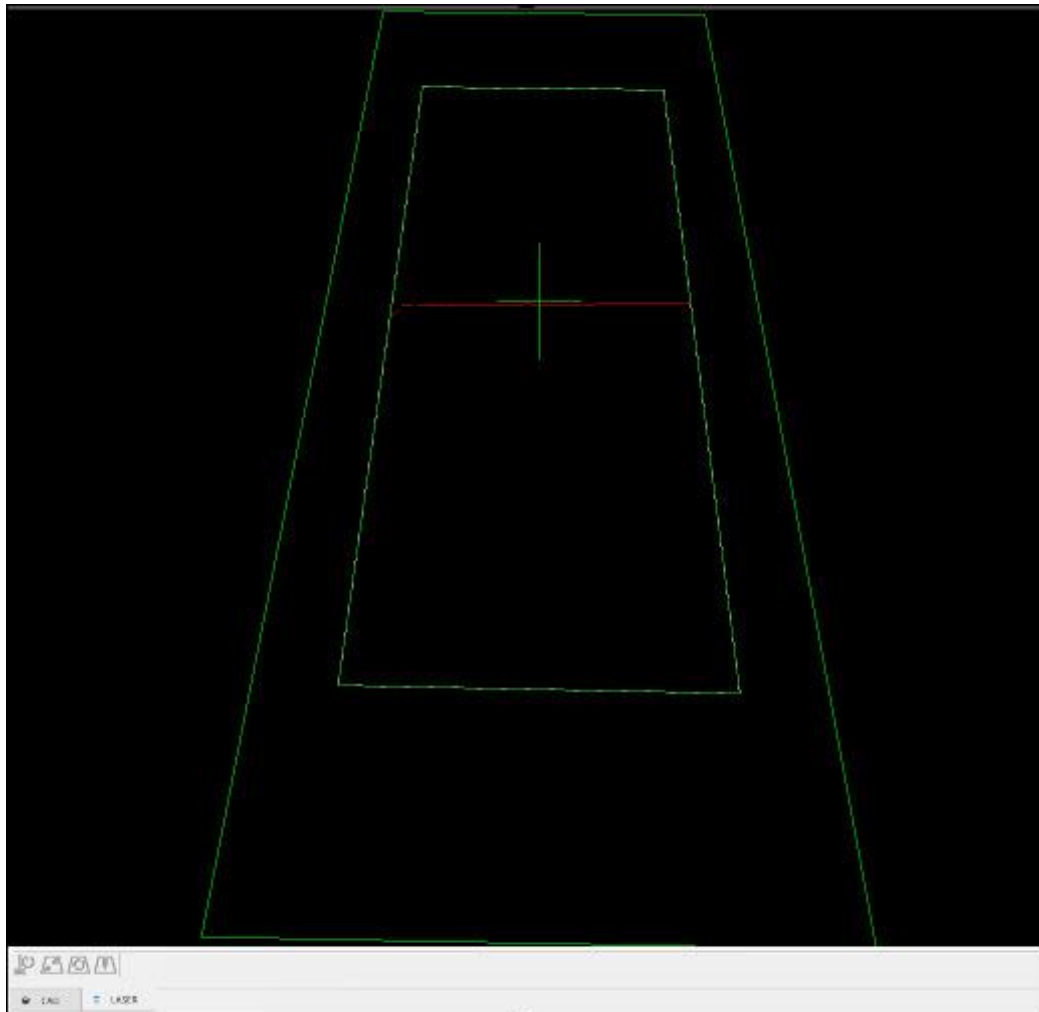
Laser Scan - This sound plays for each new step of the sensor calibration.

Using the Laser View


The Laser View is a view in the Graphic Display window that helps you visualize what the sensor "sees". You can access the Laser View whenever you click on the **Laser** tab.

You can use the Laser View during laser probe calibration, scanning, and auto feature measurement. This tab shows what information PC-DMIS uses. During the scanning process, PC-DMIS disregards any data outside the clipping region rectangle. For more information, see the screen capture in "Laser Probe Toolbox: Laser Clipping Region Properties tab".





Graphic Display window - Laser tab

To turn the laser state on or off in the **Laser** tab, click the **Start/Stop** button () . When you make changes in the **Probe Toolbox**, you need to turn the laser state off and then back on to apply the changes in the **Laser** tab.

Perceptron Sensor Additions



AutoExposure - This button automatically determines the optimum exposure to use for measurement. You need to aim the laser at the part before you click this button. For more information, see "Exposure".

Perceptron and CMS Sensor Addition

If you are using a CMS or Perceptron sensor, these buttons appear:



AutoGain - When the HP-L-5.8 sensor is in range of the part, select this button to learn the best gain setting and update the Probe Toolbox accordingly.



AutoClip - This button automatically sets the clipping according to the data present in the **Laser** tab.



Reset Clipping - This button erases the existing clipping. This resets the entire sensor view for the selected scan zoom mode. For more information, see "Scan Zoom States (for CMS Sensors)".



Center Part - This button centers the part in the sensor's field of view.

In addition, for Perceptron and CMS sensors, you can drag the clipping region with the mouse. This provides an alternative to adjusting the clipping region by typing values in the **Probe Toolbox**.

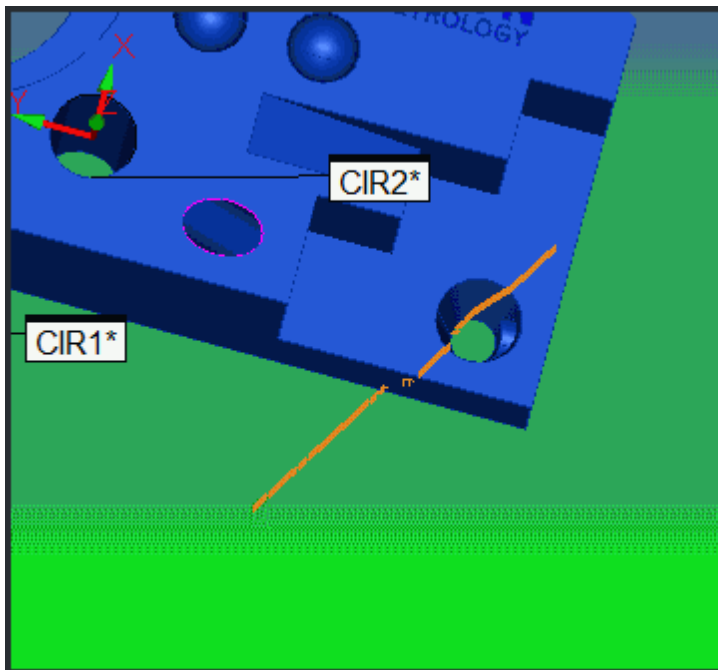
Using the Scan Line Indicator

PC-DMIS Laser displays a colored scan-line indicator in the Graphic Display window to represent the location of the actual beam's scan line in 3D space. The indicator only functions when you run PC-DMIS in Online mode with an actual laser sensor that points at the part in real time.

Click the **Start / Stop** icon in the **Laser** tab to turn the scan-line indicator (and the Laser View) on or off.

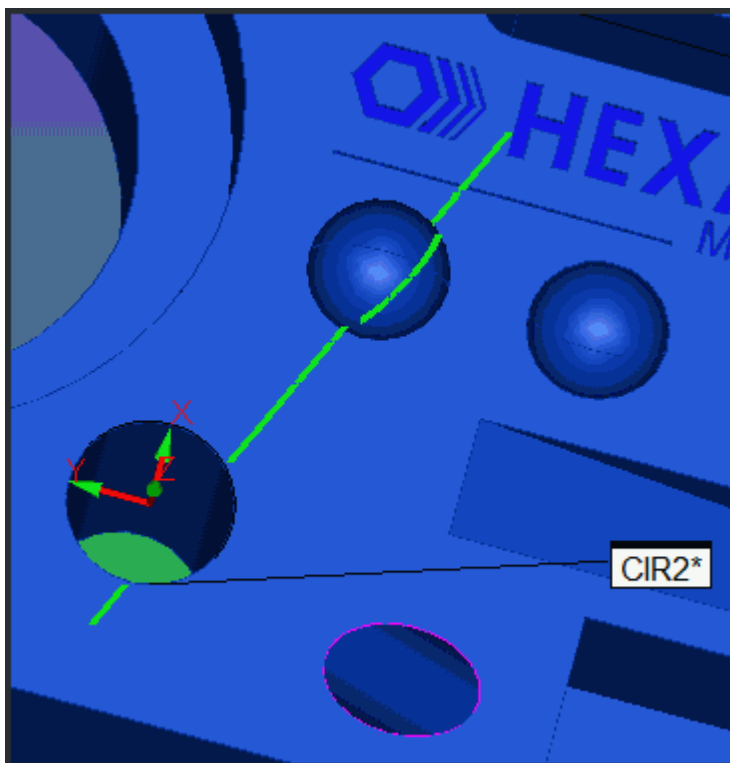


If the beam is within range, it appears in the Graphic Display window and blinks whenever the laser beam pulses. As the beam move toward the part, the indicator begins to change colors. As it nears the desired focal range, it changes colors from red, to orange, then yellow, then yellow-green, and finally to green.



A sample scan-line indicator (in orange) shows that the beam's scan-line position is too far above the part

This green color signifies that the beam is at the optimal distance away from the part for scanning.



A sample scan-line indicator (in green) shows that the beam's scan-line position is at the optimal focal distance

If you move the beam too close to the part, it again moves away from the desired green color and toward a red color.

Understanding the Visualization Tools

PC-DMIS provides you with graphical overlays that it draws on top or around features that you create or edit in the Graphic Display window. These colored overlays give a visual perspective for matching colored parameters or settings in the **Probe Toolbox** and in the **Auto Feature** dialog box.

You can turn these visualization overlays on or off with the **Visualization Tools ON/OFF** icon from the **Laser Scan Properties** tab of the **Probe Toolbox** (**View | Other Windows | Probe Toolbox**).



Visualization Tools ON/OFF icon

Below are some examples. These examples cover all of the possible graphical overlays.

Explanation of Colored Overlays

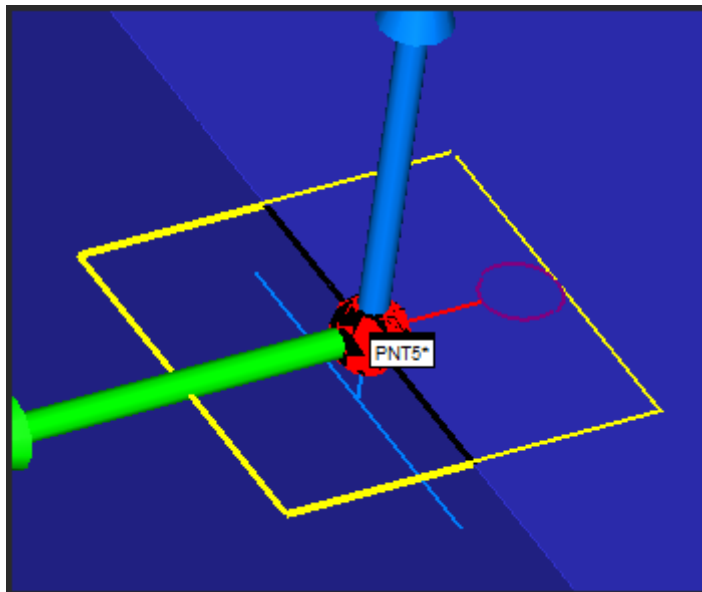
- **Yellow Line or Circle** - The **Overscan** region.
- **Blue Line or Circle** - The feature's **Depth** value.
- **Red Line** - The feature's **Indent** value.
- **Purple Circle** - The feature's **Spacer** value.
- **Pink Circles or Pink Rectangles** - The feature's **Ring Band** value.

Cones and Cylinders Overlays

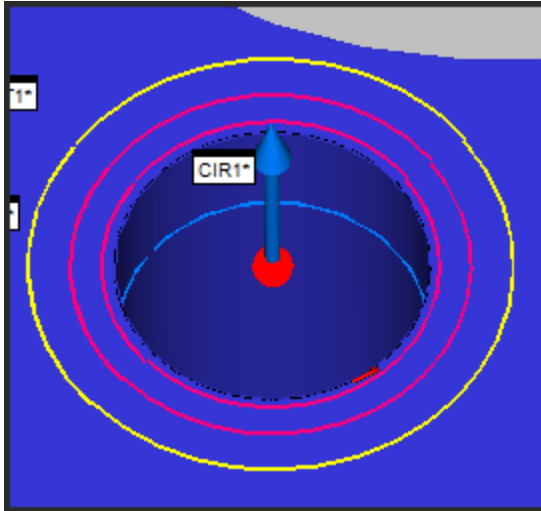
- *DCC Cylinders and Cones* show their bounds (the start and end points plus the **Overscan** value) in a light sea-green color. See the image of the sample DCC cone below.
- *Portable Cylinders and Cones (or Feature Extraction only features)* show their bounds (the start and end points minus the **Vertical Clipping** value) in a lime-green color. See the image of the sample portable cylinder below.

For information on specific parameters or features, see the appropriate topics in the "Creating Auto Features with a Laser Sensor" section of the PC-DMIS Laser documentation.

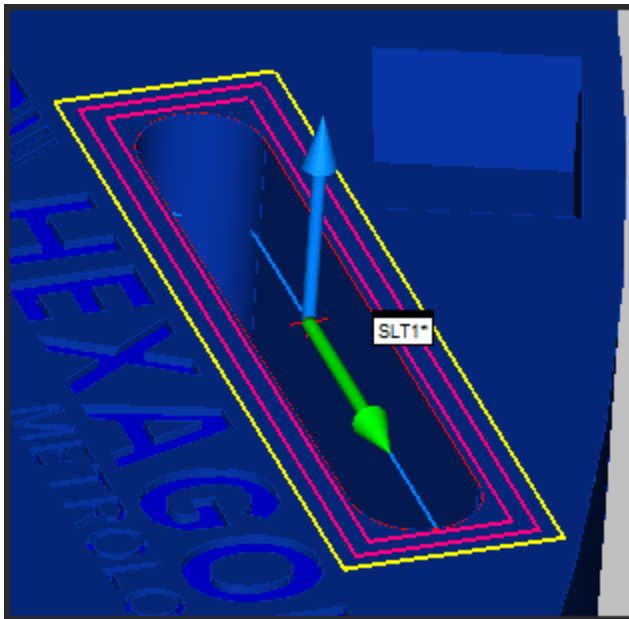
Some Sample Features with Overlays



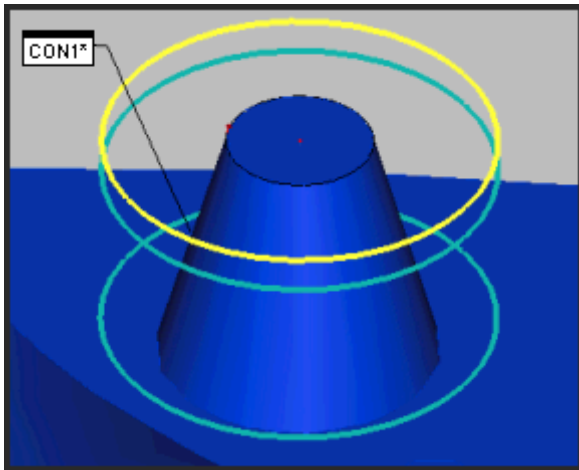
Sample Edge Point



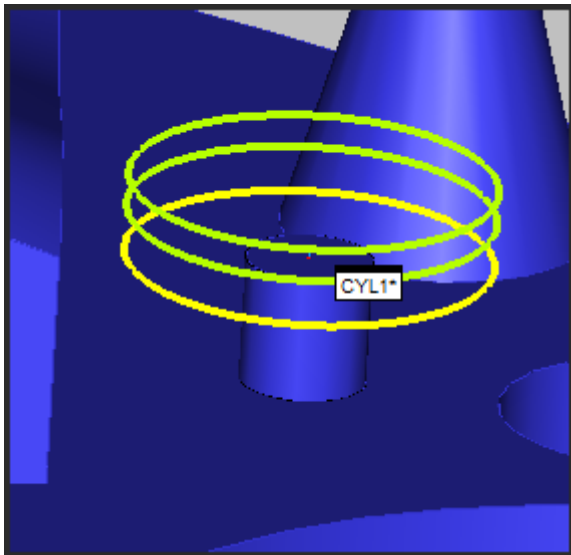
Sample Circle



Sample Slot



Sample DCC Cone



Sample Portable Cylinder

Pointcloud Scanning Colors

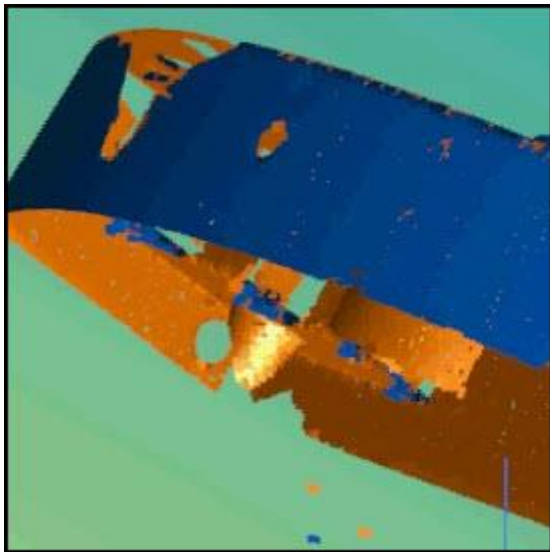
The following colors can help you interpret scanned pointclouds:

Blue - Existing scanned points of the outside of a part. Blue is the default outside color for a pointcloud. For information on how to change this color, see "Manipulating Pointclouds".

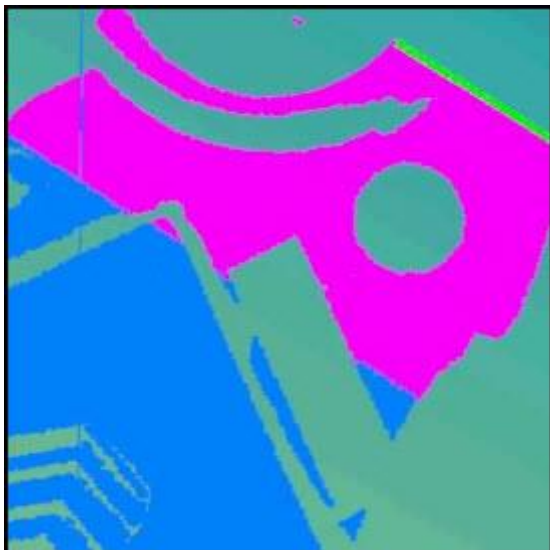
Orange - Existing scanned points of the inside of a part.

Magenta - Points currently being scanned.

Examples



Blue shows existing scanned points from the outside of a part. Orange shows the existing scanned points from the inside of a part.



Magenta shows the points currently being scanned.

Using the Laser Toolbars

To decrease the time it takes to program your part, PC-DMIS Laser offers you a variety of toolbars composed of frequently-used commands. These toolbars can be accessed in two ways.

- Select the **View | Toolbars** submenu, and select a toolbar from the menu.

- Right-click in the PC-DMIS **Toolbar** area, and select a toolbar from the shortcut menu.

For a description of the standard PC-DMIS toolbars, see the "Using Toolbars" chapter in the PC-DMIS Core documentation.

The toolbars specific to Laser functionality are:


Pointcloud Toolbar



Pointcloud Toolbar

The **Pointcloud** toolbar provides all pointcloud operations, features, and functions. It is accessible from the **View | Toolbars | Pointcloud** menu depending on your system's configuration.




 All options may not be available. Some options require specific licensing to enable them.

The following options are available from this toolbar:




Pointcloud: This button opens the **Pointcloud** dialog box you can use to create pointcloud features. For details on this dialog box and how to create pointcloud features, see "Manipulating Pointclouds" in the "Using Pointclouds" chapter of the PC-DMIS Laser documentation.



 **Pointcloud Operator:** This button opens the **Pointcloud Operator** dialog box you can use to perform different operations on Cloud of Points (COP) commands and other Pointcloud operator commands. For details on this dialog box and how to create pointcloud operators, see "Pointcloud Operators" in the PC-DMIS Laser documentation.



 **Pointcloud Mesh:** This button opens the **Mesh Command** dialog box you can use to define a mesh command for pointclouds. For details, see "Creating a Mesh Feature" in the PC-DMIS Laser documentation. This option is only available if you have the Mesh and Big COP licenses.



Pointcloud Data Collection Parameters: This button opens the **Laser Data Collection Settings** dialog box you can use to define data filtering and an exclusion plane for your pointcloud data. For details on this dialog box, see the "Laser Data Collection Settings" topic.



Pointcloud Boolean Operation: This button opens the **Pointcloud Operator** dialog box with the Boolean operator selected. For details on the dialog box and on creating a Boolean pointcloud operator, see "BOOLEAN" in the "Pointcloud Operators" chapter of the PC-DMIS Laser documentation.



Cross Section Pointcloud: This button opens the **Pointcloud Operator** dialog box with the CROSS SECTION option selected. Click the drop-down arrow to display the **Pointcloud Cross Section** toolbar:



For details on pointcloud cross sections and how to use the **Pointcloud Cross Section** toolbar, see "CROSS SECTION" in the "Pointcloud Operators" chapter of this documentation.



Clean Pointcloud: When you click this button, the CLEAN operation immediately eliminates outlier COP points based on the default MAX DISTANCE of the points to the CAD. If the distance of a point is greater than the **Max distance** value, the software considers the point as an outlier or not belonging to the part. To use this operation, you must establish at least a rough alignment (see "Creating a Pointcloud/CAD Alignment") and a CAD model. For details on the CLEAN pointcloud operator, see "CLEAN" in the PC-DMIS Laser documentation.



Empty Pointcloud: When you click this button, PC-DMIS immediately removes all the data from the currently selected COP. Be aware that this change is permanent, so use with caution. For details on the EMPTY pointcloud operator, see "EMPTY" in the PC-DMIS Laser documentation.



Filter Pointcloud: This button opens the **Pointcloud Operator** dialog box with the FILTER operation selected. The operation filters data to a smaller subset of points. For details on the FILTER pointcloud operator, see "FILTER" in the PC-DMIS Laser documentation.



Pointcloud Export: This button opens the **Pointcloud Operator** dialog box for the currently selected export option.

Click the drop-down arrow to display the **Pointcloud Export** toolbar:



The options available are:



Export Pointcloud in IGES Format: This button opens the **Pointcloud Operator** dialog box with the EXPORT IGES operation selected. The Export IGES operation exports the data in a COP or operator command in IGES format to an IGES file. For details on exporting supported file types, see "Pointcloud EXPORT" in the PC-DMIS Laser documentation.



Export Pointcloud in XYZ Format: This button opens the **Pointcloud Operator** dialog box with the EXPORT XYZ operation selected. The Export XYZ operation exports the data in a COP or operator command in XYZ format to an XYZ file. For details on exporting supported file types, see "Pointcloud EXPORT" in the PC-DMIS Laser documentation.



Export Pointcloud in PSL Format: This button opens the **Pointcloud Operator** dialog box with the EXPORT PSL operation selected. The Export PSL operation exports the data in a COP or operator command in PSL format to a PSL file. For details on exporting supported file types, see "Pointcloud EXPORT" in the PC-DMIS Laser documentation.



Pointcloud Import: This button opens the **Pointcloud Operator** dialog box for the currently selected import option.

Click the drop-down arrow to display the **Pointcloud Import** toolbar:



The options available are:



Import Pointcloud in XYZ Format: This button opens the **Pointcloud Operator** dialog box with the IMPORT XYZ operation selected. The Import XYZ operation imports data from an external file into a COP command in the XYZ format. For details on importing supported file types, see "Pointcloud IMPORT" in the PC-DMIS Laser documentation.



Import Pointcloud in PSL Format: This button opens the **Pointcloud Operator** dialog box with the IMPORT PSL operation selected. The Import PSL operation imports data from an external file into a COP command in the PSL format. For details on importing supported file types, see "Pointcloud IMPORT" in the PC-DMIS Laser documentation.



Import Pointcloud in STL Format: This button opens the **Pointcloud Operator** dialog box with the IMPORT STL operation selected. The Import STL operation imports data from an external file into a COP command in the STL format. For details on importing supported file types, see "Pointcloud IMPORT" in the PC-DMIS Laser documentation.



Purge Pointcloud: When you click this button, PC-DMIS immediately removes all data points that do not belong to this operator. It is irreversible and affects all other operator commands that refer to the same COP container, so use with caution. For details on the Purge pointcloud operator command, see "PURGE" in the PC-DMIS Laser documentation.



Reset Pointcloud: When you click this button, PC-DMIS immediately reverses the most recent Surface Colormap, Point Colormap, Select or Clean (unless Purge has been done) operations. For details on the Reset pointcloud operator command, see "RESET" in the PC-DMIS Laser documentation.



Select Pointcloud: This button opens the **Pointcloud Operator** dialog box with the Select operator selected. This pointcloud operator provides by default the Polygon selection method. Select the vertices of the polygon and then press the **End Key** to close it. For details on the Select pointcloud operator command, see the "SELECT" topic in the PC-DMIS Laser documentation.



The **Select Pointcloud** option differs from the use of the pointcloud operator as it only applies the function and is not added as a command. To create the command, open the pointcloud operator, and choose the **Select** method.



TCP/IP: This button performs the currently selected operation described below.

Click the drop-down arrow to display the **TCP/IP** toolbar:



The options available are:



TCP/IP Pointcloud Server receive data: This button places PC-DMIS in a "watch" state, where it is ready to receive a pointcloud file from a client application. The client application must initiate sending the pointcloud data. This button only appears when you run PC-DMIS in Offline mode.



TCP/IP Pointcloud Server Connection with Local Copy: This button establishes the connection with the client and sends the point cloud data directly to the client. When the scan finishes, the point cloud data remains inside the measurement routine. For details on the TCP/IP Pointcloud server connection, see "TCP/IP Pointcloud Server".



TCP/IP Pointcloud Server Connection without Local Copy: This button establishes the connection with the client and sends the point cloud data directly to the client. When the scan finishes, the point cloud data is deleted from the measurement routine. For details on the TCP/IP Pointcloud server connection, see "TCP/IP Pointcloud Server".



Pointcloud Alignment: This button opens the **Pointcloud/CAD Alignment** dialog box that you can use to create Pointcloud to CAD and COP to COP alignments. For details, see "Pointcloud/CAD Alignment Dialog Box Description" in the "Pointcloud Alignments" chapter of the PC-DMIS Laser documentation.



Pointcloud Colormap: This button opens the dialog box for the operator shown on the button.

Click the drop-down arrow to display the **Pointcloud Colormap** toolbar:



The **Pointcloud Colormap** toolbar allows you to select between the **Surface Colormap**, **Point Colormap** and **Thickness Colormap** options.

From left to right, the buttons are:



Surface Colormap - This button opens the **Pointcloud Operator** dialog box with the Surface Colormap operator selected. The SURFACE COLORMAP operation applies a colored shading to the CAD model. The software shades the model according to the deviations of the pointcloud compared to the CAD. The Pointcloud Surface Colormap operator uses the colors defined in the **Edit Dimension Color** dialog box, and the tolerance limits specified in the **Upper tolerance** and **Lower tolerance** boxes. For details on the Pointcloud Surface Colormap operator, see "SURFACE COLORMAP" in the PC-DMIS Laser documentation.

You can create multiple surface colormaps in a PC-DMIS measurement routine. However, only one is active. The last surface colormap that was applied and created, or the last one executed, is always the currently active colormap. You can also select which colormap is the active one from the **Colormaps** list box. When you activate a new colormap, PC-DMIS displays its associated scale with tolerance values and any annotations in the Graphic Display window.

To do this, click the **Colormaps** list box and select the colormap from the list of defined Surface or Point Colormap operators:



Point Colormap - This button opens the **Pointcloud Operator** dialog box with the Point Colormap operator selected. The Point Colormap operation

evaluates the deviations of the data points contained in a COP command compared to a CAD object. You can use this command to color the entire Pointcloud, or to display the points as dots, needles and/or text. For details on the Pointcloud Point Colormap operator, see "POINT COLORMAP" in the PC-DMIS Laser documentation.



Thickness Colormap - This button opens the **Pointcloud Operator** dialog box with the Thickness Colormap operator selected. The Thickness Colormap allows you to show and measure the part thickness as a colormap using only the Mesh or Pointcloud (COP) data object. You can also compare the measured thickness to the nominal CAD model thickness. For details on the **Thickness Colormap** option, see "Thickness Colormap" in this documentation.

QuickCloud Toolbar



QuickCloud toolbar

The **QuickCloud** toolbar is only available when PC-DMIS is licensed and is configured as a Portable device. It provides the buttons to complete all the steps from beginning to end for working with COP.

For detailed information on this toolbar, see "QuickCloud Toolbar" in the "PC-DMIS Portable" documentation.



For details on all **Pointcloud** toolbar functions, see "Pointcloud Toolbar" in this documentation.

Mesh Toolbar



Mesh toolbar

The **Mesh** toolbar provides all mesh operations, features, and functions. It is accessible from the **View | Toolbars | Mesh** menu.



The Mesh license must be enabled to use or view this option.

The following options are available from this toolbar:



Mesh: This button opens the **Mesh Command** dialog box that you can use to create mesh features from any number of pointclouds. For details on this dialog box and creating mesh features, see the "Creating a Mesh Feature" topic.



Mesh Operator: This button opens the **Mesh Operator** dialog box that you can use to perform different operations on a Mesh and other Mesh operator commands. For details on the dialog box and creating mesh operators, see the "Creating a Mesh Operator" topic.



Mesh Cross Section: This button opens the **Mesh Operator** dialog box that you can use to create a cross section from an existing mesh. Click the drop-down arrow to display the **Mesh Cross Section** toolbar:



For details on Mesh cross sections and using the **Mesh Cross Section** toolbar, see "Mesh CROSS SECTION Operator" in this documentation.



Import Mesh in STL Format: This button opens the **Mesh Import Data** dialog box that you can use to import an STL mesh data file. If a Mesh object does not exist in the PC-DMIS Edit window, then a new Mesh object is created, and the software imports the STL data. If a Mesh object already exists in the PC-DMIS Edit window, then the software adds the STL data to the Mesh object.

For details, see the "Mesh IMPORT Operator" topic.



Export Mesh in STL Format: This button opens the **Export Mesh Data** dialog box that you can use to export a Mesh in an STL ASCII or STL Bin file format.

For details, see the "Mesh EXPORT Operator" topic.



Empty a Mesh: This button empties the first mesh relative to the cursor position in the Edit window.



Once you apply this command to a Mesh, there is no way to restore the Mesh data. You cannot click **Undo** to restore the lost data.

For details, see the "Mesh EMPTY Operator" topic.

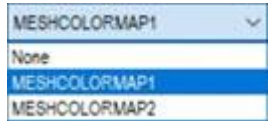


Colormap a Mesh: This button opens the **Mesh Operator** dialog box that you can use to create a Mesh COLORMAP operator. For details, see the "Mesh COLORMAP Operator" topic.

The **Colormap a Mesh** operation applies a colored shading to the selected mesh. PC-DMIS shades the model according to the deviations of the mesh compared to the CAD. The **Colormap a Mesh** operation uses the colors defined in the **Edit Dimension Color** dialog box and the tolerance limits specified in the **Upper tolerance** and **Lower tolerance** boxes. For details on the **Colormap a Mesh** operator, see the "Mesh COLORMAP Operator" topic in the PC-DMIS Laser documentation.

You can create multiple colormaps in a PC-DMIS measurement routine. However, only one is active. The last colormap that was applied and created (pointcloud surface colormap or mesh colormap), or the last one executed, is always the currently active colormap. You can also select which colormap is the active one from the **Colormaps** list box. When you activate a new colormap, PC-DMIS displays its associated scale with tolerance values and any annotations in the Graphic Display window.

To do this, click the **Colormaps** list box and select the colormap from the list of defined colormap operators:



Click the drop-down arrow to display the **Mesh Colormap** toolbar:



The **Mesh Colormap** toolbar allows you to select between the **Colormap a Mesh** and **Thickness Colormap** options. For details on the **Thickness Colormap** option, see "Thickness Colormap" in this documentation.




Mesh Alignment: This button opens the **Mesh/CAD Alignment** dialog box. It is used to create Mesh to CAD alignments.

For details, see the "Mesh ALIGNMENT" topic.



Receive a mesh from OptoCat: When you click this button ON, PC-DMIS is placed in a state where it is waiting and ready to receive a mesh from the OptoCat application. When the **Receive a mesh from OptoCat** button is ON, it has a

darker background color: . For details on how this works, see the "Receive a Mesh from OptoCat" topic.

Using Pointclouds


The Cloud of Points command (COP) allows you to store XYZ coordinate data that can come directly from a laser sensor through one or more referring scan commands. You can also enter data directly into a COP from other PC-DMIS features or external data files.

You can add pointclouds to your measurement routine in these ways:

- Select the **File | Import | Pointcloud** submenu, and select a data file to import (XYZ, PSL, or STL).

STL: The STL file type is the same file type that is covered in the "Importing an STL File" topic of the PC-DMIS Core documentation, except that instead of importing the file as a CAD model, it imports the file as a pointcloud.

XYZ: The XYZ file type is the same file type that is covered in the "Importing an XYZ File as CAD Data" topic of the PC-DMIS Core documentation, except that instead of importing the file as a CAD model, it imports the file as a pointcloud.

- Select the **Insert | Pointcloud | Feature** menu item to open the **Pointcloud** dialog box.
- Manually type the COP command into the Edit window. Press **F9** on the COP command in the Edit window to open the **Pointcloud** dialog box. For information on the COP command mode text, see "COP Command Mode Text".
- From the **Pointcloud** toolbar, click the **Pointcloud** button () to open the **Pointcloud** dialog box.

For information on how to manipulate pointclouds from the **Pointcloud** dialog box, see the "Manipulating Pointclouds" topic.

PC-DMIS uses additional, laser sensor-related commands and tools that support pointcloud functionality. They are:

- Pointcloud Operators
- Pointcloud Alignments
- Pointcloud Point Information
- Laser Data Collection Settings



Your LMS license or portlock must contain a license with either the **Small COP (COP)** or **Big COP** option to use COP capability.

About the Small COP (COP) and Big COP Laser Options

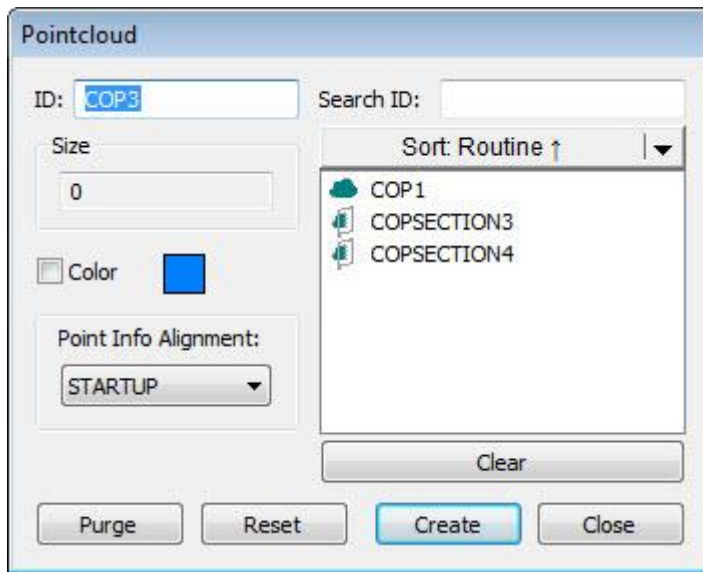
The PC-DMIS CAD++ license includes the **Small COP (COP)** option. It provides limited pointcloud functionality.

The PC-DMIS Laser option (does not include Vision probes) includes the **Big COP** option. This option provides full pointcloud functionality. You can purchase it separately for other configurations.

The following list describes the differences in functionality between the **Small COP (COP)** and **Big COP** licensing options:

- If **Small COP (COP)** is enabled, and **Big COP** is disabled, PC-DMIS limits the pointcloud size to 500,000 points. The pointcloud automatically re-sizes to stay within the limit.
- Pointcloud alignment is enabled only when **Big COP** is enabled.
- Meshing is enabled only if both **Big COP** and **Mesh** are enabled.
- If the **Small COP (COP)** and **Big COP** options are disabled, pointcloud functionality is disabled.

Manipulating Pointclouds



Pointcloud dialog box



The **Pointcloud** dialog box only has an effect if the COP command contains data.

To open the **Pointcloud** dialog box, click the **Pointcloud** button () on the **Pointcloud** toolbar, or select **Insert | Pointcloud | Feature**.

The dialog box contains the following elements:

ID - Contains a unique identity of the pointcloud command being edited.

Search ID - If there's a long list of operators defined, you can search using the **Search ID** box to locate specific operators in the list. When you start to enter the operator's ID into the box, the list automatically filters based on your entry.

Size - Total number of points in the pointcloud.

Color - Sets the color for the scanned points in the pointcloud on the outside of a part. To change the pointcloud color, select the **Color** check box and then click on the **Color** box to select a color from the **Color** dialog box. For additional information on pointcloud colors, see "Pointcloud Scanning Colors".

Commands List - This area contains the list of features or scans that send data to the COP command in the dialog box. A **Sort** functionality is available to organize the list by **ID**, **Type**, **Routine**, or **Time**. Select the option from the drop-down list and then click the **Sort** button.

Point Info - With the **Pointcloud** dialog box open, you can click on a pointcloud point in the Graphic Display window to open the **Pointcloud Point Information** dialog box. The **Pointcloud Point Information** dialog box contains information about the point with respect to the alignment. This box contains the point's numerical ID, its coordinates, and the estimated normal of the point. Corresponding CAD points also appear with CAD coordinates and CAD normal. Finally, the deviation between the point and CAD is shown with the scale for the deviation arrow specified in the dialog box. Point selection has no associated operator command. With the **Pointcloud Point Information** dialog box open, two scenarios are possible when you click the **Create Point** button:

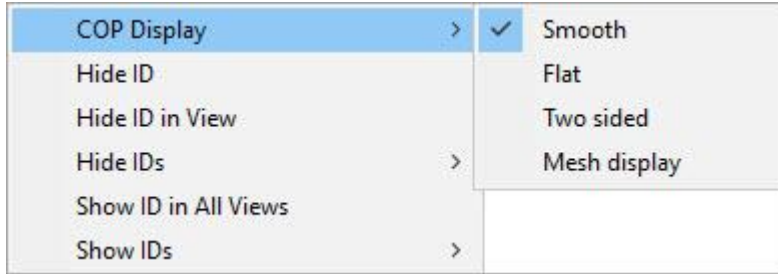
- If there is a CAD model in the measurement routine, and the pointcloud is aligned, a **Laser Surface Point** is created, inserted, and resolved at the selected position.
- Otherwise, a **Constructed Offset** point is created and inserted in the measurement routine.

Purge / Reset - The **Reset** button restores all the data stored in a COP command. The **Purge** button permanently deletes all the data in a cloud of points that is not currently shown, selected, or filtered. This causes the cloud of points to only keep the visible data.

For information on viewing pointcloud point deviation information, see "Pointcloud Point Information".

Pointcloud Graphical Representation

You can set the graphical representation of a selected pointcloud (COP). PC-DMIS stores the setting when you save the measurement routine. To do this, right-click a COP in the Edit window, or right-click the COP label in the Graphic Display window to view the **COP Display** menu options:



The **COP Display** options are:

Smooth: This option provides a shaded appearance using the defined COP color.



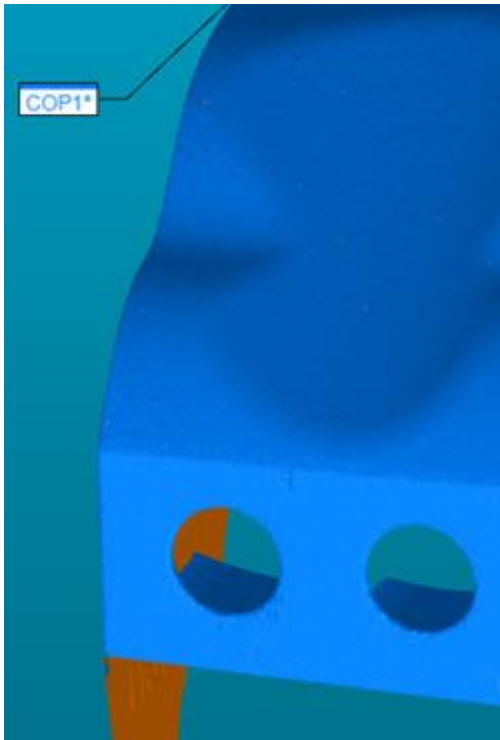
Example of the Pointcloud Display set to Smooth

Flat: This option shows the COP in a non-shaded graphical view. This selection requires the least amount of graphical memory.



Example of the Pointcloud Display set to Flat

Two sided: This option shows a shaded appearance where the scanned side of the part is the defined COP color, and the non-scanned side is the contrasting color.



Example of the Pointcloud Display set to Two sided

Mesh display: This option allows the software to show the pointcloud as a Mesh display.



Example of the Pointcloud Display set to Mesh display



The **Mesh display** option is only available if you have the Mesh license and you scanned the COP using the **Mesh display** option (Portable only). For details, see "Pointcloud Display Section".

The **Mesh display** is a display setting only. The underlying data is a pointcloud.

However, the **Mesh display** is lost and the display reverts to points if you edit the COP (for example, if you perform any COP operation on the pointcloud).

COP Command Mode Text

The COP command inside the Edit window's Command Mode looks like this:

```
COP1 =COP/DATA, SIZE=0
REF, ,
```

The COP command must precede any scan referring to it in the measurement routine.



For example, `REF, SCN2` shown below points to the `SCN2` scan and uses its data:

```
COP2 =COP/DATA, SIZE=0
REF, SCN2, ,
```



You can have more than one scan refer to a COP command.



Be aware that if you cut a COP command and paste it again, the resultant command gets pasted without the data points. If you need to move your COP command to a different location in the Edit window, you need to re-create the COP command at the desired location and delete the earlier one.

Pointcloud Point Information

With the **Pointcloud Point Information** dialog box, you can view point-specific information.

To access this dialog box:

1. Click the COP command in the Edit window to select it and then press the F9 key. The **Pointcloud** dialog box for the COP command appears.
2. Click a point on the cloud of points (COP) in the Graphic Display window. The **Pointcloud Point Information** dialog box appears.

Pointcloud		CAD	
	Point	Normal	
X:	41.764	0.3120192	41.768
Y:	15.107	0.0281713	15.107
Z:	14.217	0.9496580	14.228

Deviation: -0.013
 Thickness: 0
 Scale: 10

Create Point Done

Pointcloud Point Information dialog box

From this dialog box, you can view the **XYZ** and the **Normal** point vector values for the Pointcloud point, as well as the **ID** for the selected point. It also shows the corresponding CAD's **XYZ** and **Normal** vector values.

Deviation - Displays the distance from the Pointcloud point to the corresponding CAD point.

Thickness - The software adds this value to the deviation from the CAD value that it computes when you click on a Pointcloud point. This value is useful, for example, if you have a CAD surface model and you want to add a material thickness.

Scale - This value determines the scale that the deviation arrow uses in the Graphic Display window. For example, A scale of 10 displays an arrow with a length that is ten times the length of the deviation.

The deviation arrow appears when you select a point from the Graphic Display window. The arrow indicates the direction of the point deviation from the CAD.



Point Deviation Arrow

Create Point button - This creates a constructed offset point for the selected point. The software names the constructed offset point with the following convention and then adds the point to the measurement routine: **<pointcloud name>_P<point ID>** (for example, COP1_P185048).



If you use a laser sensor when you click **Create Point**, the software creates a laser surface point instead of a constructed offset point.




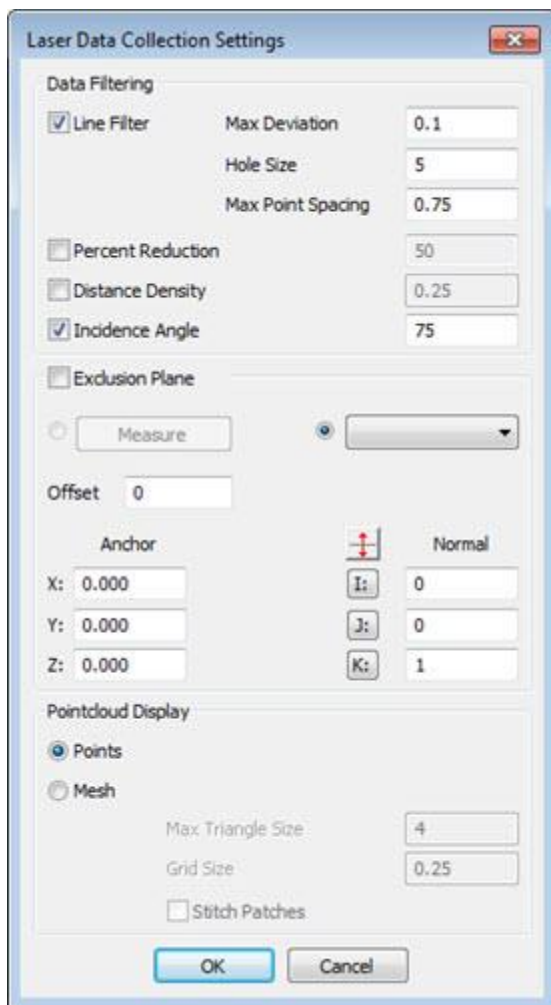
Constructed Point from Pointcloud

Using Point Data for Auto Features

With the **Auto Feature** dialog box open, you can click on desired points from the pointcloud to provide input data for a given auto feature. For more information, see "Auto Feature Extraction".

Laser Data Collection Settings

Access the **Laser Data Collection Settings** dialog box (**Operation | Pointcloud | Data Collection**) or click the **Pointcloud Data Collection Parameters** button () on the **Pointcloud** toolbar or **QuickCloud** toolbar.



Laser Data Collection Settings dialog box

The **Laser Data Collection Settings** dialog box allows you to define the types of data filtering, the exclusion plane, and the pointcloud display for laser scanned data.

Data Filtering Section

Data Filtering		
<input checked="" type="checkbox"/> Line Filter	Max Deviation	0.1
	Hole Size	5
	Max Point Spacing	0.75
<input type="checkbox"/> Percent Reduction		50
<input type="checkbox"/> Distance Density		0.25
<input checked="" type="checkbox"/> Incidence Angle		75

Data filtering allows real-time filtering of the data. It removes the data as you scan.

The **Data Filtering** section provides the following options:

Line Filter: A real-time filter for individual lines, which provides smoothing and point reduction of incoming data from the laser sensor.

Mark the **Line Filter** check box to enable these options:

Max Deviation: As each incoming scan line is evaluated, points may be moved or smoothed in relation to their neighboring points. This setting defines the maximum allowed value that a point may be moved or smoothed.

Hole Size: When the software evaluates a scan line and detects a hole or gap of the specified size (or larger), the filter treats the scan segments as separate lines. In most cases, this can be set to the size of the smallest hole on the physical part.

Max Point Spacing: When analyzing the incoming scan data and reducing the number of points, this setting defines the maximum distance between two consecutive points. If the scan surface is curved, the resulting point spacing is typically smaller than the **Max Point Spacing** value.

When this parameter is set to zero, no point reduction is done. Typically, this value should be set to less than 1/3 of the hole size.

The **Max Point Spacing** setting determines the resolution of the scanned points. For most parts, the default values listed below can be used. To obtain higher resolution when scanning parts with small details, a smaller **Max Point Spacing** may be used. Using a smaller **Max Point Spacing** results in fewer points being filtered out and increases the total COP size.

	Max Point Spacing
--	--------------------------

Large Details	1 mm / 0.03937 inch
Default	0.75 mm / 0.02953 inch
Small Details	0.5 mm / 0.01968 inch
Fine Details	0.25 mm / 0.00984 inch

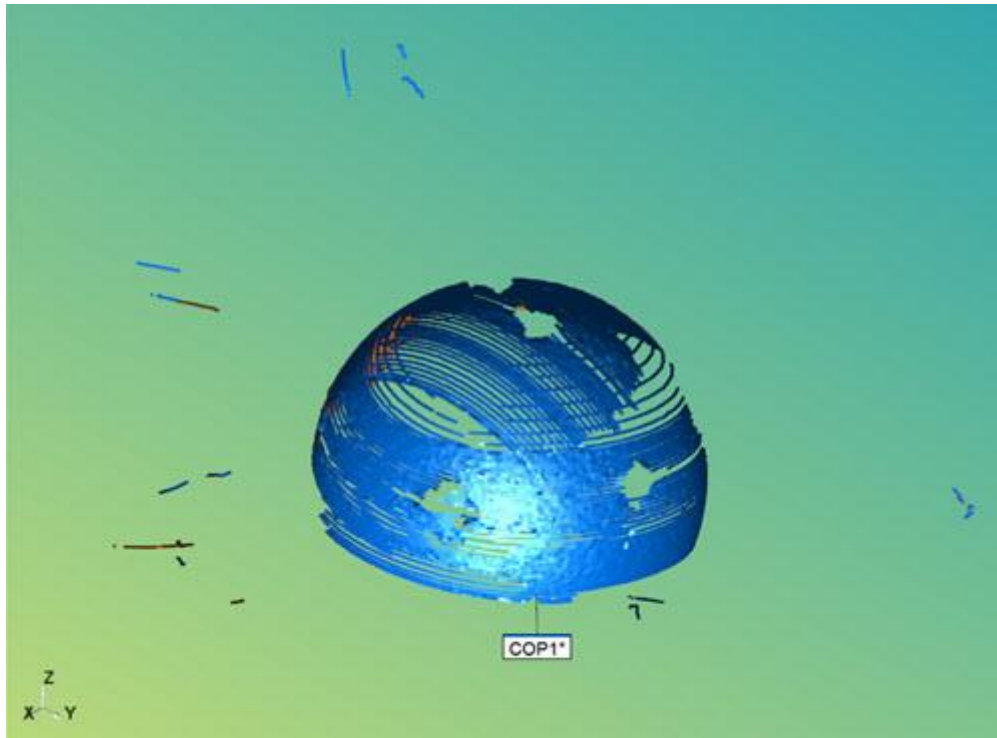
Percent Reduction: Removes a percentage of the pointcloud data collected.

1. Select the **Percent Reduction** option and in the box to its right, type a percentage value between 0-100. The value is the percent of the collected pointcloud data you want the software to filter out. If you enter zero, no filtering takes place.
2. Click **OK** to apply this to your measurement routine.

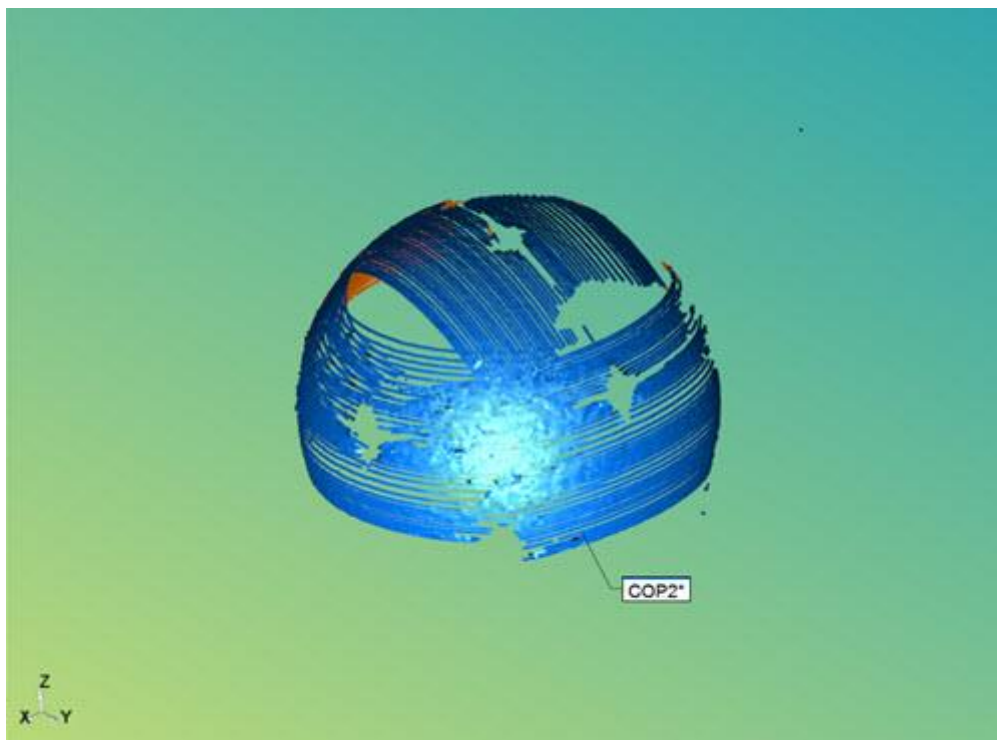
Distance Density: Provides a filtering based on the point distance value. If the distance between a point and its neighboring points is less than this value, the software discards the point. This option becomes available if you select the **Points** option in the **Pointcloud Display** section of the dialog box.

1. Select the **Distance Density** option and in the box to its right, type a distance value in the measurement routine units. Values greater than or equal to zero are valid. 1 mm is the default value. If your measurement routine uses inches, the software converts 1 mm to inches.
2. Click **OK** to apply the filtering.

Incidence Angle: Filters out all scanned points that have an incident angle greater than the entered value. The **Incidence Angle** check box is marked by default with a default value of 75. The angle is calculated between the estimated surface normal and the scan direction of the laser sensor. The smaller the value, the more points are filtered out.



Shiny sphere with no incidence angle applied



Shiny sphere with incidence angle on at default value 75


The **Incidence Angle** filter can be applied real-time while scanning. During scanning, the angle of the scan line relative to the measured surface is determined; any points outside the specified angle are automatically removed and discarded.

Exclusion Plane Section

You can use exclusion planes to remove all points within the defined area of the plane. To enable this feature, select the **Exclusion Plane** check box.

When the **Exclusion Plane** check box is selected, the software activates the defined exclusion plane. If the icon on the toolbar is in the pushed state, filtering is enabled. Once activated, the software uses the exclusion plane the next time the measurement routine is executed.



You can tell when the exclusion plane is active in your measurement routine by how the **Pointcloud Data Collection Parameters** button () appears on the **QuickCloud** or **Pointcloud** toolbar. If the button appears to be pressed in, the exclusion plane is active; otherwise, it is not active.

There are three ways to define the exclusion plane:


- **Measure**

Use a contact probe or laser sensor to measure the exclusion plane.

Click the **Measure** button and then take three hits with a contact probe to measure the exclusion plane. With a laser sensor, scan the area of the plane. If an alignment already exists, the plane is automatically defined in that alignment. If not, the plane is defined using the machine coordinates. If that changes, you need to redefine the plane.

- **Entering the XYZ and IJK values**
- You can also define the exclusion plane by its normal vector and an anchor point. The exclusion plane is independent of the data filtering.

To define an exclusion plane:

1. Edit the XYZ anchor positions if necessary.
2. Click the **I**, **J**, or **K** normal button that your plane is relative to, and edit the value if necessary. To automatically change the direction of the normal value, you can click the **Reverse Direction** button .
3. If PC-DMIS is in Online mode, you can click the **Measure** button to measure your defined exclusion plane.
4. Click **OK** to save your settings.

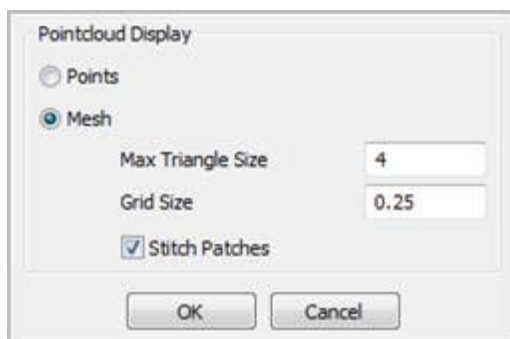
- **Select an existing plane**

Select an existing plane (a plane that already exists in the measurement routine) from the **Exclusion Plane Feature** list. The Anchor and Normal (vector) fields update accordingly.

By selecting an existing plane, when the measurement routine is re-executed and the plane is re-measured, this becomes the new exclusion plane used for the COP. This is useful for portable devices if the device is moved or if the part is moved to a different surface.

Offset - This offsets the plane in the defined Normal direction by the value entered (in measurement routine units).

Pointcloud Display Section



The **Pointcloud Display** section allows you to display the pointcloud as points or as a mesh when performing laser scans. It facilitates the identification of areas not covered with data.

Points - This option displays the pointcloud as a set of points. The **Distance Density** filter in the **Data Filtering** section of the dialog box is enabled when this option is selected. It is used to define valid point distances for the points used to create the pointcloud.

Mesh - This option causes laser data to appear as a mesh during scanning. The software displays the current scan pass as a pointcloud and previous passes as a mesh. This option is only available for Portable systems.



The Mesh display is relative to the orientation of the laser sensor. While scanning, if the laser sensor orientation changes more than 25 degrees in a single scan pass, the software meshes the collected data, and automatically creates a new scan.

The **Max Triangle Size** and **Grid Size** values define the displayed mesh. After you scan, the software displays the data as a mesh until you close and reopen the measurement routine. The data then appears as a pointcloud. The mesh display functionality requires the Mesh license.

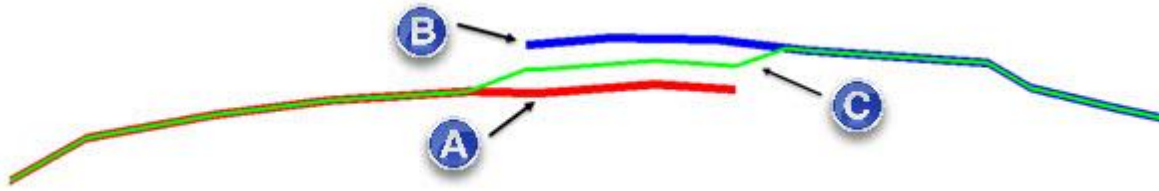
- If scanning speed is slow, and more than one point is in a grid square, PC-DMIS keeps the best point.
- If scanning speed is fast, it is possible to have a grid square without any data, which may cause gaps in the displayed mesh.

Max Triangle Size - This value determines the largest possible triangle in the mesh display. If the distance between any two points is greater than this value, the software does not create any triangles. If there are hole features on your part, you typically need to set this value to be slightly smaller than the smallest hole. This prevents the mesh from filling the hole.

The default value for the **Max Triangle Size** is 5 mm. The software converts this to inches if your measurement routine is using that unit. Valid range values depends on the size of the part.

Grid Size - This value defines the size of the triangles used to create the mesh. This value also affects the mesh's resolution and how refined it appears. The smaller the value, the more time it takes to generate the mesh but the higher the resolution of the resulting mesh. Be aware that this value is critical; it can affect the speed of data collection if it is set too small.

Stitch Patches check box - When scanning as **Mesh** display and the **Stitch Patches** check box is marked, multiple scan passes are blended and the overlapping data is removed.

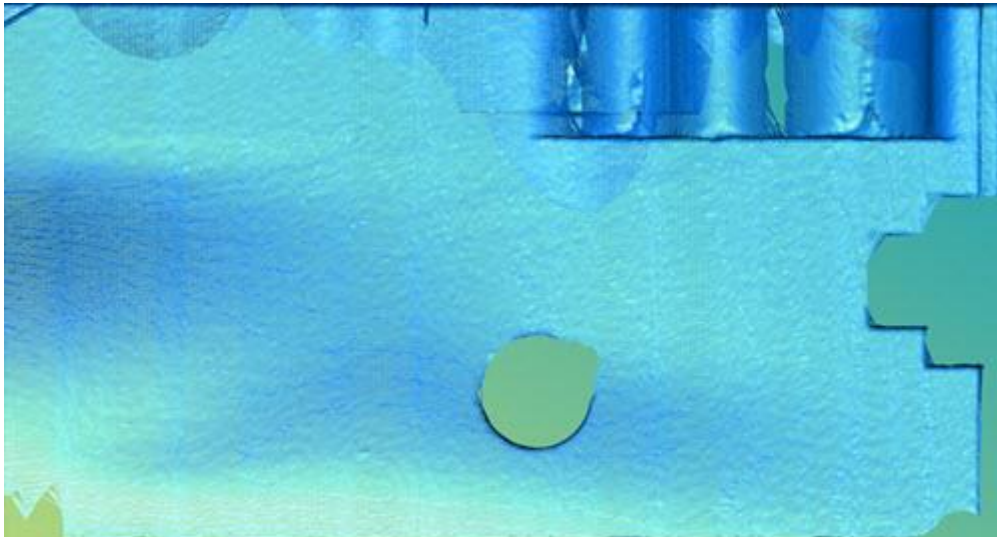


(A) - Scan Pass 1

(B) - Scan Pass 2

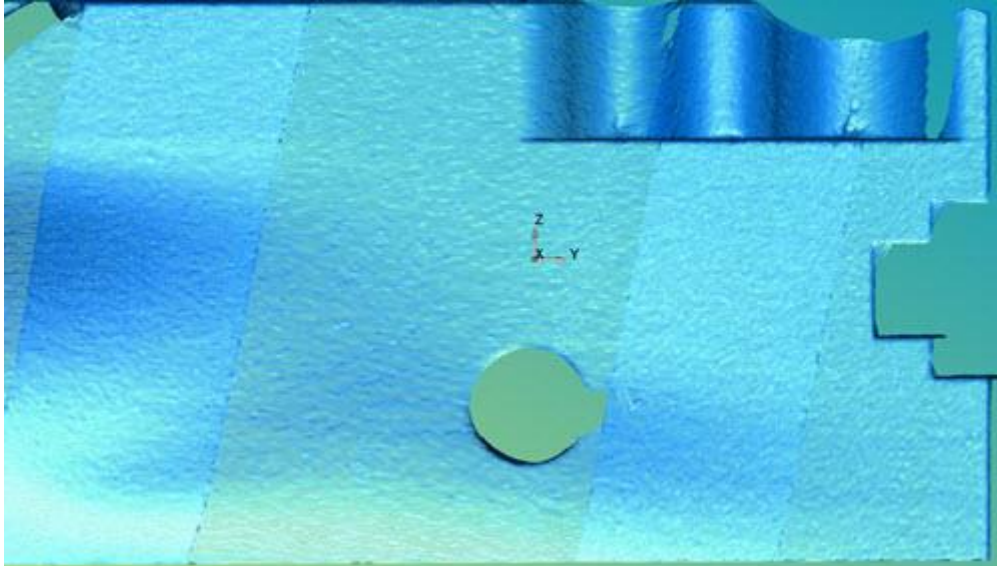
(C) - Stitched area

The overlapping scan passes must be within a distance lower than the point density in order to be stitched.



Example of Stitch Patches turned ON when scanning as Mesh display

When scanning as **Mesh** display and the **Stitch Patches** check box is NOT marked, multiple scan passes are overlayed on top of each other.



Example of Stitch Patches turned OFF when scanning as Mesh display

To use this feature:

1. From the **Pointcloud Display** section of the dialog box, click **Mesh**.
2. In the **Grid Size** box, type the value to define the mesh triangle size. A recommended starting value is 0.25mm (at 1/64 inches). A smaller grid size provides a smaller (higher quality) resolution when creating the mesh.
3. If the distance between any two points is greater than the **Max Triangle Size** value, the software does not create any triangles. If there are hole features on your part, you typically need to set this value to be slightly smaller than the smallest hole. This prevents the mesh from filling the hole.
4. Click **OK** to finish.

Using the Simulate Pointcloud Function

The **Simulate Pointcloud** function allows you to create and view the pointcloud from the **Scan** dialog box (Linear, Freeform, and so on) when the CMM is in Offline mode.

Using the laser probe orientation, field of view and scan settings, the software projects the laser lines onto the CAD model. This way you can see if the simulated pointcloud is acceptable and make changes if needed for an individual scan. PC-DMIS holds the simulated points in a COP.

Adjust the settings found on the **Animation** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**) to control the speed of the simulated laser scan. For details, see "Using the Animation Parameters for Pointcloud Simulation".

Follow the "Getting Started" chapter to define the active sensor tip and scan speed. If you like, you can pre-define the laser width and density of the scan from the **Measure Laser Probe** dialog box when you define the sensor. To access this dialog box, open the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**), and then click **Measure**. For details on laser probe measurement options, see "Measure Laser Probe Options".

Define the scan path properties from any **Scan** dialog box (linear, freeform, and other properties). You can also define the laser width and density settings from the same dialog box. For details, see "Scan Zoom States (for CMS Sensors)".

Click the **Simulate** button from any **Scan** dialog box to display the pointcloud in the Graphic Display window. You can also simulate the pointcloud when you execute the scan from the Edit window in Offline mode.

After you create the scans, you can execute the entire offline measurement routine and display all scans in different probe orientations. This allows you to check if you can extract scanned Auto Features (for example) based on the scan settings.

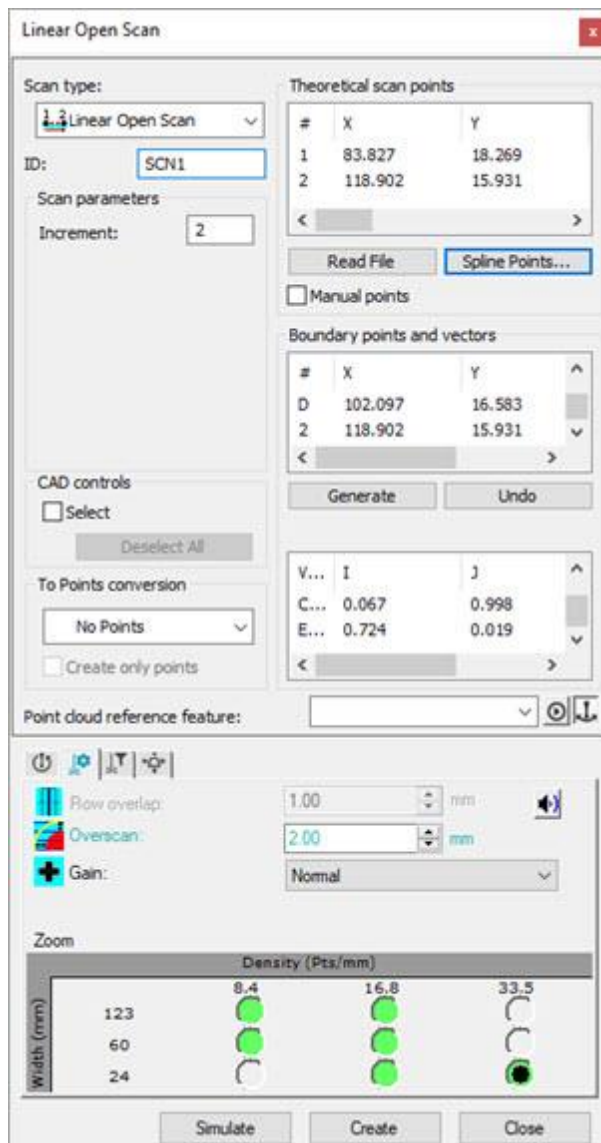


Warning: If the CMM is online and the **Simulate** button is pressed on the **Laser Scan** dialog box (Freeform, Linear Open, etc.), the software immediately drives the machine and scans online. To prevent injury, make sure you are clear of the machine prior to pressing the button.

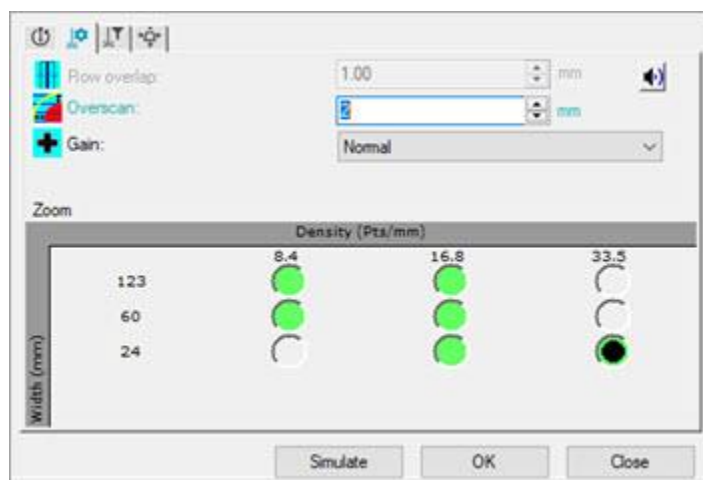
Example Using the Simulate Pointcloud Function

For example, to use the Simulate pointcloud function on a linear open scan:

1. Create a COP (**Insert | Pointcloud | Feature**). For details on pointcloud features and creating a COP, see the "Using Pointclouds" chapter.
2. Set the scan speed. For details, see "Getting Started".
3. Open the **Linear Open Scan** dialog box (**Insert | Scan | Linear Open**).

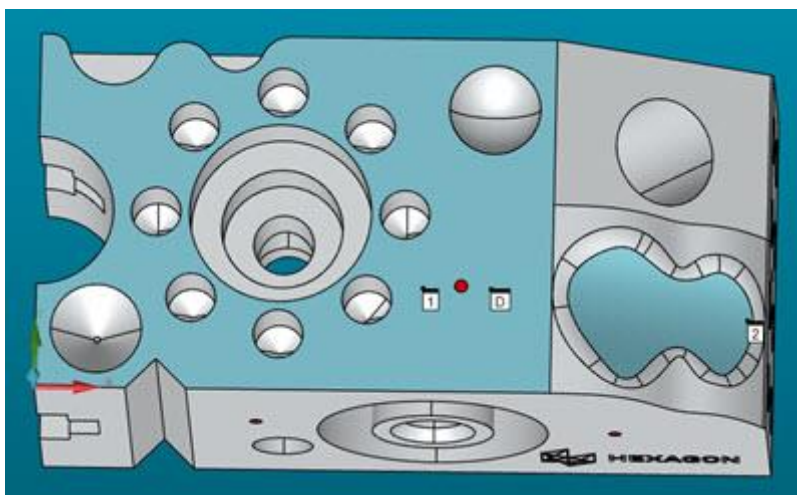


4. In the **Scan parameters** section, set the **Increment** value.
5. At the bottom of the dialog box, click the **Laser Scan Properties** tab, and set these options:
 - Enter the **Overscan** value.
 - Select the **Gain** option from the list.
 - Select the stripe **Width** and scan **Density** setting.



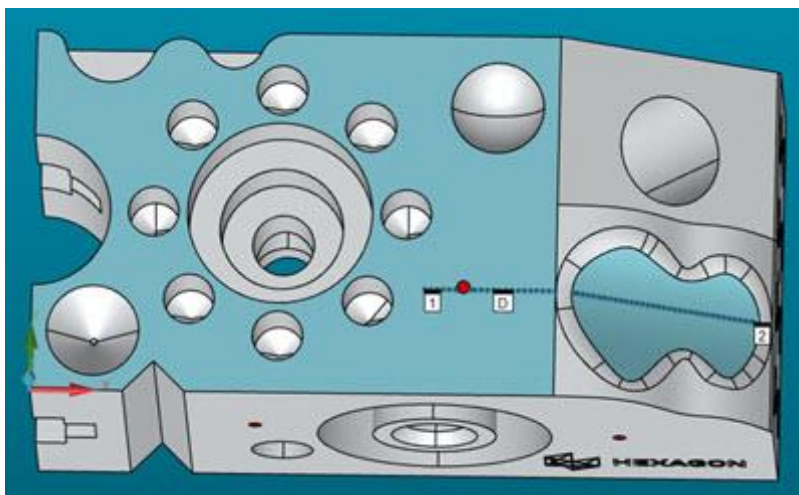
Laser Scan Properties tab

6. In the Graphic Display window, click the three points on the CAD model to define the boundary points and vectors.



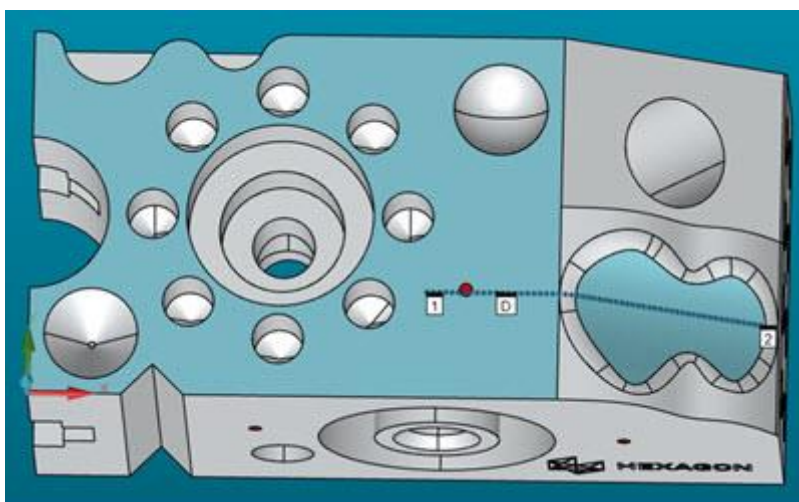
Example showing the three points to set up the scan

7. From the **Boundary points and vectors** section, click **Generate**.



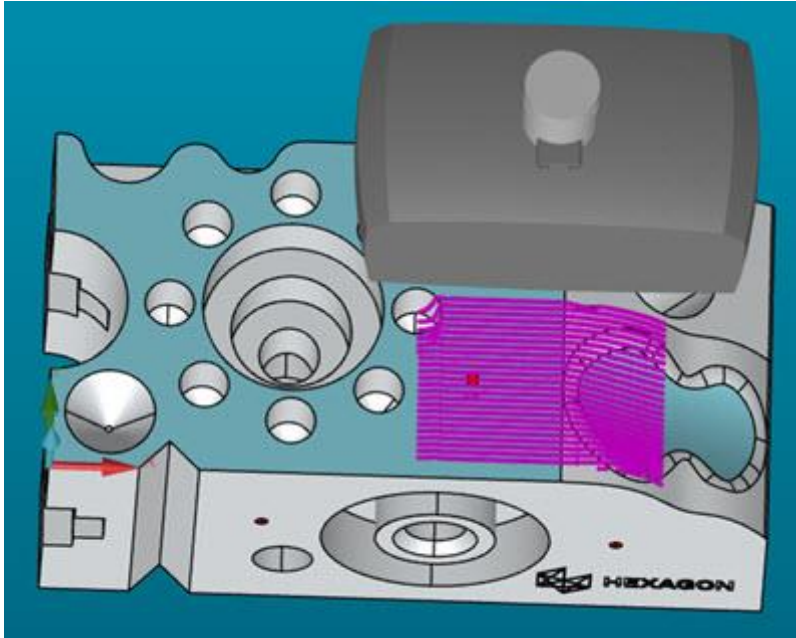
Example showing a Linear Open scan generated

8. From the **Theoretical Scan Points** section, click **Spline Points**.

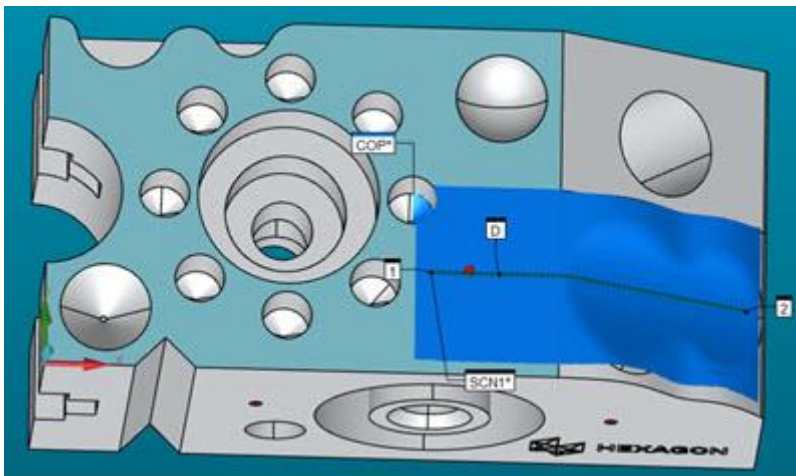


Example showing a Linear Open scan splined

9. Click the **Simulate** button to show the simulated pointcloud based on the current probe orientation (active tip) and laser scan settings.



Example showing the Pointcloud Simulation in progress



Example showing the Pointcloud Simulation completed

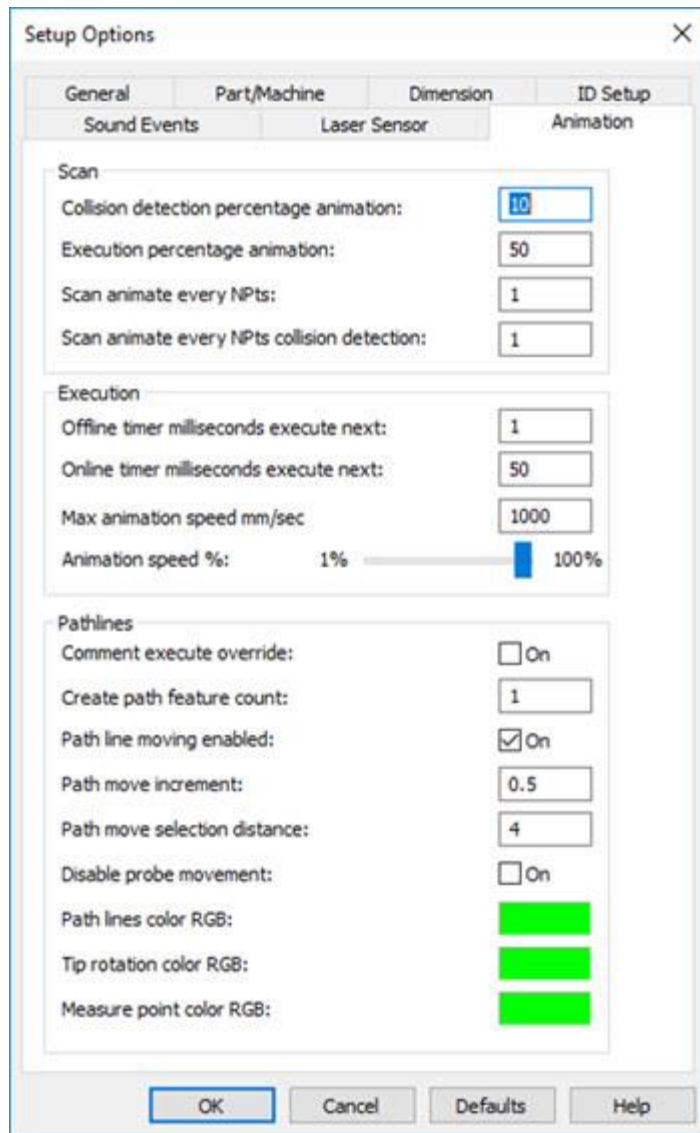
If needed, you can make changes to the scan and simulate it to check the results.

10. When everything looks correct, click the **Create** button to implement the scan in your measurement routine.

Using the Animation Parameters for Pointcloud Simulation

You can control the speed of the simulated laser scan in the **Scan** and **Execution** areas on the **Animation** tab of the **Setup Options** dialog box (**Edit** | **Preferences** | **Setup**, or

press the F5 key). For details, see "Setup Options: Animation Tab" in the PC-DMIS Core documentation.



Setup Options - Animation tab

Scan Area

Scan animate every NPts - This value determines the number of scan path points that PC-DMIS uses for the animation.

- For pointcloud simulation, if you enter a value of "1", the software uses every scan point, which results in a smoother animation.
- If you use a larger value (for example "10") for pointcloud simulation, the laser scanner probe moves from point 1 to point 10 and immediately shows all the

purple pointcloud stripes between those scan path points. The result is a faster but less-smooth animation. The default value is 50.



You can also set this value in the PC-DMIS Settings Editor. For details, see "ScanAnimateEveryNpts" in the PC-DMIS Settings Editor documentation.

Execution Area



For Pointcloud simulation, the values in this area are typically set to the maximum values.

Max animation speed mm/sec - This lets you define the maximum animation speed that the animated probe will use in the Graphic Display window during measurement routine execution. The speed is in mm per second. You may find it useful to alter this value for complex measurement routines where the animation renders too slowly. To increase the duration between redrawn views of the animation, increase this value. This causes the software to draw fewer animation steps.



You can also set this value in the PC-DMIS Settings Editor. For details, see "MaxAnimationSpeed" in the PC-DMIS Settings Editor documentation.

Animation speed % - The slider allows you to adjust the actual percentage of the **Max animation speed mm/sec** value PC-DMIS uses.



You can also set this value in the PC-DMIS Settings Editor. For details, see "AnimateSpeed" in the PC-DMIS Settings Editor documentation.

Pointcloud Operators

The Pointcloud operator commands listed below perform different operations on cloud of points (COP) commands and other pointcloud operator commands. The software defines the units for these commands by the measurement routine.



Versions earlier than PC-DMIS 2014 used a COPOPER keyword prior to the operator command. This COPOPER command is no longer available, and the commands now use a COP prefix. For example, the Filter operator is now COPFILTER.

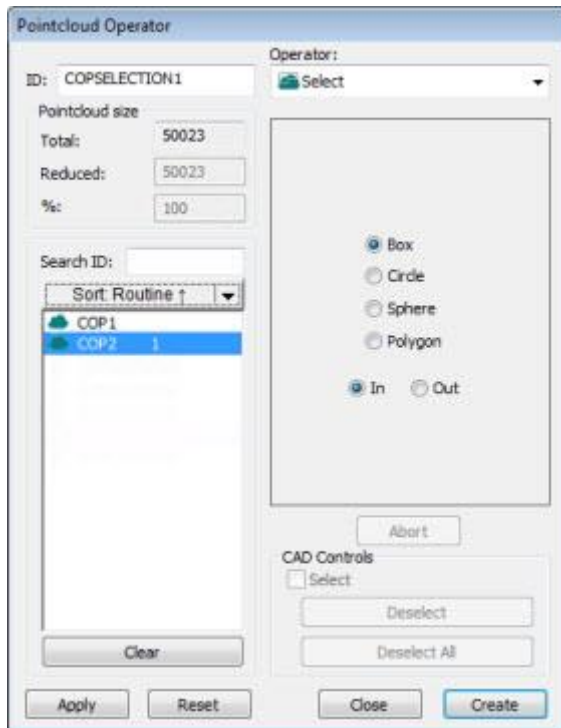
You can add pointcloud operator commands into your measurement routine in any of these ways:

- Select the **Insert | Pointcloud | Operator** menu item.
- Select menu items from the following submenus:
 - **File | Import | Pointcloud** - Import from data files to a COP.
 - **File | Export | Pointcloud** - Export to data files from a COP.
 - **Insert | Pointcloud** - Add basic pointcloud commands from this submenu. These include COP and specific point cloud operator commands (**Cross Section**, **Surface Colormap**, or **Point Colormap**) that alter the display of pointclouds in the Graphic Display window.
 - **Operation | Pointcloud** - Alter the number of points that PC-DMIS includes in COP commands. Items included in this submenu are: **Clean**, **Empty**, **Filter**, **Purge**, **Reset**, and **Select**.
- Manually type pointcloud operator commands into the Edit window. If the cursor is on the command in the Edit window, and you press **F9**, the **Pointcloud Operator** dialog box opens.
- From the **Pointcloud** toolbar, click the appropriate **Pointcloud Operator** button to open the associated **Pointcloud Operator** dialog box. The software applies the pointcloud operator to the COP.



You must be licensed with the **COP** option to use Pointcloud operator commands. You cannot use these commands if you are only licensed with the Vision option. You should disable **Vision** when you use Laser.

Manipulating Pointcloud Operators



Pointcloud Operator dialog box

The **Pointcloud Operator** dialog box is displayed by selecting **Insert | Pointcloud | Operator** from the main menu. The dialog box contains the following elements:

ID - Contains a unique identity of the pointcloud operator command being edited.

Pointcloud size - This area contains the **Total** size of the pointcloud operator selected in the list box. The **Reduced** size and the percentage (%) of reduction in size are also shown.

Command List - The list of commands on the left shows the COP or pointcloud operator commands that send data to the pointcloud operator command in the **ID** box. The Command List section also has these two functions:

Search ID - If there's a long list of operators defined, you can search using the **Search ID** box to locate specific operators in the list. When you start to

enter the operator's ID into the box, the list automatically filters based on your entry.

Sort - The **Sort** functionality is available to organize the list by **ID**, **Type**, **Routine**, or **Time**. Select the option from the list and then click the **Sort** button.

Apply - Applies the operator to the COP or pointcloud operator commands selected.

Reset - Restores all the data stored in a COP command.

CAD Controls - Lets you apply the operation to selected CAD elements. See the "CAD Controls" topic, which describes scanning in more detail.

Operator - This list shows the operator commands that you can select and apply to pointcloud or other operator commands. Depending on the type of operator selected, different options become available in the dialog box. Refer to the following operator types for details:

Thickness Colormap

The Thickness Colormap allows you to show and measure the part thickness as a colormap using only the Mesh or Pointcloud (COP) data object. You can also compare the measured thickness to the nominal CAD model thickness.

- For a Pointcloud, access the Thickness Colormap from the **Pointcloud** toolbar (**View | Toolbars | Pointcloud**) or, from the **Insert | Pointcloud | Thickness Colormap** menu option.
- For a Mesh, access the Thickness Colormap from the **Mesh** toolbar (**View | Toolbars | Mesh**), or from the **Insert | Mesh | Thickness Colormap** menu option.

To use this function, the measured data object must have data on two opposing sides which have the opposite normal orientations. When using a Pointcloud, the data must have XYZIJK values or stripe information. For details, see "Example Pointcloud File Formats for Thickness Colormaps" in this documentation.

When performing a thickness colormap of the data object (Pointcloud or Mesh), PC-DMIS calculates the measured thickness up to a **Max Thickness** value. The software does not evaluate any data value greater than the **Max Thickness**.

When the measured data is aligned to a CAD model, you can choose to create a thickness colormap which shows the deviation of the measured thickness in comparison to the nominal CAD model thickness.



If you use a large **Max Thickness** value on a large data object, it can result in a longer processing time.

You can create the following thickness colormaps:

- Measure part thickness using either a Pointcloud or Mesh data object
- Compare to CAD thickness colormap, which shows the deviation of the Pointcloud or Mesh data object thickness in comparison to a CAD model.

Measured Thickness Colormap

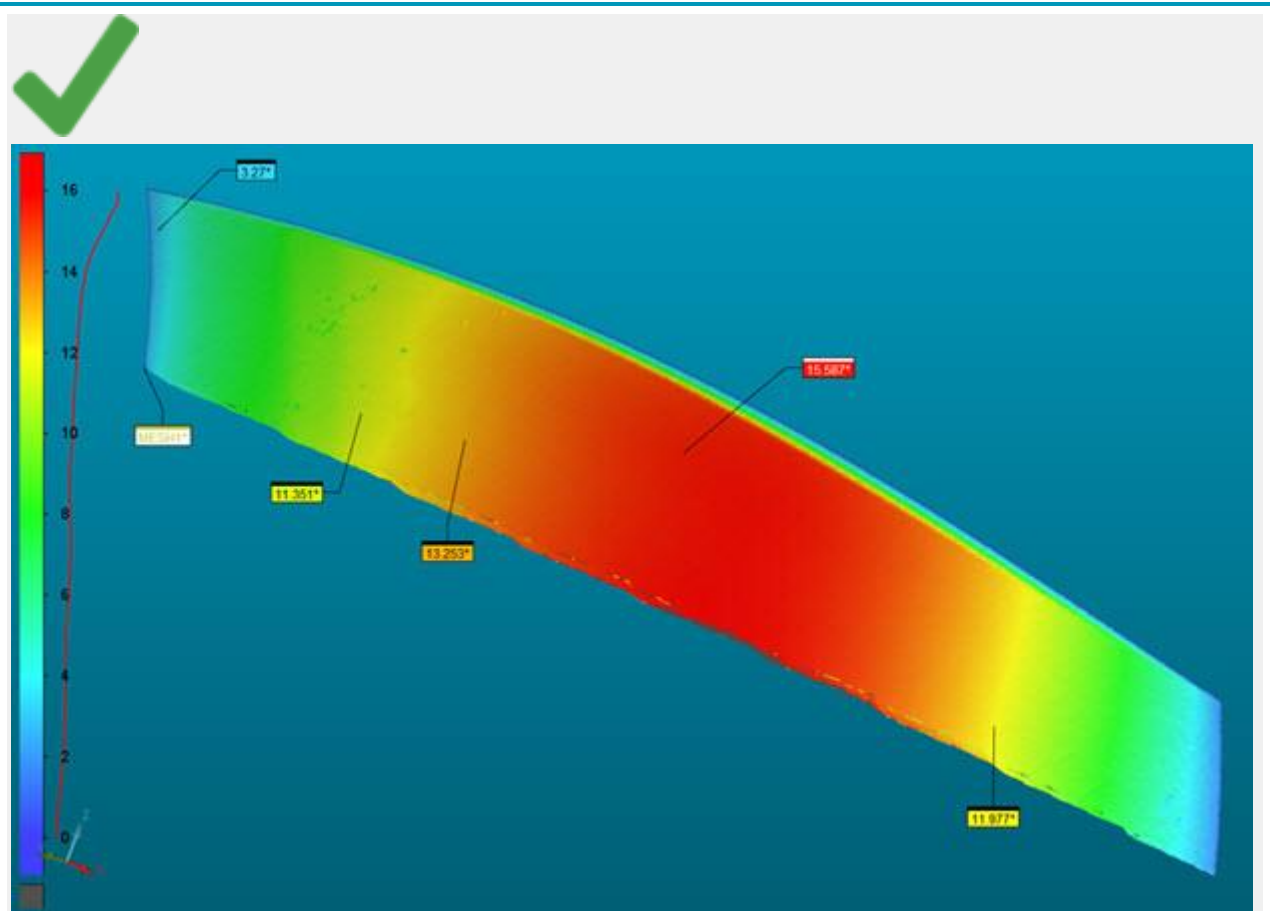
To measure a Thickness Colormap:

1. From the Thickness Colormap dialog choose the data object, Pointcloud or Mesh.
2. Select the Method: **Ray based** or **Sphere**. For details, see "Thickness Colormap Method".



When you measure the thickness of a Pointcloud data object, you cannot select a method to use. PC-DMIS automatically uses the Sphere method.

3. Enter the value for **Max Thickness**. The software does not evaluate any data value greater than the **Max Thickness** value.
4. Click **Apply**.
5. Create Annotations. For details, see "Thickness Colormap Annotations".
6. Click **Create**.



Example of a Thickness Colormap Using a Mesh Data Object

With the above example, PC-DMIS creates this command in the Edit window:

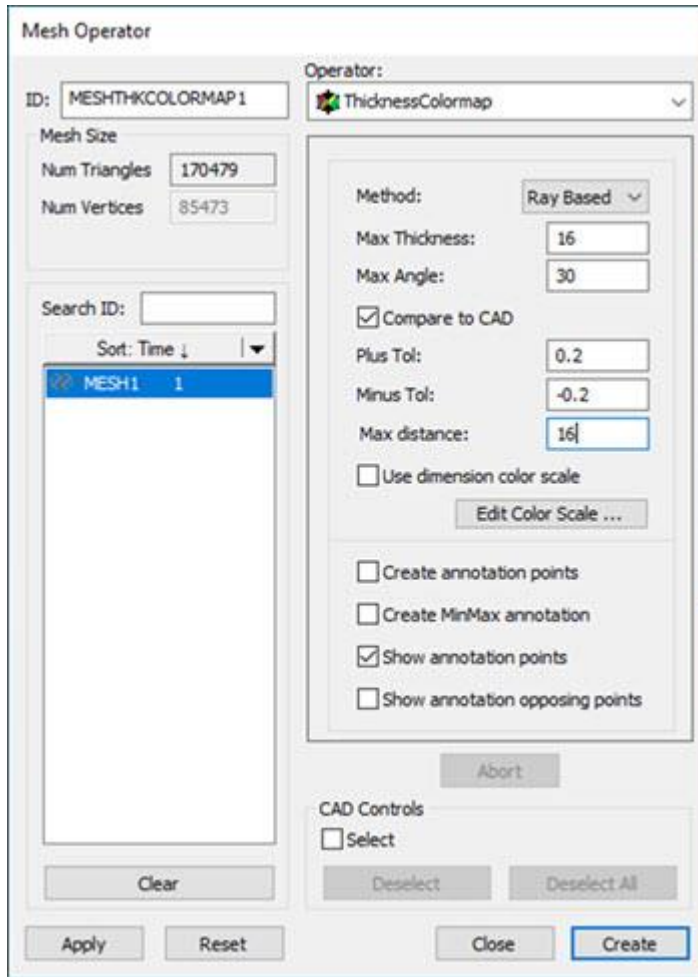
```
MESHTHKCOLORMAP1=MESH/OPER,THICKNESSCOLORMAP,,SHOW  
PARAMETERS=NO TRIANGLES=170479,VERTICES=85473,REF=MESH1,,,
```

Compare to CAD Thickness Colormap



The **Compare to CAD** thickness colormap processing can be very time-consuming when working with a large pointcloud (>1 million points) or mesh. It is recommended that you filter the pointcloud prior to performing the **Compare to CAD** operation. For details on filtering a pointcloud, see "FILTER" in this documentation.

You can create a thickness colormap of a Pointcloud or Mesh data object in comparison to a CAD model. In this case, from the **Pointcloud Operator** or **Mesh Operator** dialog box, select the **Compare to CAD** check box.



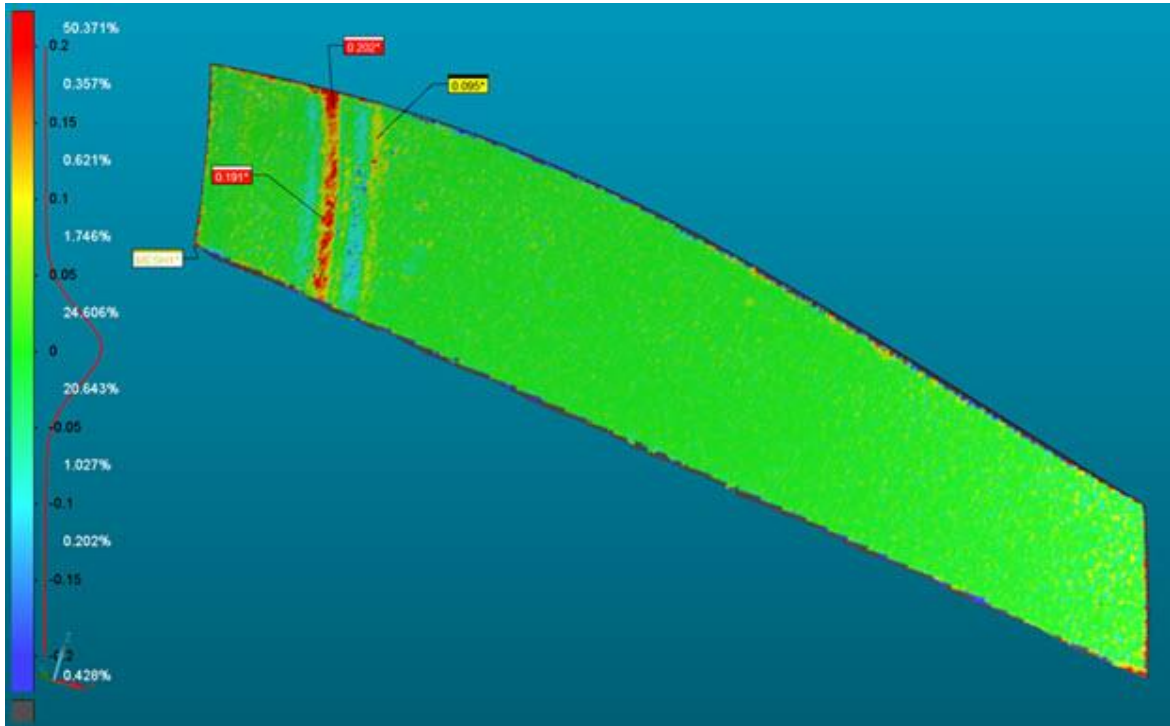
PC-DMIS computes the deviation of the data object thickness in comparison to a CAD model.

The software colormaps the Pointcloud or Mesh data object used for the comparison to CAD to show the deviations.

To do this:

1. From the **Pointcloud Operator** or **Mesh Operator** dialog box, select **ThicknessColormap** from the **Operator** list.
2. Choose the corresponding Pointcloud or Mesh data object.
3. From the **Method** list, select the method: **Ray Based** or **Sphere**. For details on these methods, see "Thickness Colormap Method".
4. Enter the **Max Thickness**. The software does not evaluate any data value greater than this value.
5. Click **Compare to CAD** and enter the appropriate tolerance values in the **Plus Tol** and **Minus Tol** boxes. You must use a minus sign when you enter a negative number.

6. Enter the **Max distance**. PC-DMIS uses the data within this distance from the CAD model for the colormap.
7. Click **Apply**.
8. Create annotations. For details on creating annotation points for the Thickness Colormap operator, see "Thickness Colormap Annotations".
9. Click **Create**.



Example of a Thickness Colormap, Mesh data object compared to CAD model

Thickness Colormap Method



When you measure the thickness of a Pointcloud data object, you cannot select a method to use. PC-DMIS automatically uses the Sphere method.

You can choose between two mathematical methods for computing the Thickness Colormap:

- **Ray Based Method**

Mesh Operator

ID: MESHTHKCOLORMAP1

Operator: THKCOLORMAP

Mesh Size

Num Triangles: 125676

Num Vertices: 63624

Search ID:

Sort: Time ↓

MESH1 1

Clear

Method: Ray Based

Max Angle: 30

☐ Compare to CAD

Max Thickness: 5

☒ Use dimension color scale

☐ No data color

No data

☐ Create annotation points

☐ Create MinMax annotation

☒ Show annotation points

Abort

CAD Controls

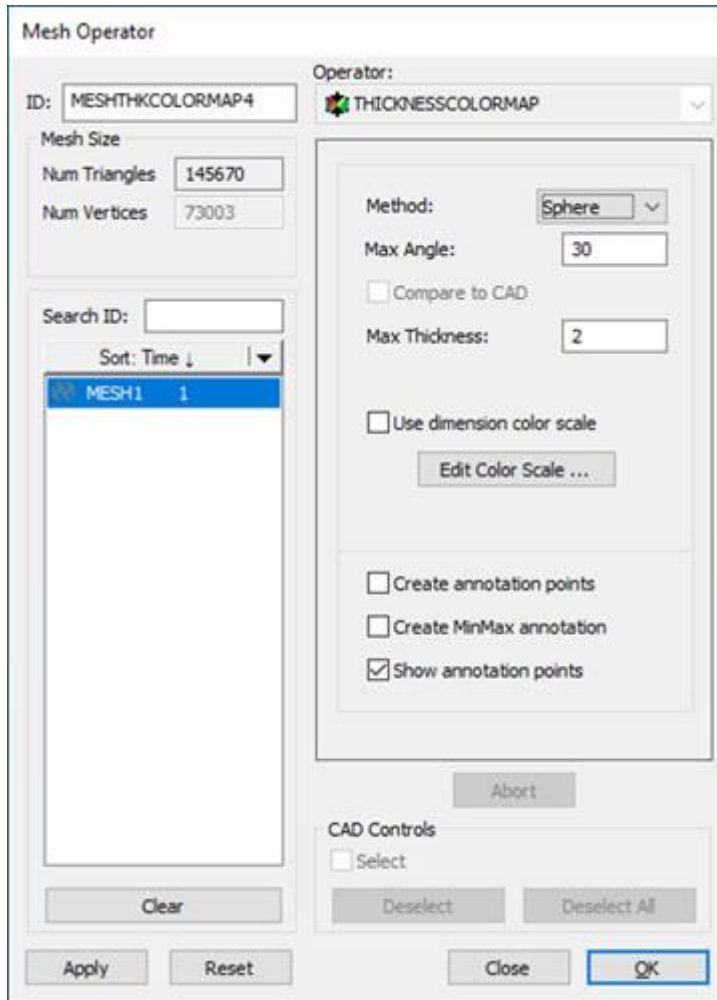
☐ Select

Deselect Deselect All

Apply Reset Close OK

When you select the **Ray Based** method for a Mesh, PC-DMIS pierces the Mesh at each vertex along its normal to the opposing side. The software then uses the data within the **Max Angle** to calculate the thickness.

- **Sphere Method**



When you select the **Sphere** method, PC-DMIS uses a maximum inscribed sphere to calculate the thickness between the two opposing sides.



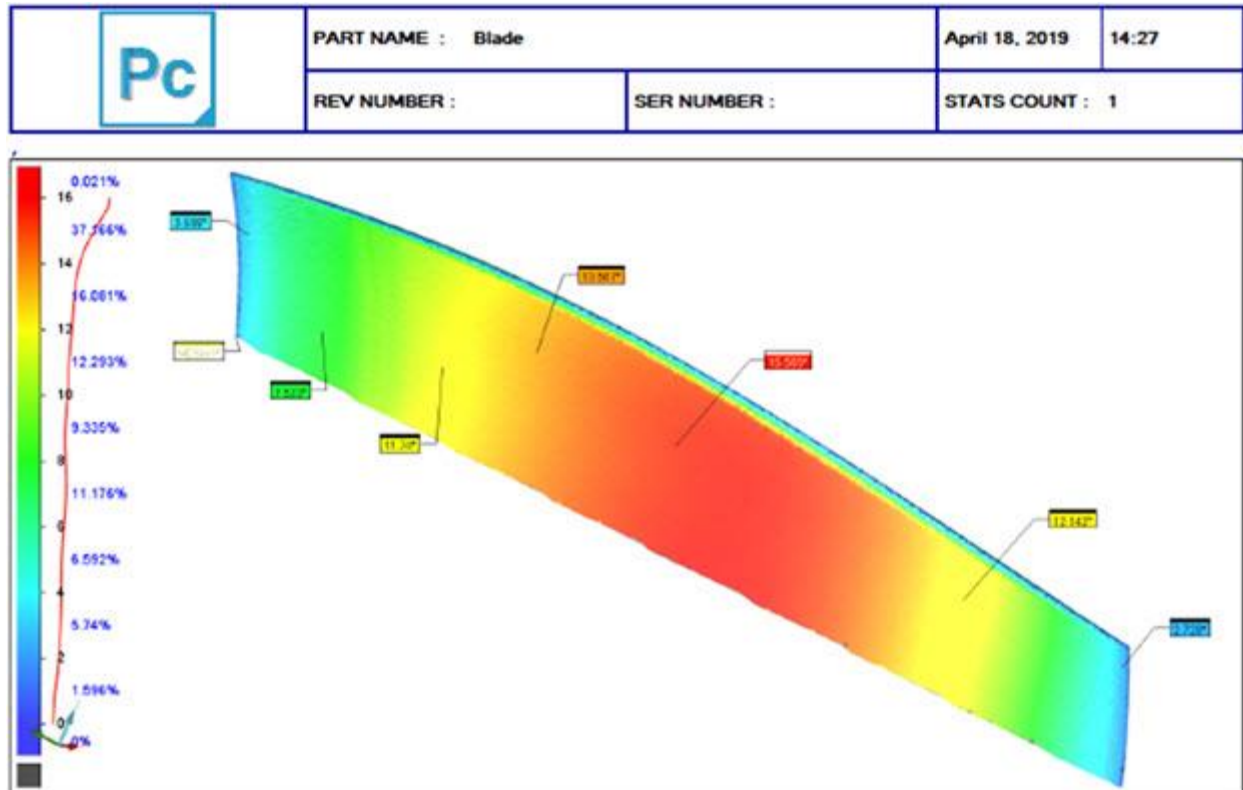
For either method, if you select the **Use the dimension color scale** option and no data exists on the opposing side, PC-DMIS uses the No data color in that area. If you do not select the **Use dimension color scale**, you can define the No data color with the **Edit Color Scale**.

For details on editing the dimension color scale, see "Using the Dimensions Colors Window (Dimensions Color Bar)" in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.

For details on using the **Edit Color Scale** option, see "Edit the Color Scale" in this documentation.

Show the Thickness Colormap in the Report

To show the Thickness Colormap in a report, select **Insert | Report command | Snapshot**. PC-DMIS inserts a snapshot of the colormap image when it creates the report.



Example of a Thickness Colormap report

Thickness Colormap Annotations

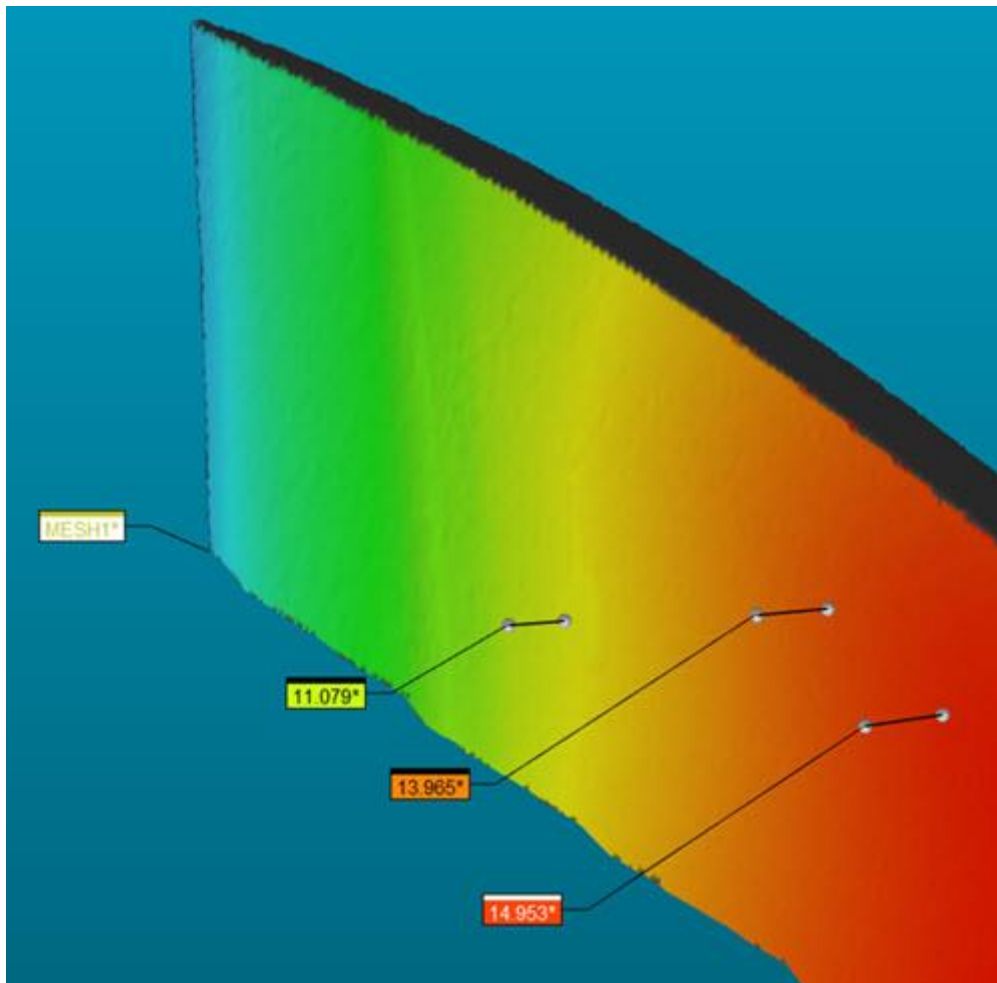
You can create annotation points, Min/Max annotations and show/hide annotations for Thickness colormaps, similar to Surface Colormaps.

- When you create annotations for a Pointcloud or Mesh data object (not compared to a CAD model), the annotation shows the measured thickness.
- When you create annotations for a Pointcloud or Mesh which has been compared to a CAD model, the annotations show the thickness deviation between the measured thickness and the CAD.

Show Annotation Opposing Points

To understand how PC-DMIS measures the annotation point, you can display the initial clicked point, the point on the opposite side of the part thickness and the connecting line. Click the **Show annotation opposing points** check box to enable this function.

You can change the size of the annotation points and the thickness of the connecting line from the **Pointcloud** area of the **OpenGL** tab on the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | OpenGL**). For details, see "Changing OpenGL Options" in the "Setting Your Preferences" chapter of the PC-DMIS Core documentation.



Thickness Colormap example with the "Show annotation opposing points" check box selected

Example Pointcloud File Formats for Thickness Colormaps

When using a pointcloud, the data must have XYZIJK values or stripe information.

Example file formats are shown below:

```
20.91911 -3.91231 6.62312 0.52816 -0.84145 -0.11401
21.09812 -3.96453 6.52849 0.48867 -0.86438 -0.11851
21.98763 -4.04430 6.50748 0.47940 -0.88303 -0.09803
22.49231 -4.05894 6.51137 0.50725 -0.85229 -0.12762
22.89023 3.93331 6.52312 0.52616 -0.85145 -0.12401
```

Example Pointcloud file in XYZIJK format

```
L0##1##1##0.724029##-0.499422##0.475746
827.932922 34.322559 186.829498
827.927063 34.331051 186.841080
827.922791 34.338451 186.853577
827.922607 34.343029 186.868881
827.924866 34.345963 186.885864
827.927795 34.348576 186.903214
827.934082 34.353867 186.937988
827.942688 34.362518 186.989517
827.953796 34.373577 187.058304
827.969788 34.389599 187.161560
827.992676 34.409428 187.300430
828.029541 34.437286 187.510300
828.089600 34.476681 187.827393
828.137268 34.509426 188.090515
828.191040 34.551125 188.403336
828.259766 34.602585 188.785507
828.335510 34.659737 189.218796
828.387390 34.701157 189.529175
828.455322 34.758785 189.940521
828.519897 34.820339 190.347870
828.587646 34.881676 190.772919
828.625549 34.920185 191.025818
828.665955 34.975124 191.340225
```

Example Pointcloud file with stripe information



Edit the Color Scale

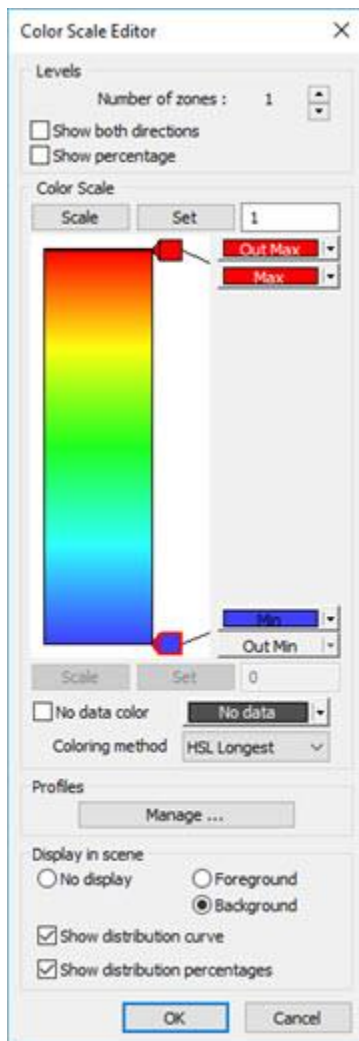
Edit Color Scale ...

The **Edit Color Scale** button is available on the **Pointcloud Operator** dialog box for the Point Colormap and Surface Colormap operators. It allows you to change the color

scale for these operators. By default, the Min/Max values of the scale are set to the +/- tolerance values of the colormap. You can save different color bars and then recall them with this function.

To begin:

1. From the **Pointcloud** toolbar, select **Pointcloud Point Colormap** () or **Pointcloud Surface Colormap** () to open the **Pointcloud Operator** dialog box for the operator.
2. Click the **Use dimension color scale** check box to clear it and display the **Edit Color Scale** button.
3. Click the **Edit Color Scale** button to display the **Color Scale Editor** dialog box:



Color Scale Editor dialog box

The following areas of the dialog box are described.

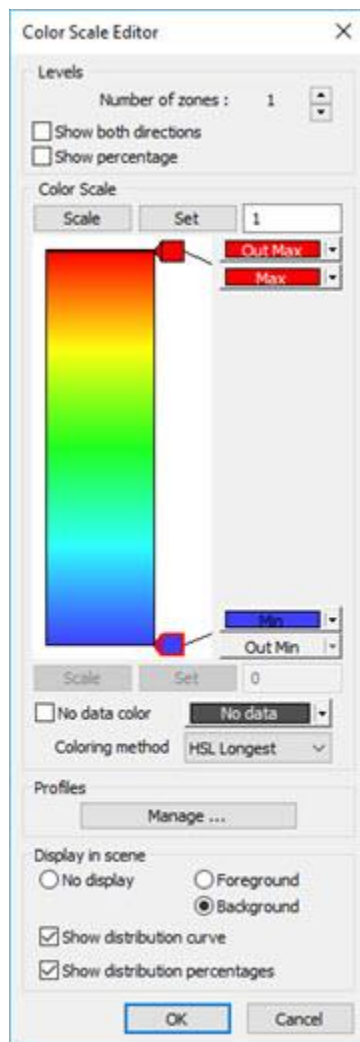
- **Levels** area
- **Color Scale** area
- **Profiles** area
- **Display in scene** area

Color Bar Levels Area



Levels area of the Color Scale Editor dialog box

Number of zones - This setting allows you to change the number of color zones the software displays in the color bar. A setting of one (1) displays the gradient view shown below:



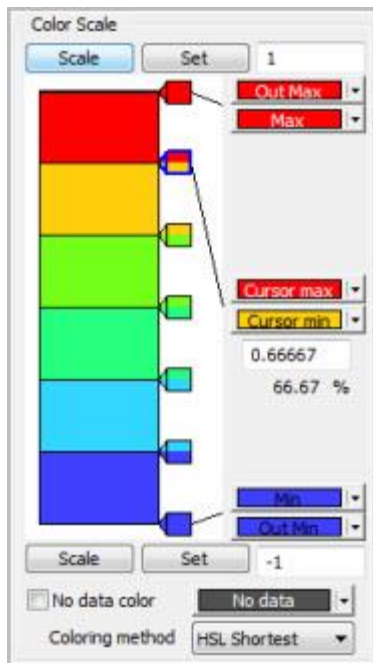
Color Scale Editor dialog box

Click the up and down **Levels** arrows to change the number of tolerance zones. You can also click any of the current zones to create a new zone at that location.

Show both directions check box - Select this check box to enable the **Min** value's **Scale** and **Set** controls. If you do not select this check box, the **Min** value's **Scale** and **Set** controls are disabled. The **Min** value in this case is the negative of the **Max** value.

Show percentage check box - If you select this check box, the software displays the Color Scale with percentages.

Color Scale Area



Color Scale area of the Color Scale Editor dialog box

Color Scale section - This section determines the tolerance zones and the colors associated with the measured values in relation to the respective tolerances. The **Scale** and **Set** buttons change the Max and Min tolerance values with the following differences:

Scale button - If you click this button, the software appropriately scales the intermediate zone values designated by the tolerance markers around the new Max and Min values.

1. Enter a new Max or Min value and then click **Set**. If you change the Min and Max values on the color bar, this also changes the Plus and Minus tolerance values on the colormap.
2. Click the respective **Scale** button. All the zones in the color bar appear the same, except the software appropriately scales the values of each marker around the new Max and Min values.

Set button - Click this button to change the upper value of the highest zone, or the lower value of the lowest zone. The intermediate zone values designated by the tolerance markers remain the same.

1. Enter a new Max or Min value.
2. Click the respective **Set** button. The corresponding Max or Min zone changes accordingly. All intermediate zone values remain the same.



To change Zone values, click and drag one of the zone markers. You can also enter the Zone values. To enter new zone values:

1. Click the zone marker to display a leader line from the marker to the selected zone and a field appears.
2. Enter an appropriate value in the field then click outside the field for the value to take effect.

No data color check box - Select this check box to set the color where no data exists based on the colormap Max Distance. To define the color for this option:

1. Click the drop-down arrow to the right of the check box to display the standard color picker dialog box.
2. Select the color for this option and click **OK**.
3. Click the check box to select it and apply this option to your surface colormap.

Coloring method - The drop-down list provides pre-defined color bar color schemes you can select. Click the drop-down arrow to display the list and select the color scheme you want to apply.

Color Bar Profiles Area

Use the **Profiles** area of the **Color Scale Editor** dialog to manage color bar schemes.

Click the **Manage** button to open the **Profile Manager** dialog box.



Profile Manager dialog box

The dialog box has these options:

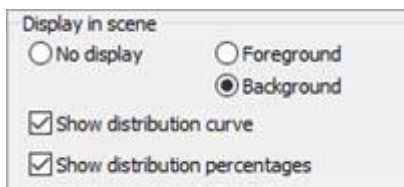
- If this is a new color scheme, enter a unique name for the color scheme in the **Name** field and click **Save**. The software saves the current color bar profile under the name entered.

- To load a color scheme profile, select one from the **Name** list and click **Load**. You can also begin typing a profile name in the **Name** field to filter the list based on your entry.
- To delete an existing profile, select one from the **Name** list and click **Delete**. You can also begin typing a name in the **Name** field to filter the list based on your entry. The software permanently deletes the profile - this cannot be undone so use caution when you delete a color scheme.



PC-DMIS saves the files with a .cbr extension in the same folder as measurement routines.

Color Bar Display in Scene Area




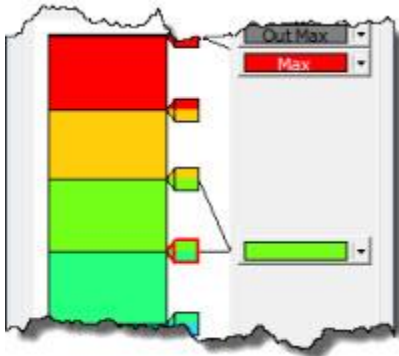
Display in scene area of the Color Scale Editor dialog box

The **Display in scene** area of the **Color Scale Editor** dialog box defines how the color scheme appears in the Graphic Display window. The options are:

- **No Display** - The Color Bar does not appear in the Graphic Display window.
- **Foreground** - The Color Bar appears on top of the CAD objects in the Graphic Display window.
- **Background** - The Color Bar appears behind the CAD objects in the Graphic Display window.
- **Show distribution curve** check box - When you select this check box (default), the software displays the distribution curve histogram layered on top of the color scale data values. The curve provides a visual indicator of the colormap deviations within the tolerance zones.
- **Show distribution percentages** check box - When you select this check box (default), the software displays the percentage values along with the color scale data values. This shows the percentage of deviation within the tolerance zones.

Changing a Zone's Color

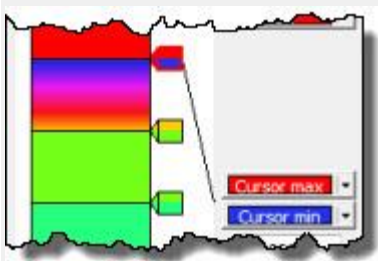
1. Click the Max tolerance marker  for the particular zone, then press the Ctrl key on your keyboard and click the Min tolerance marker for the same zone.



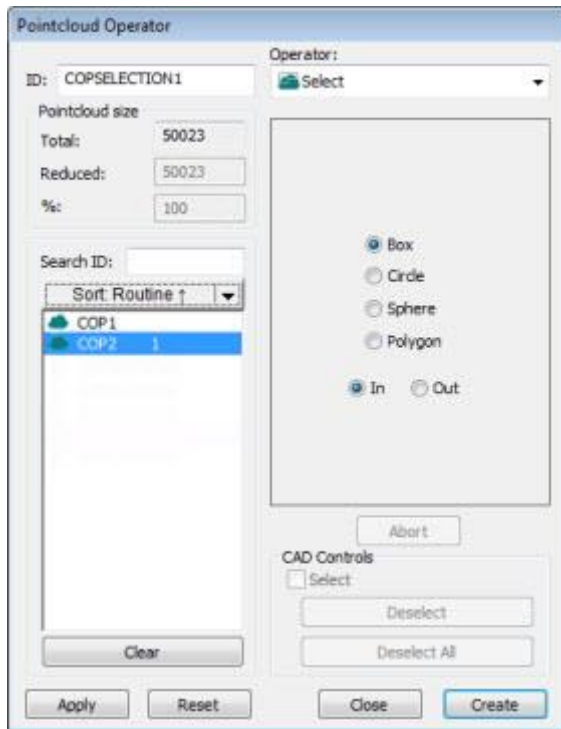
2. Once selected, click the drop-down arrow to display the standard color-picker dialog.
3. Select the new color and click **OK**. The software changes the color for the selected zone to the new color.



If you only change the Max or Min value of a zone, PC-DMIS only changes that zone's color to a gradient scheme. For example, if you change just the Max color of a zone, the zone's gradient color scheme is based on the new Max color selected and the current color of the Min value as shown below.




SELECT



Pointcloud Operator dialog box - SELECT Operator

The SELECT operation selects a subset of data contained in a COP command.

To apply the SELECT operation to a pointcloud, click **Select Pointcloud** () from the **Pointcloud** toolbar, or select **Operation | Pointcloud | Select**. By default, the **Polygon** option is used when you click the **Select Pointcloud** button.

To select a region of points:

1. Select the desired option button inside the dialog box:

Box

Circle

Sphere

Polygon



Press the **End** key to close the polygon selection.

2. Select the **Pointcloud** command that you want to apply the selection to form the list of commands.
3. Make the selections that define your selection type by clicking and dragging in the CAD in the Graphic Display window. The axis of the selection entities should be perpendicular to the current view. See the table below as a guide for what you should select.
4. If you want to keep the points inside the selection domain, select **In**. If you want to keep the points outside the selection domain, select **Out**.
5. After clicking the necessary points in the Graphic Display window, click the **Apply** button to define the selection type. PC-DMIS displays the points inside or outside the selected domain in the Graphic Display window. If you use the **Sphere** selection type, the closest pointcloud point is used for the center of the sphere.
6. When you're finished, click **Create**. PC-DMIS inserts a `COP/OPER, SELECT` command.



If you want to select the complement data instead, you can use the **BOOLEAN** operator to do that. For information on the **Complement** option inside **BOOLEAN**, see the "BOOLEAN" topic.

Type	Points Needed
Box	Select two corners.
Circle	Select the center and a point specifying the radius of the circle.
Sphere	Click one point. PC-DMIS projects it onto the cloud of points to find the closest point. This represents the center of the selected sphere. Click another point. PC-DMIS uses this second point to determine the radius of the sphere.
Polygon	Select the vertices of the polygon. Press the End key to close the polygon.

Click **Create** to insert a `COP/OPER, SELECT` command into the Edit window.



For example:

```
COPSELECT4=COP/OPER, SELECT, BOX, SIZE=27377
REF, COP1, ,
```

CROSS SECTION

The screenshot shows the 'Pointcloud Operator' dialog box with the 'CROSS SECTION' operator selected. The 'ID' field is set to 'COPSECTIONS'. The 'Pointcloud size' section shows 'Total: 0', 'Reduced: 0', and '%: 100'. The 'Search ID' field is empty, and the 'Sort: Program' dropdown is set to 'Program'. The 'COP1 1' entry is selected in the list. The 'Start point' section shows X: 14.295, Y: 31.38, and Z: -0.013. The 'Direction' section shows I: 0.997, J: -0.079, and K: 0. The 'Width' is 109.973, 'Height' is 81.935, 'Delta' is 5.08, 'Step' is 20, 'Length' is 0, 'Smoothing Factor' is 0, 'Gap Fill Distance' is 2, 'Point Spacing' is 1, and 'Max Distance to CAD' is 2. The 'Profile Dimension', 'Analysis View', and 'Annotation Min/Max' sections each have a '+' button. The 'CAD Controls' section has a 'Select' checkbox and 'Deselect' and 'Deselect All' buttons. The 'Apply', 'Reset', 'Close', and 'Create' buttons are at the bottom.


Pointcloud Operator dialog box - CROSS SECTION Operator

The CROSS SECTION operation generates a subset of polylines determined by the defined intersection of a set of parallel planes with the COP or Mesh object. The set of planes is defined by the start point, direction vector, step distance between the planes, and length. The number of planes is determined by the **Step** distance divided into the **Length** plus one.



The CROSS SECTION operator can be evaluated by the profile dimension.

To apply the CROSS SECTION operation to a pointcloud, click **Cross Section**

Pointcloud () on the **Pointcloud** toolbar, or select **Insert | Pointcloud | Cross Section**.

From the **Pointcloud** or **QuickCloud** toolbar, click the **2D Section Slide Show** button



to display cross sections in the 2D view. For details, see the "Cross Section Slide Show" section of the "Show and Hide Cross Section Polylines" topic.

The list underneath the **Operator** list contains these options: **Vector**, **Axis**, **Curve**, and **2 Points**. For details on how the **Curve** function works, see the "Creating a Cross Section along a Curve" topic. For details on the **2 Points** option, see the "Creating a Cross Section between 2 Points" topic.

The CROSS SECTION operator uses the following options:

- **Start point:** This value indicates the coordinates of a point that belongs to the first plane that cuts the pointcloud. The software displays the start point as a blue ball in the Graphic Display window. You can use the ball as a handle to drag to a new location. The start point is defined by the first click in the Graphic Display window. In the actual Edit window command, the start point value is held in the START PT parameter.
- **Direction** (applies only to the **Vector** and **2 Points** option): This value indicates the direction of the normal vector. It can be defined by the first click in the Graphic Display window. In the actual Edit window command, the **Direction** value is held in the NORMAL parameter.
- **Axis** (applies only to the **Axis** option): Use this option to create a cross section along the X, Y, or Z axis. Select the desired axis (the default is X), set a start point in the Graphic Display window, and set an end point. The section plane will cut the part at a given step value over the length of the cross section.
- **Width:** This value indicates the width of the section under consideration. If the value is 0, the system calculates the value as the CAD and COP bounding box value.
- **Height:** This value indicates the height of the section under consideration. If the value is 0, the system calculates the value as the CAD and COP bounding box value.

- **Delta:** This value indicates the maximum distance from the plane for a point to be considered part of the cross section. In the actual Edit window command, the **Delta** value is held in the TOLERANCE parameter. The **Delta** property is only available when a COP object is selected.
- **Step:** This value indicates the distance between the planes. In the actual Edit window command, the step value is held in the INCREMENT parameter.



If the value in the **Step** box is greater than the value in the **Length** box, then only one section cut is created at the start point.

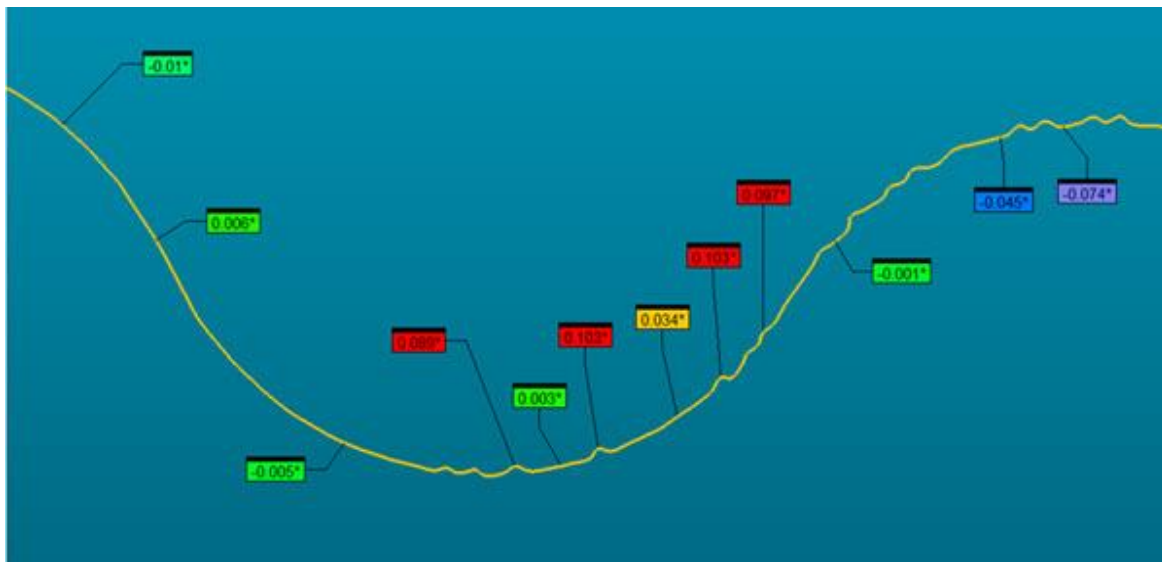
- **Length:** This value indicates the maximum distance between the first and last plane. The length value appears in the **Length** parameter of the dialog box and appears as a purple line in the Graphic Display window.
- **Smoothing Factor:** When this value is set to 0 (zero), a very small internal smoothing factor is applied.

When you use the **Smoothing Factor**, PC-DMIS applies a Least Square fit with a smoothing constraint to an ordered set of points. There is a trade-off when using the **Smoothing Factor**, since, as the roughness of the spline is smoothed, the points move from their original position. Consequently, you should use **caution** when you apply the smoothing factor since it moves or changes the data. This option may be useful for smoothing points that you consider to be "noise".

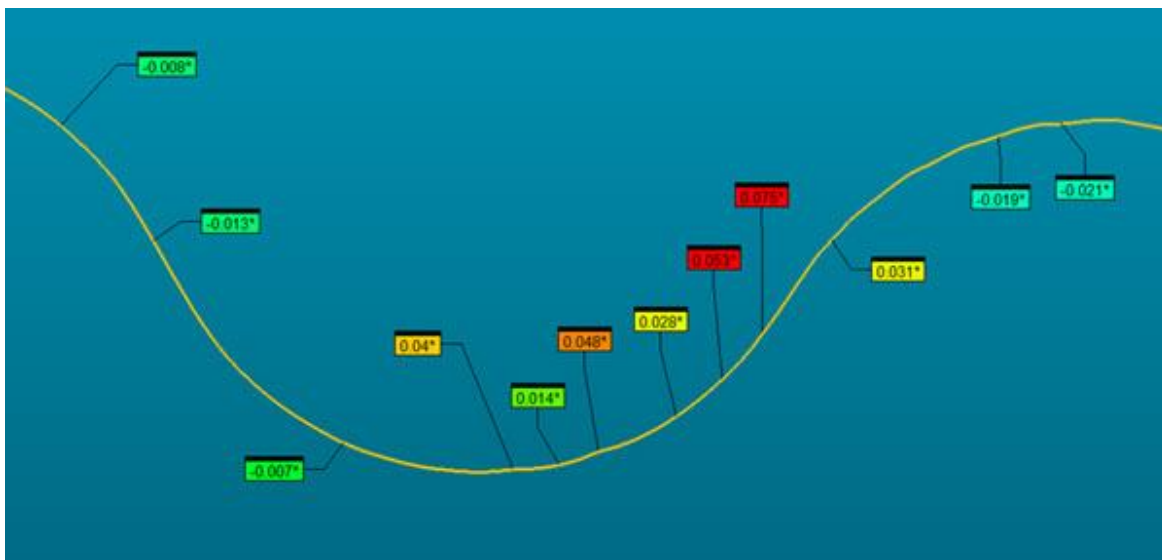
There is no upper limit to the **Smoothing Factor** value. However, the more you increase this value, the less affect it has on your data until little if any noticeable change is seen. When you use a very large smoothing factor, changes will occur to the original shape of the measured cross section polyline.



You should test the **Smoothing Factor** option with a starting value between 0.1 and 0.25. Then, depending on the result, increase the value to 0.5, 1 or 2 and re-test until you achieve the desired result.



Example with the Smoothing Factor set to 0 (zero)



Example with the Smoothing Factor set to 0.5


- **Gap Fill Distance:** This value defines the maximum gap distance along the yellow measured polylines of a cross section. If gaps equal to or smaller than this value appear, the gaps are filled in with calculated points. You can also set this value in the PC-DMIS Settings Editor. For details, see the

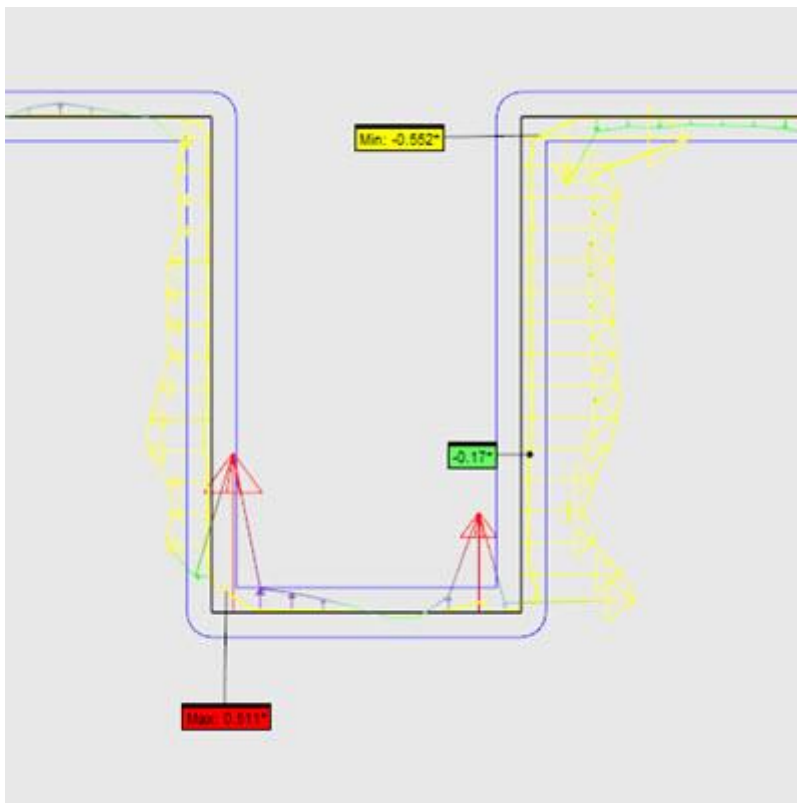
"CrossSectionMaximumEmptyLength" topic in the PC-DMIS Settings Editor documentation.

- **Point Spacing:** This value is used only when the `CrossSectionCopCadCrossSectionDrivenByCad` registry entry is set to 1 (True). This value is the step used along the CAD polylines to look for the best interpolated COP point. For greater accuracy, or if the CAD model is very small, you can set this value to a smaller value.



Point Spacing is also defined by the `CrossSectionCopCadCrossSectionStep` registry entry. For details on this registry entry, see "CrossSectionCopCadCrossSectionStep" in the PC-DMIS Settings Editor documentation.

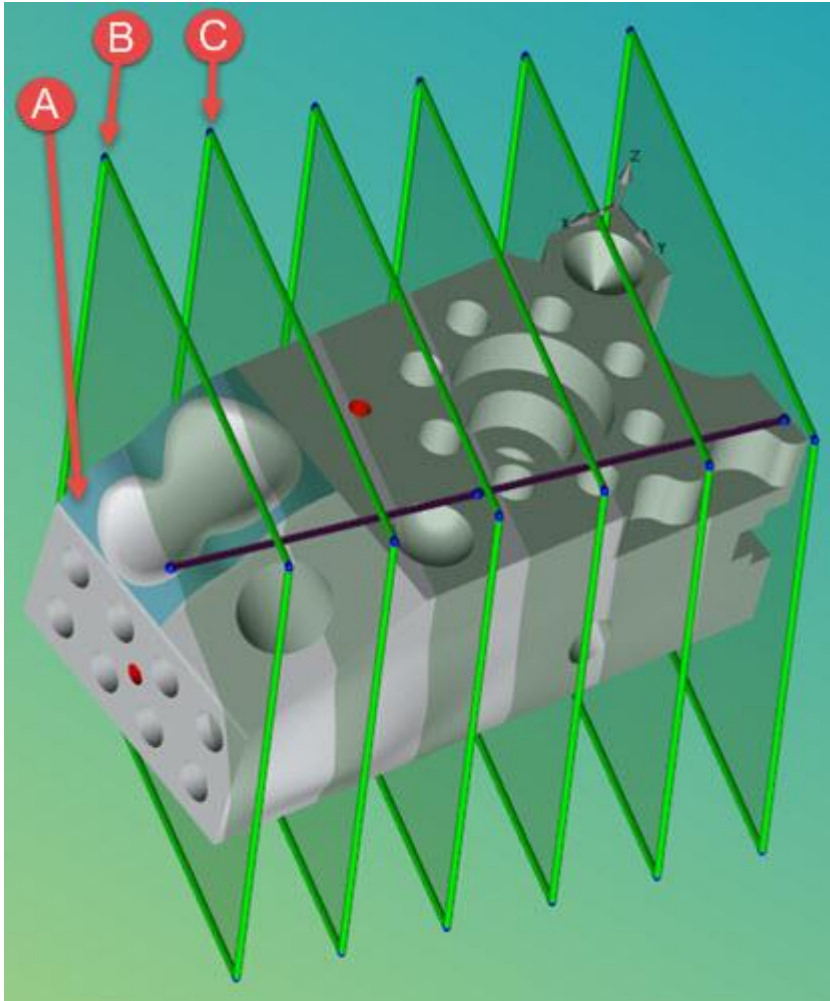
- **Max Distance to CAD:** This value defines the maximum distance of the pointcloud data relative to the nominal CAD model. The default value is 2 mm. If the scanned part deviates more than the Max Distance value from the CAD model, the software may not compute the yellow measured cross section. You can adjust this value to account for large deviations of the scanned data relative to the CAD model.
- **Profile Dimension:** Click the **Add** button  to create a new profile dimension for each cross section. For details on Profile Dimension, see the "Dimensioning Profile - Line or Surface" topic in the "Using Legacy Dimensions" chapter in the PC-DMIS Core documentation.
- **Analysis View:** Click the **Add** button to create the `ANALYSISVIEW` command in the Edit window. For details on the `ANALYSISVIEW` command, see "Create Analysis View Command" in the "Inserting Report Commands" chapter in the PC-DMIS Core documentation.
- **Annotation Min/Max:** Click the **Add** button to create the minimum and maximum values in the form of annotation labels for the active cross section.



The minimum and maximum points are recalculated each time the measurement routine is executed.

- **CAD Controls:** Mark the **Select** check box to select surfaces in the Graphic Display window. PC-DMIS filters out any cross sections that do not pass through the selected surfaces when you click **Create**.

For example, if you selected surface A after the start and end points are defined, only the cross sections at B and C would be generated:



Example of a selected surface (A) limiting the cross sections to only (B) and (C)

Selected surfaces do not affect what you see when you click the **View** button.

When the cutting planes are visible in the Graphic Display window, you can manipulate them as follows:

- Select a plane's edge handle, and drag to resize the height and width of the cutting planes.
- Select a plane's corner handle, and drag to rotate the set of planes around their axis.
- Select the first or last purple length line's blue point handle, and drag to redefine the purple line's **START** or **END** definition. While the direction is changing, the values in the dialog box and the number of planes in the Graphic Display window are updated. In Axis mode, the direction of the planes does not change.
- Select and drag the purple length line's middle blue point handle to move the set of planes.



When a cross section is created or edited, the cutting planes appear in a transparent view as shown above.

Click **Create** to:

- Insert a `COP/OPER, CROSS SECTION` command for each plane into the Edit window.



For example:

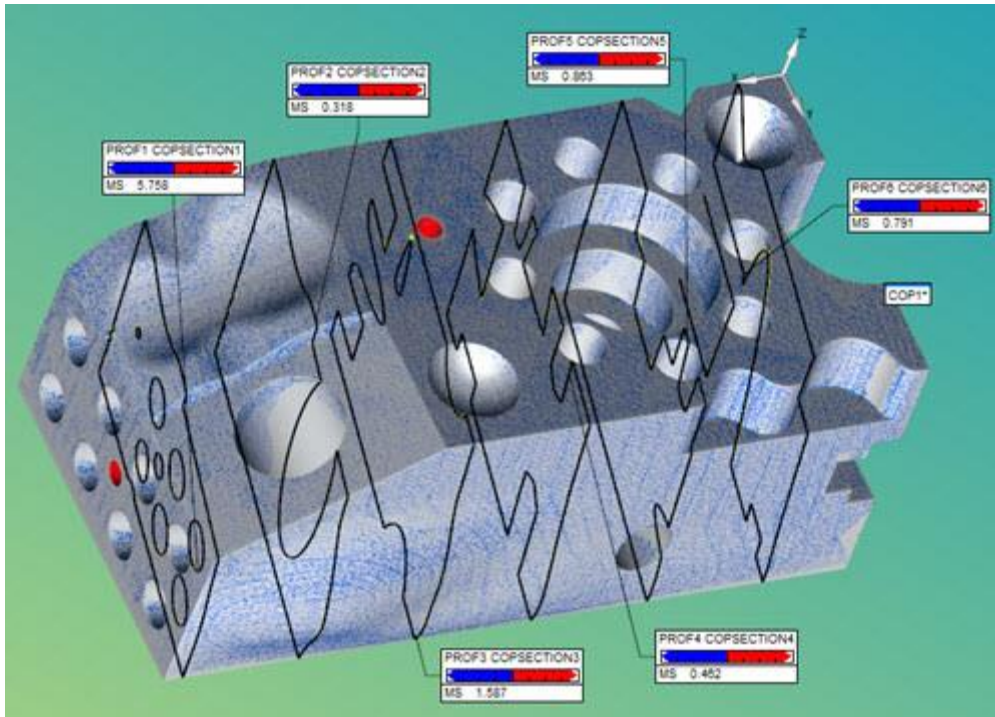
```
COPSECTION3=COP/OPER, Cross
Section, TOLERANCE=0.05, WIDTH=117.715, HEIGHT=227.086,
```

```
START PT = -6.439, 60.097, 6.276, NORMAL = 0.9684394, -
0.2221293, -0.1130655, SIZE=76
```

```
REF, COP1, ,
```

The black polylines represent the nominal CAD, the yellow polylines represent the COP polyline.

- Insert a label for each plane into the Graphic Display window as shown below:



Finished cross sections showing six planes

Defining the Cross Section by Typing Values

Use the **Pointcloud Operator** dialog box to type the values:

- **START PT:** Specify the cross section's starting point in the **Start Point X**, **Y**, and **Z** boxes.
- **NORMAL:** Specify the cross section's vector in the **Direction I**, **J**, and **K** boxes.
- **WIDTH:** Specify the cross section's width property in the **Width** box.
- **HEIGHT:** Specify the cross section's height property in the **Height** box.
- **TOLERANCE:** Specify the value that determines the maximum distance from the plane for a point to be considered part of the cross section in the **Delta** box.
- **INCREMENT:** Specify the value between cutting planes in the **Step** box.
- **LENGTH:** Specify the value between the first and last cutting planes in the **Length** box.
- **SMOOTHING TOLERANCE:** Specify the tolerance value to refine the points associated with the generated cross section in the **Smoothing Tol** box.

Defining the Cross Section by Using the Graphic Display Window

To define some of the cross-section parameters, click the CAD model in the Graphic Display window to select the **Start Point**. A pink line appears. Click a second point on the CAD model to determine the **Direction** vector and the **Length**.

Creating a Profile Dimension from the Graphic Display Window


When you double-click a cross section label, a new profile dimension is created that evaluates the selected cross section.

Measuring a Radius on a Cross Section with the 2D Radius Gage

PC-DMIS provides the 2D Radius Gage to quickly measure the radius on a pointcloud cross section. For details, see "2D Radius Gage Overview".

2D View of Cross Sections

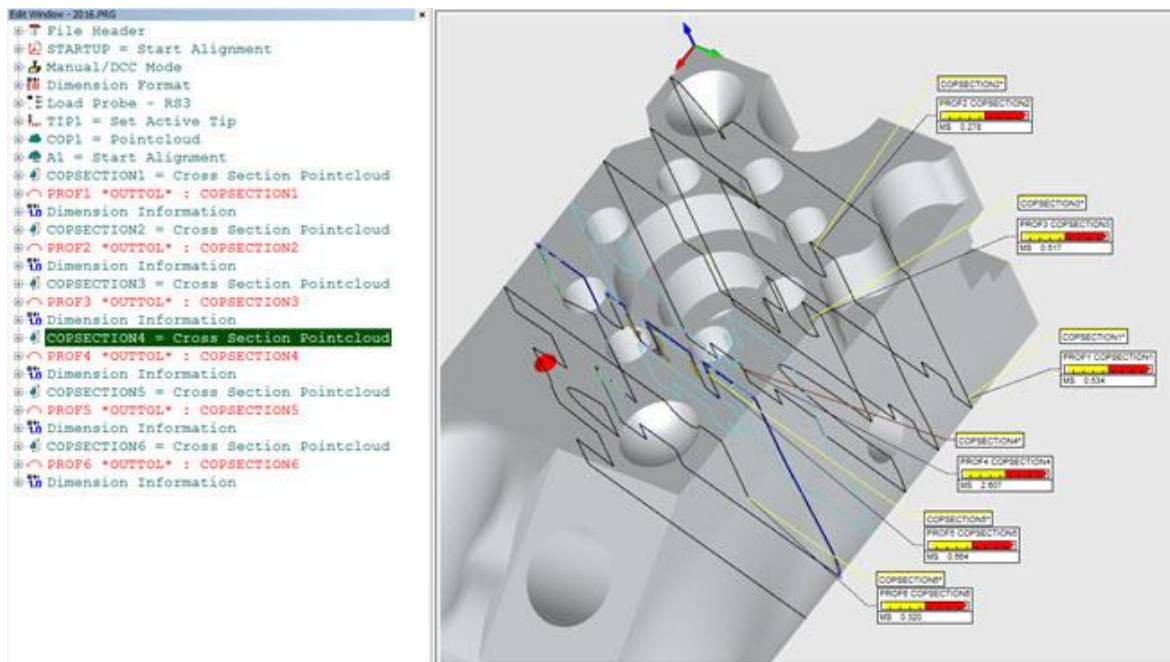
Once you define a cross section, each section can be individually displayed in a 2D view. The view is normal to the cross section. Any annotation points created on the cross section appear in the 2D view.

To view the cross sections in 2D, from the **Graphic Modes, Mesh, Pointcloud**, or **QuickCloud** toolbar (**View | Toolbars**), click the **2D Section Slide Show** button ().

For more information on the 2D Section Slide Show, see the "Cross Section Slide Show" section of the "Show and Hide Cross Section Polylines" topic.

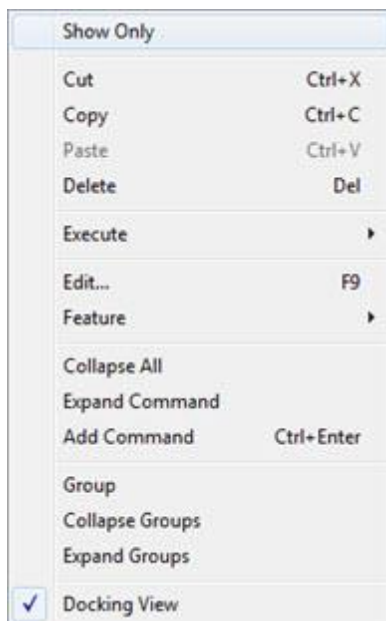
How to view the cross sections in 2D from the Edit window.

- a. From the Edit window, click the cross section you would like to view in 2D. The selected section appears in light blue in the Graphic Display window.



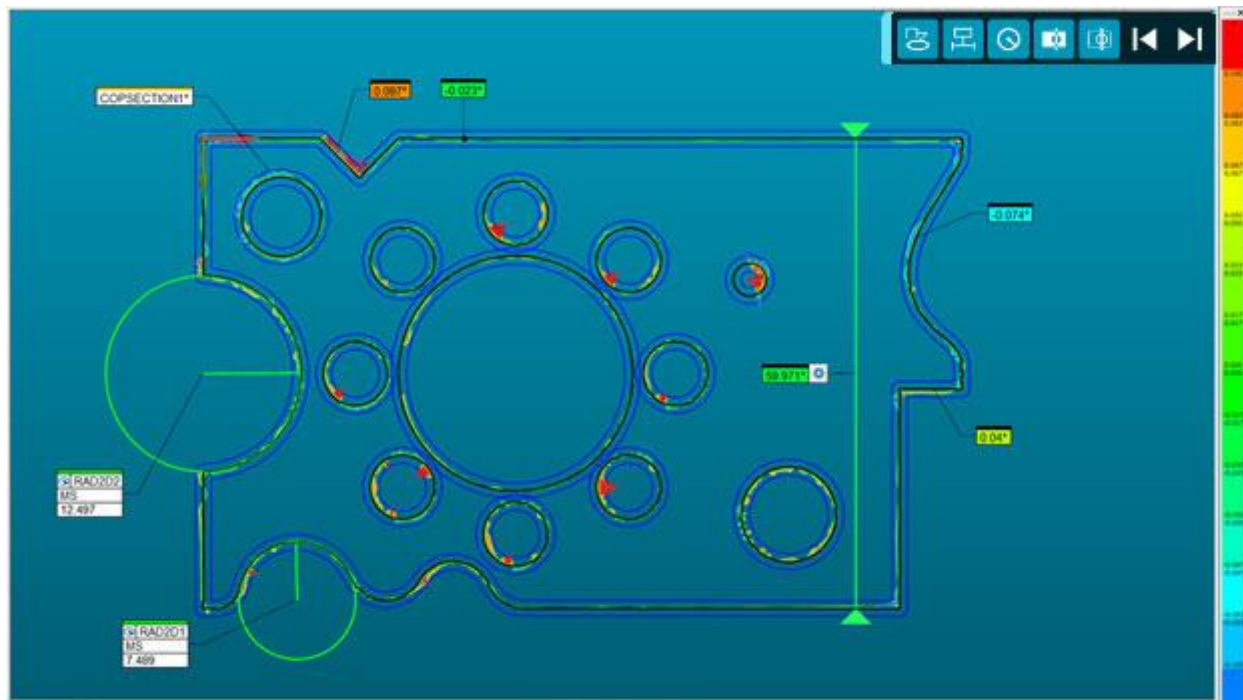
Example of a selected section of a cross section

- b. Right-click the selected cross section to display the Edit window's pop-up menu.



- c. Click the **Show Only** option to display the 2D view of the selected cross section. When you enable the option, PC-DMIS displays a check mark to the left of it.

When in 2D view, the **Cross Section Graphic Control** toolbar is available.



Example of a section view normal to the cross section



As you hover and move the cursor over the cross section in the Graphic Display window, the labels appear and update in real time. Click any point on the cross section when in 2D view to create an annotation label for that location.

The **Cross Section Graphic Control** toolbar is a floating toolbar that you can position anywhere in the Graphic Display window.



Cross Section Graphic Control toolbar

The buttons from left to right perform these functions:

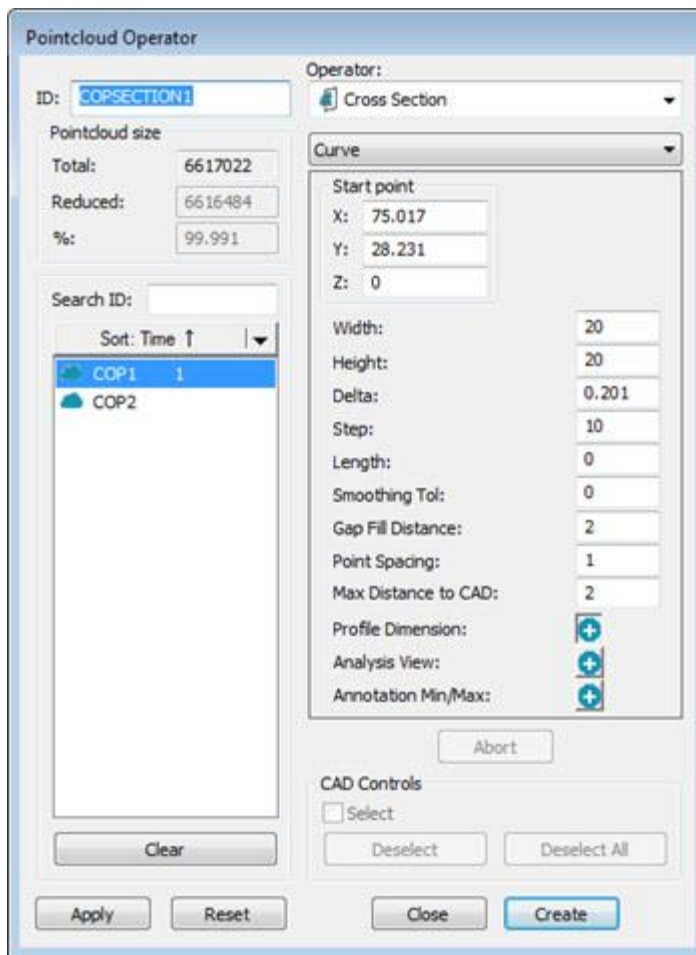
- **Show/Hide annotations** - Shows and hides the annotations.
- **Show/Hide distance gages** - Shows and hides the distance gages.
- **Show/Hide 2D radius gages** - Shows and hides the 2D radius gages.
- **Show/Hide nominal polylines** - Shows and hides the nominal polylines.
- **Show/Hide measured polylines** - Shows and hides the measured polylines.

- **Show the previous 2D Section** - From the currently-selected cross section in the Edit window, each time you click this button, the software displays the previous cross section through to the first cross section.
- **Show the next 2D Section** - From the currently selected cross section in the Edit window, each time you click this button, the software displays the next cross section through to the last cross section.

Click the **Show the previous 2D Section** or the **Show the next 2D Section** button to toggle backward or forward to view the cross sections in a slide show pattern. For details, see the "Cross Section Slide Show" section in the "Show and Hide Cross Section Polylines" topic.

Creating a Cross Section along a Curve

You can create a cross section along a curved feature with the **Curve** function of the **Pointcloud Operator** or **Mesh Operator** dialog box. The cross section is created normal to the CAD curve.



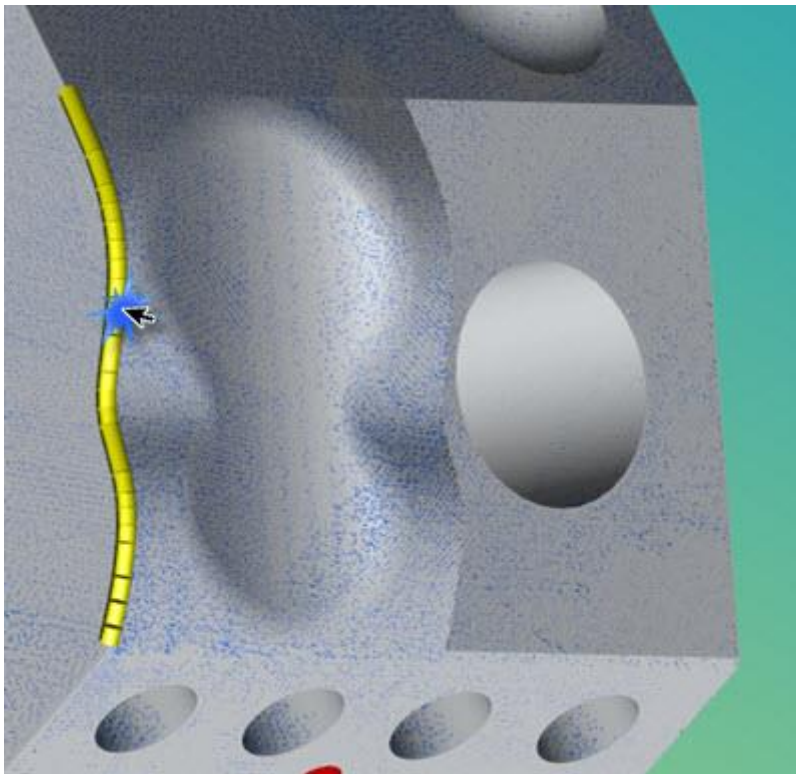
Pointcloud Operator dialog box - CROSS SECTION Operator, Curve function selected

To create a cross section along a curve:

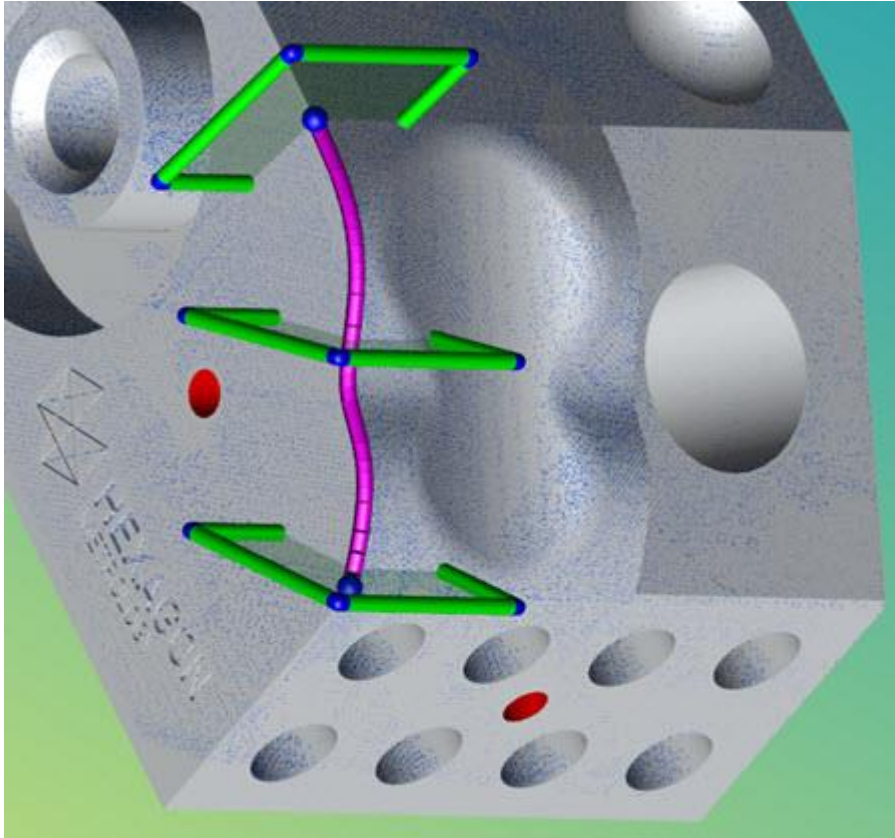
1. For cross sections created with a COP as the input, click **Insert | Pointcloud | Operator** to display the **Pointcloud Operator** dialog box.

For cross sections created with a Mesh as the input, click **Insert | Mesh | Operator** to display the **Mesh Operator** dialog box.

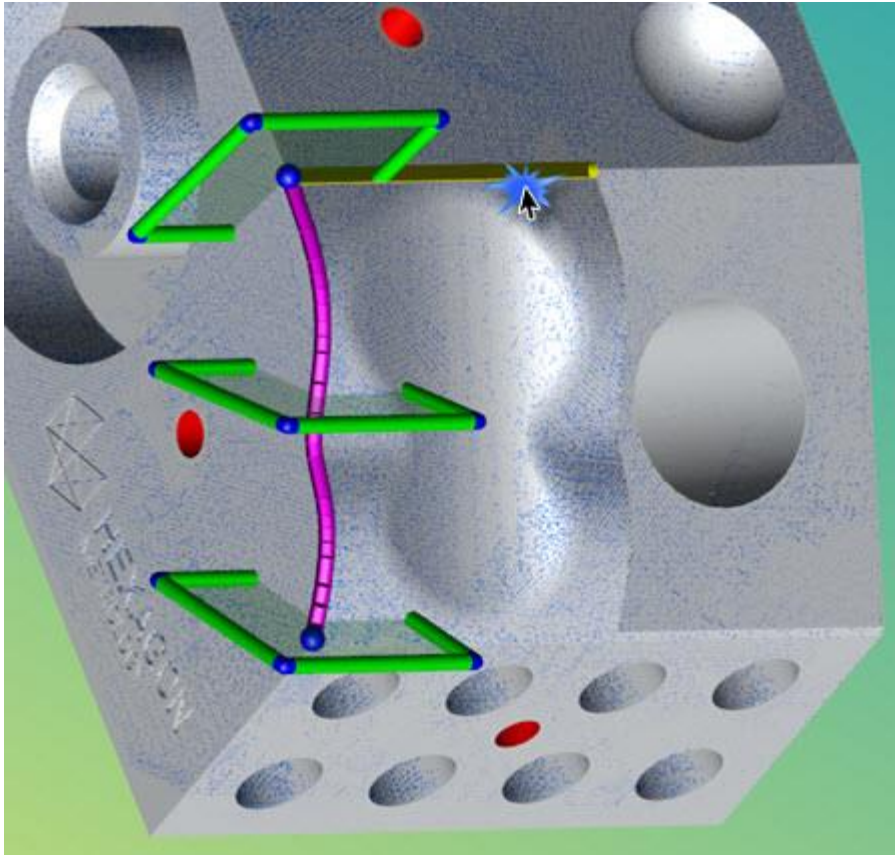
2. Select the **Cross Section** operator from the **Operator** list, and then the **Curve** function from the list underneath the **Operator** list.
3. In the Graphic Display window, hover over any curved feature. PC-DMIS automatically detects and highlights the curve.



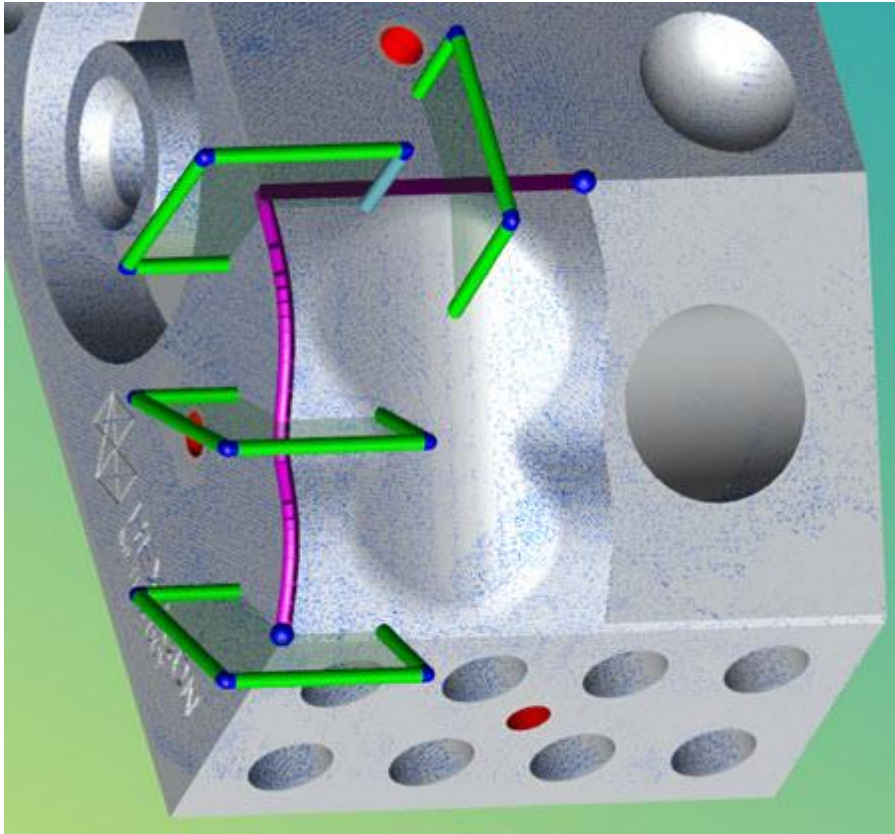
4. Click the highlighted edge that you want to create cross sections on. PC-DMIS automatically generates the cross sections.



Hold down the Ctrl key while hovering over the next edge to select multiple contiguous edges.



Click the edge to select it.

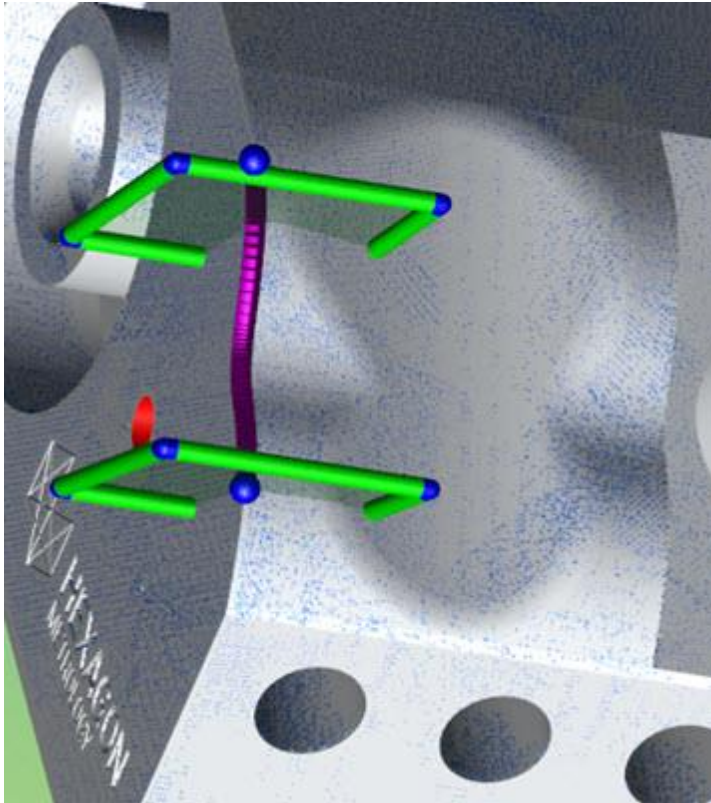


Select as many edges in this manner as required.

To de-select an edge, press the Ctrl key and hover over the first or last edge (it turns red) and then left-click it.

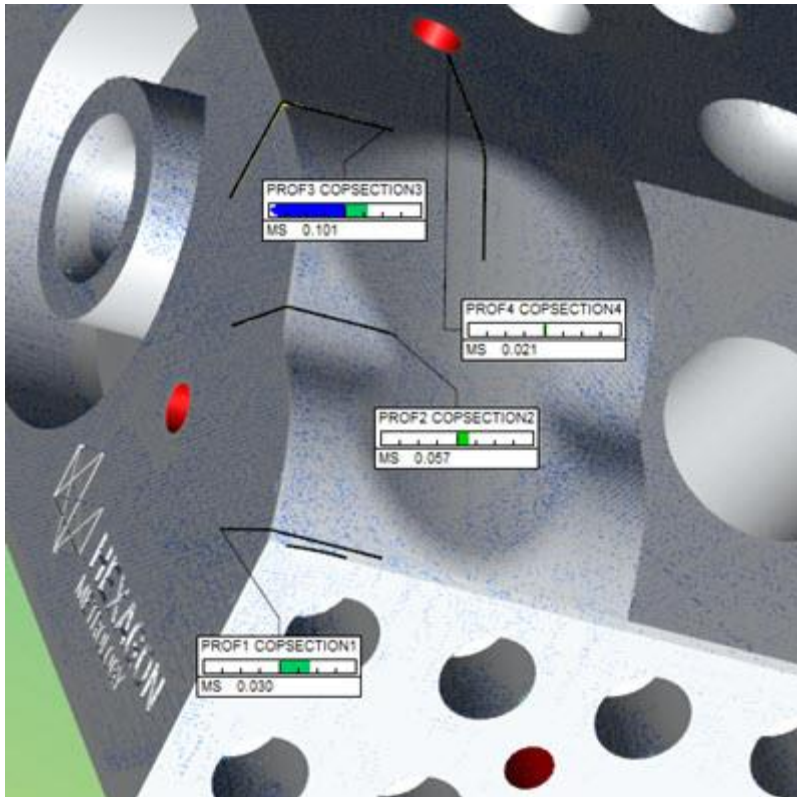
To de-select all edges, click the **Reset** button.

5. Drag the **Start** or **End** points (the blue ball handles) of the curve length line (the purple line) to define only a portion of the curve. If the updated section is too short, click the **Reset** button to cancel and repeat from step 3.



The dialog box values update automatically when changes are made to the **Start** or **End** points of the defined cross section.

6. When done, click **Apply** to create the polylines. Click **Create** to generate the cross sections in the Edit window.



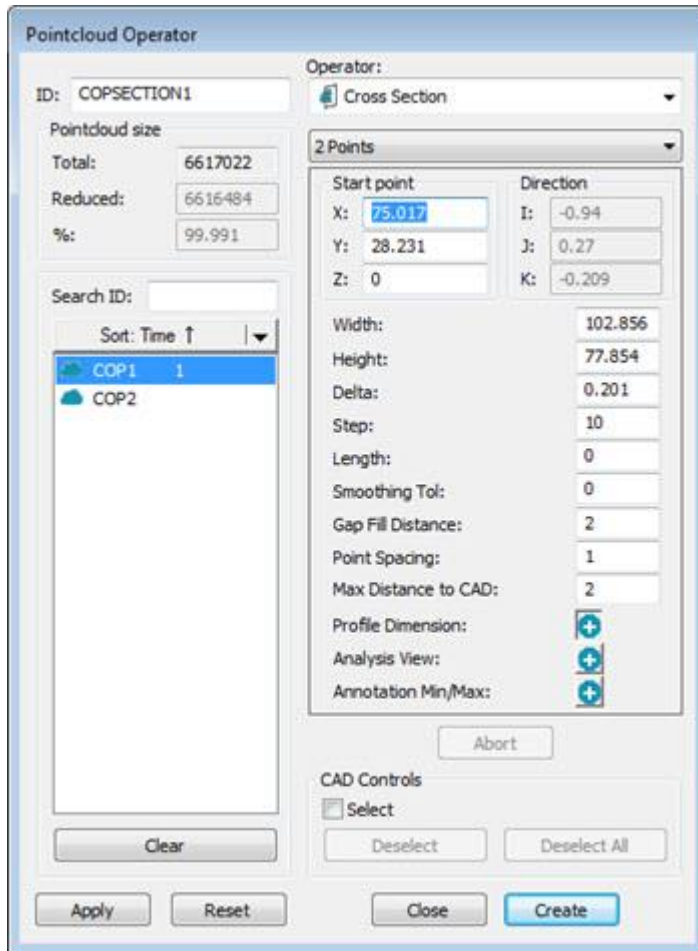
The black polylines represent the nominal CAD. The yellow polylines represent the measured polyline.

Smoothing the Cross Section along the Curve

You can smooth the cross section created along a curve with the **Smoothing Tol** option in the **Pointcloud Operator** or **Mesh Operator** dialog box. For details, see the "**Smoothing Tol**" description in the "Cross Section" topic.

Creating a Cross Section between 2 Points

You can create a cross section between two points with the **2 Points** function from the **Pointcloud Operator** or **Mesh Operator** dialog box.



Pointcloud Operator dialog box - CROSS SECTION Operator, 2 Points function selected

The 2 Points cross section is created between two selected points and is oriented normal to the current Graphic View. The cross section's purple **Length** line is perpendicular to the line defined by the two selected points. It is created at the midpoint of this line and defaults to 0 (zero).

To create a cross section between two points:

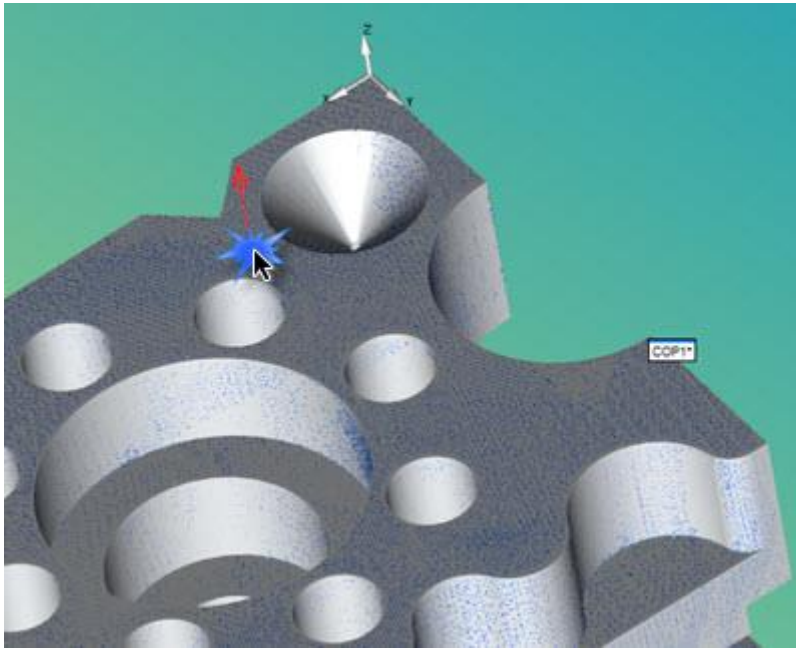
1. For cross sections created with a COP as the input, click **Insert | Pointcloud | Operator** to display the **Pointcloud Operator** dialog box.

For cross sections created with a Mesh as the input, click **Insert | Mesh | Operator** to display the **Mesh Operator** dialog box.

2. Select the **Cross Section** operator from the **Operator** list, and then select the **2 Points** function from the list underneath the **Operator** list.
3. From the **QuickMeasure** or **Graphic View** toolbar, select the correct Graphic View for the cross section orientation. For details on the **QuickMeasure** toolbar,

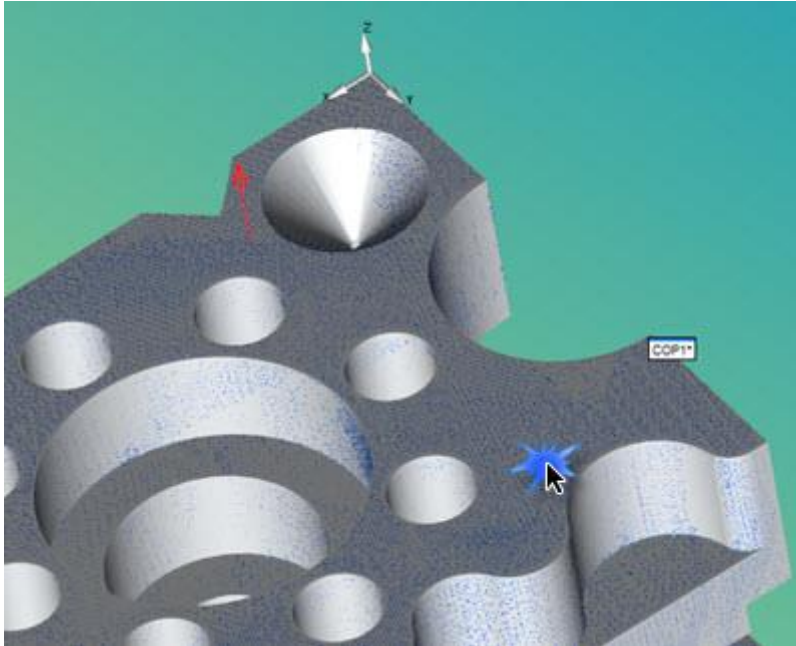
see "QuickMeasure Toolbar" in the PC-DMIS CMM documentation. For details on the **Graphic View** toolbar, see "Graphic View Toolbar" in the "Using Toolbars" section of the PC-DMIS Core documentation.

4. From the Graphic Display window, click where you want to define the cross sections's first point.

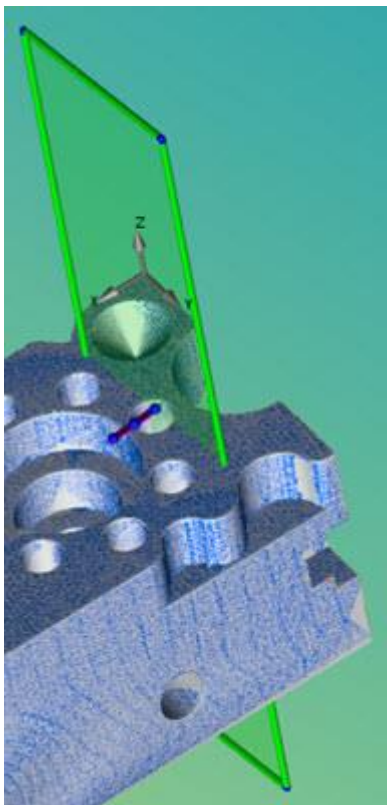


The point's vector appears as a red arrow normal to the selected surface.

5. From the Graphic Display window, click where you want to define the cross sections's second point.



Once the second point is clicked, the cross section is displayed.



6. Adjust the cross section's properties as needed.

Show and Hide Cross Section Polylines

You can show or hide cross section features that have been created.

Show or Hide Cross Section Polylines from the Mesh, Pointcloud, or QuickCloud Toolbar

To show or hide cross section polylines:


1. From the **Mesh**, **Pointcloud**, or **QuickCloud** toolbar (**View | Toolbars**), click the **Cross Section** drop-down arrow to display the **Cross Section** toolbar:



Pointcloud cross section drop-down toolbar



Mesh cross section drop-down toolbar

2. Click the **2D Section Slide Show** button  to display the 2D view of the cross sections in the Graphic Display window.
3. From the floating **Cross Section Graphic Control** toolbar in the Graphic Display window, click the appropriate button to perform the described action:




Show/Hide nominal polylines button - Click to hide or show the black nominal polylines.



Show/Hide measured polylines button - Click to hide or show the yellow measured polylines.

Cross Section Slide Show

The **2D Section Slide Show** button enables the floating **Cross Section Graphic Control** toolbar in the Graphic Display window. From the floating toolbar, use the **Show the previous 2D Section** and **Show the next 2D Section** buttons to display each

cross section in the respective order. You can tell that the cross section slide show is enabled when the button appears to be pressed in .



If the measurement routine contains both COPSECTIONS and MESHSECTIONS, the **Show the next 2D Section** and **Show the previous 2D Section** buttons allow you to navigate to the next section, either pointcloud or mesh.

Once you enable the cross section slide show, from the floating toolbar, click **Show the previous 2D Section** and **Show the next 2D Section** to display individual cross sections in 2D view (Show Only view):

1. From the **QuickCloud** toolbar, click the **Cross Section** drop-down arrow to display the **Cross Section** toolbar.
2. Click the **2D Section Slide Show** button. The software displays a 2D view of the cross section and the **Cross Section Graphic Control** floating toolbar. You can reposition the toolbar anywhere in the Graphic Display window. The floating toolbar contains these buttons you can use to navigate through each cross section in 2D view in the Graphic Display window:



Show the previous 2D Section - Click to display the cross section *before* the currently selected one in the Edit window in 2D view. The CAD graphic disappears. Click the button repeatedly to cycle backwards until you reach the first cross section.



If you do not select a cross section, the software selects the first one above the current cursor position in the Edit window. Consequently, nothing happens if there are no cross sections defined above the current cursor position. The same occurs if you select the *first* cross section in the list and you click this button.



Show the next 2D Section - Click to display the cross section *after* the currently selected one in the Edit window in 2D view. The CAD graphic disappears. Click the button repeatedly to cycle forward until you reach the last cross section.



If you do not select a cross section, the software selects the first one below the current cursor position in the Edit window. Consequently, nothing happens if there are no cross sections defined below the current cursor position. The same occurs if you select the *last* cross section in the list and you click this button.

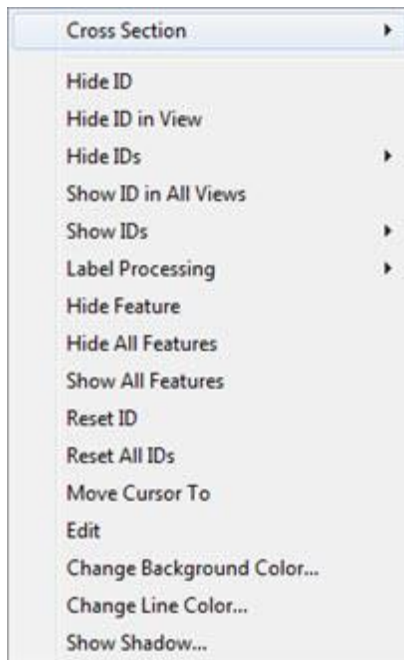
For details on the **Cross Section Graphic Control** floating toolbar, see "2D View of Cross Sections".

3. Click the **2D Section Slide Show** button a second time to exit the slide show and get the CAD graphic (3D view) back.

Show or Hide Cross Section Polylines from the Graphic Display Window

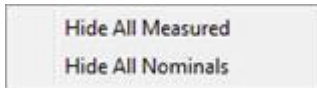
To hide cross section polylines from the Graphic Display window:

1. Right-click any cross section label in the Graphic Display window to display the pop-up menu.



2. Hover over the **Cross Section** option to display the **Cross Section** menu.

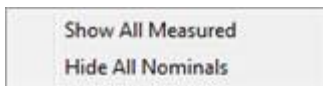
If the measured and nominal cross section polylines are visible, the **Cross Section** menu has these options:



If the measured and nominal cross section polylines are NOT visible, the **Cross Section** menu has these options:



You may also have a mixture of the above options depending on the visible state of the polylines, such as:



3. Click the appropriate option to show or hide the associated polylines.

Measuring Cross Section Distances

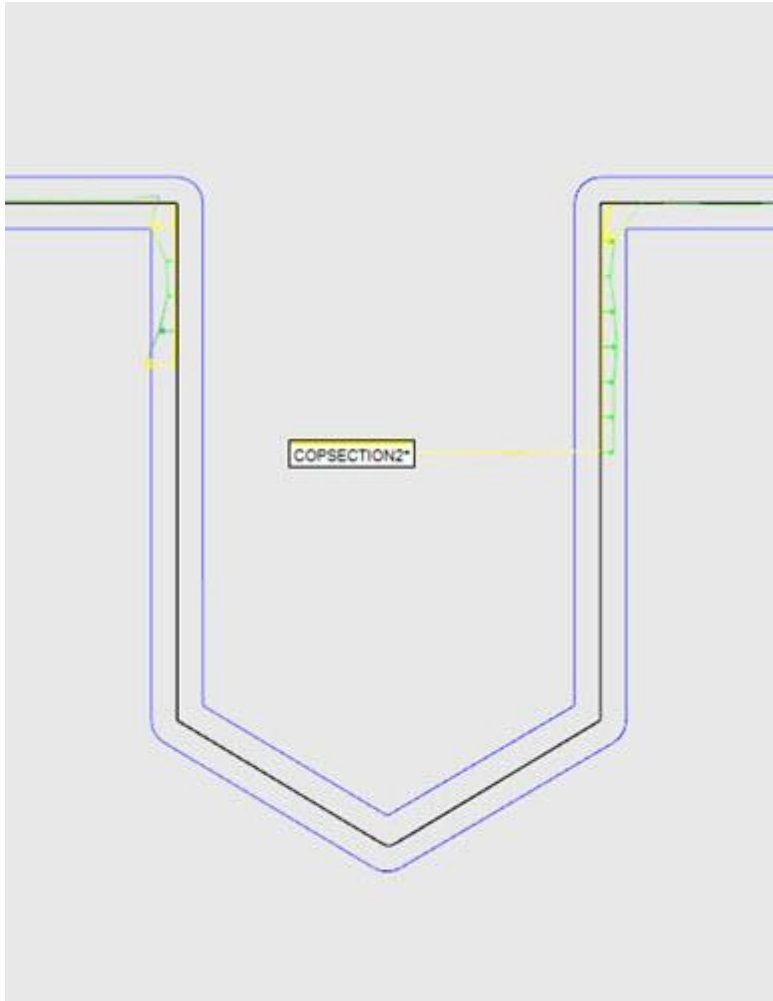
Distances can be measured on 2D cross sections in the Graphic Display window. You must already have the cross sections created and be in cross-section 2D view. For details on how to view cross sections in 2D view, see "Show and Hide Cross Section Polylines".

To create a cross-section Distance Gage:

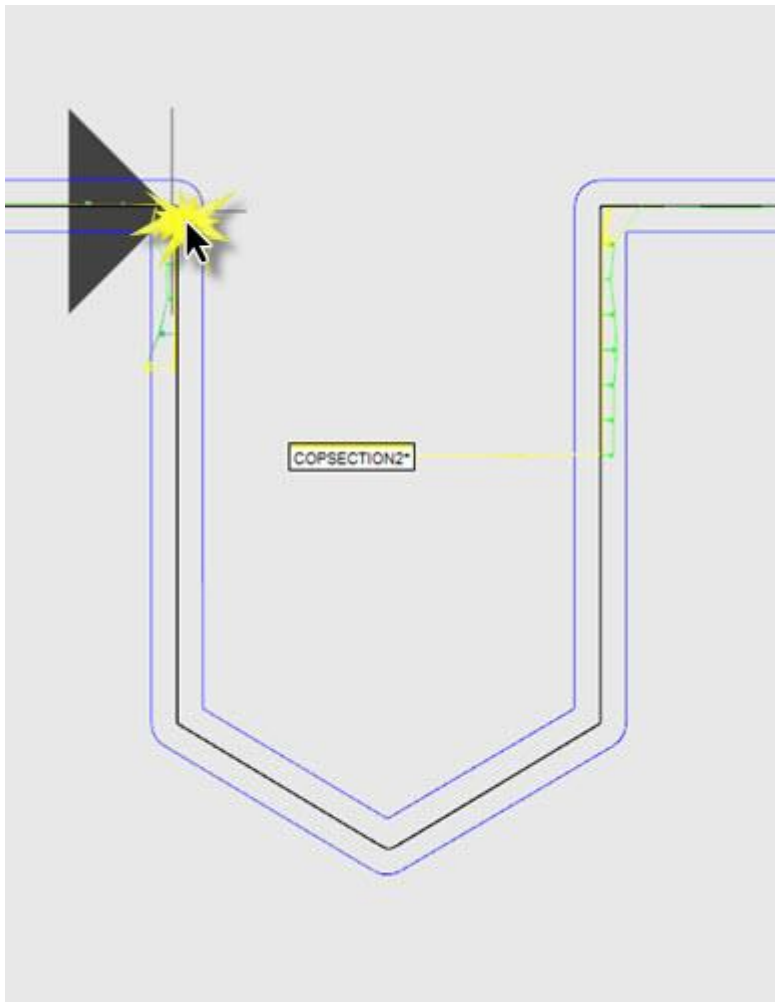
1. From the **Pointcloud**, **QuickCloud**, or **Mesh** toolbar (**View | Toolbars**), click the **Cross Section** drop-down arrow to display the **Cross Section** toolbar.



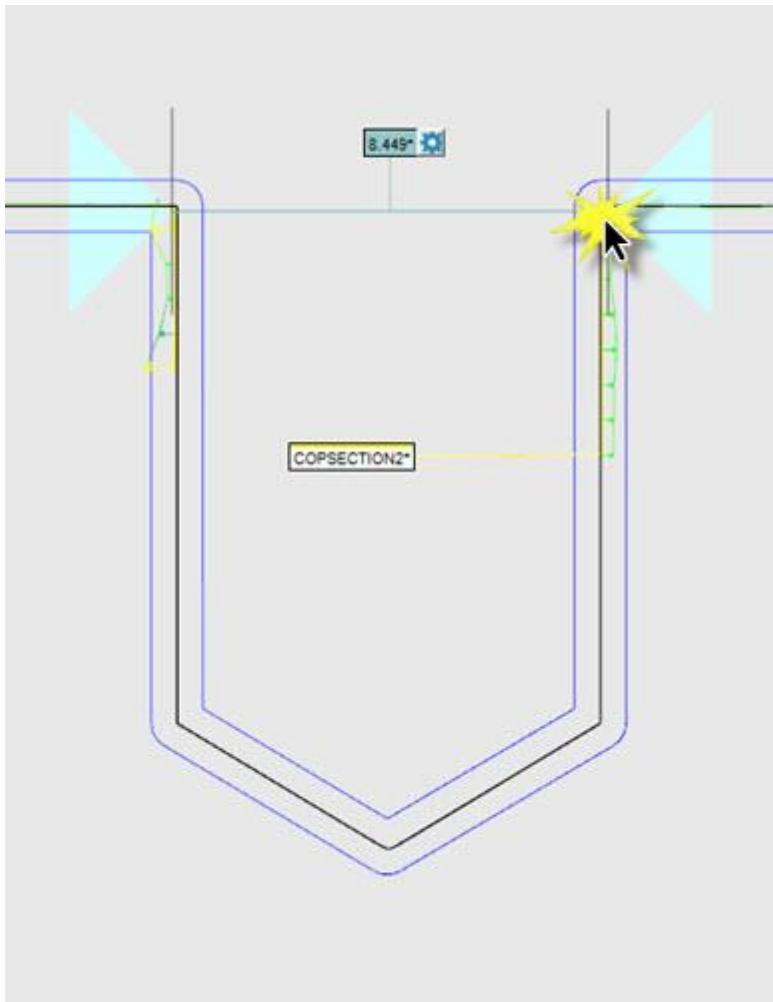
2. Click the **2D Section Slide Show** button () to enter 2D view.
3. Click the **Show the previous 2D Section** or **Show the next 2D Section** button until the cross section is displayed in the Graphic Display window.



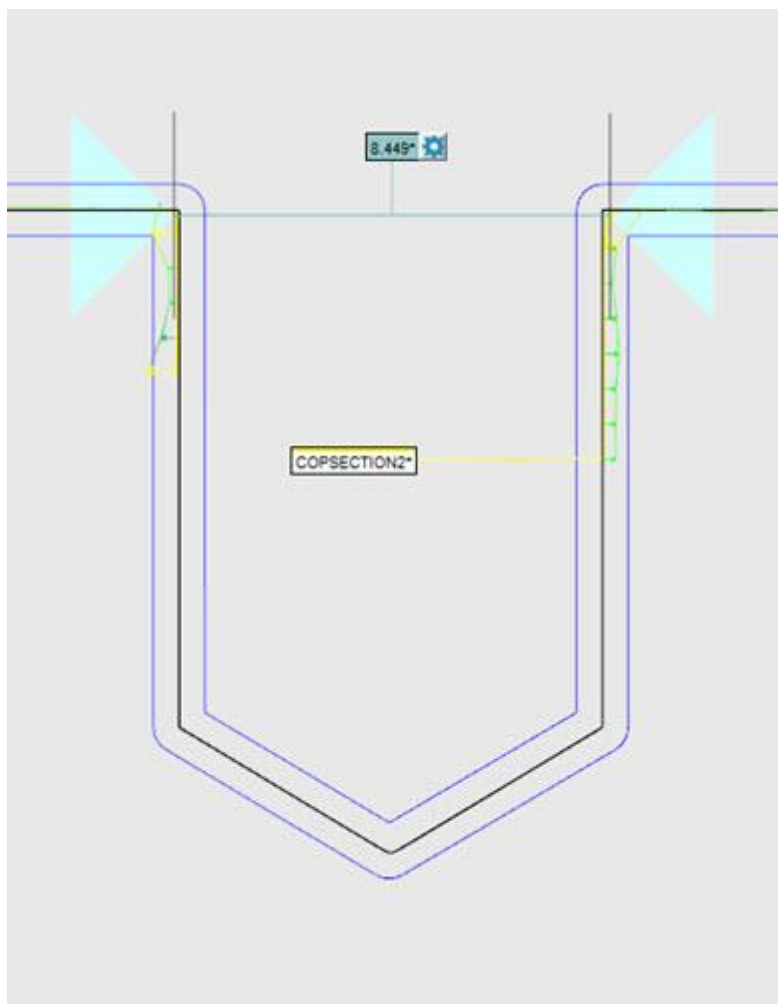
4. In the Graphic Display window, hover over the cross section, and then click and drag to display the start point.



5. Drag the cursor to the end point, and click to select it. The Distance Gage is calculated, created, and displayed in the 2D view with its associated label.



As you drag the cursor, the software intuitively detects if the start and end points are along an axis. If they are, the direction is recognized and constrained parallel to that axis.



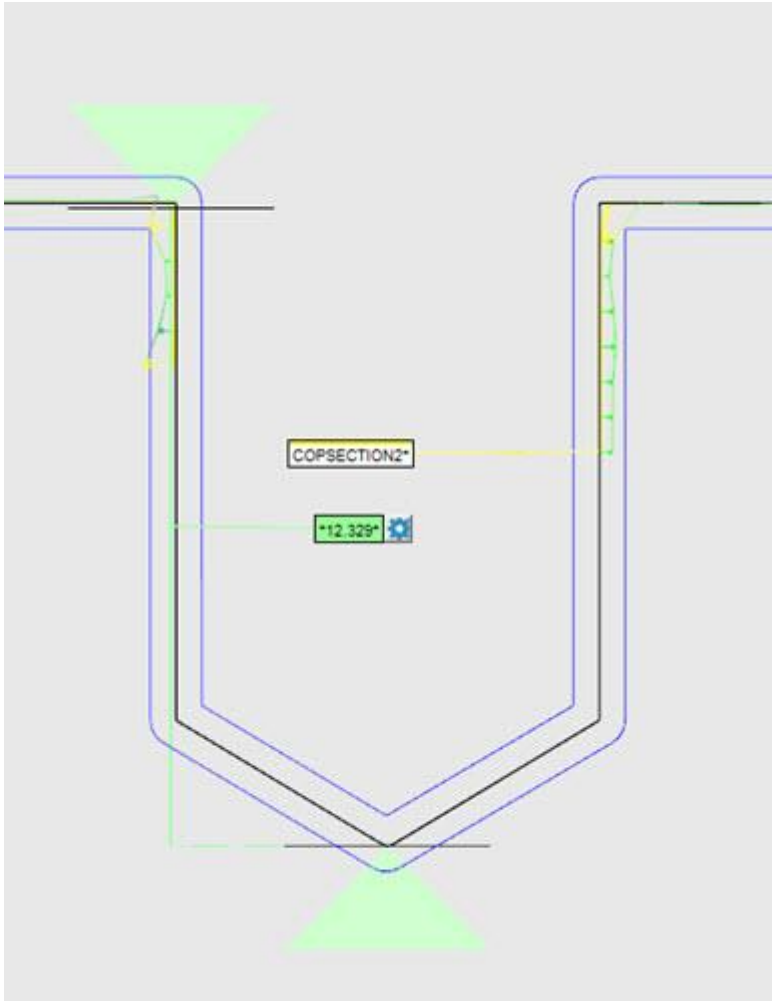
Parallel Distance Gage example

To create a Distance Gage parallel to the first side picked:

- a. Press and hold the Shift key.
- b. Click the start point, and drag and click the end point.

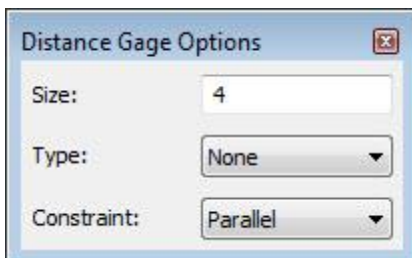
An example of this would be if the cross section was not created along the X, Y, or Z axis.

If the start and end points are offset from one side to another, the axis direction is still recognized. The distance, however, is calculated parallel but between the offset points.

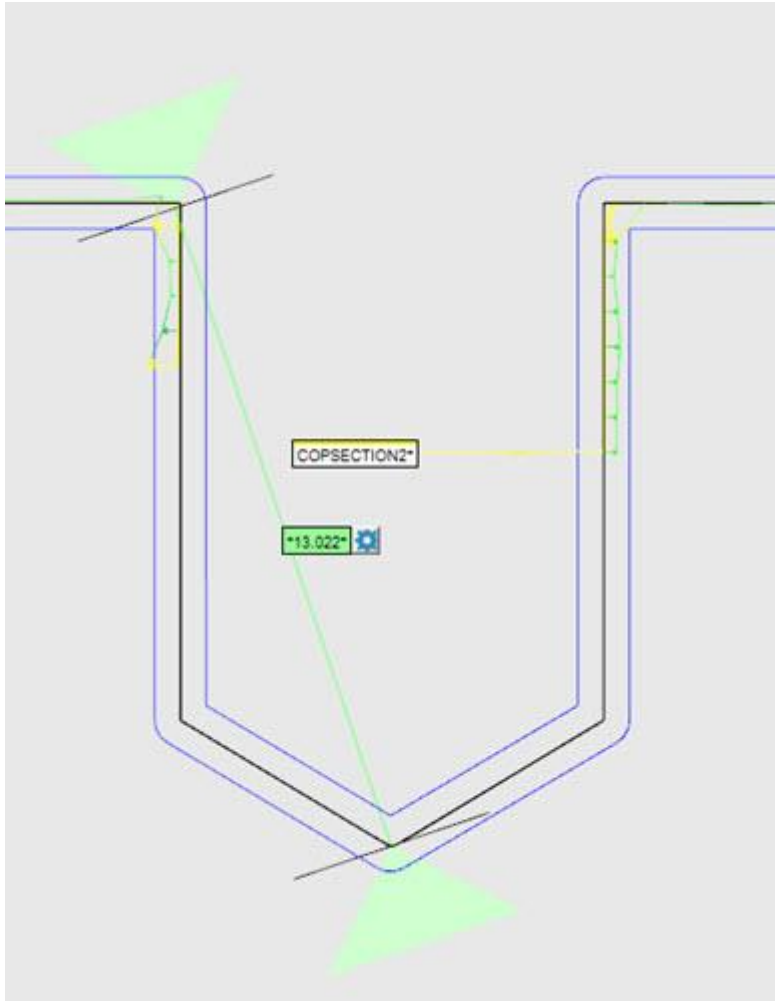


Offset Distance Gage example

- To change the properties of the Distance Gage, click the **Distance Gage Options** button (⚙️) on the label. The **Distance Gage Options** dialog box appears.



For example, if you do not want the Distance Gage calculated as an offset calculation, select the **Parallel** option from the **Constraint** list. Click the start and end points as before; the Distance Gage is calculated between the two points.



Example of a Distance Gage calculated with the Parallel Constraint option selected

7. Edit the properties of the Distance Gage:

Size - If the **None** option is selected in the **Type** list, the **Size** value is used to determine the size of the start and end point icons in the Graphic Display window. If either the **Best Fit**, **Max Fit**, or **Min Fit** option is selected from the **Type** list, the **Size** value is used as described below. A size of 4 is the default.

Type - Click the drop-down arrow to display these options:

- **None** (default) - A Point-to-Point distance calculation between the closest cross section polyline points based on the selected start and end points.
- **Best Fit** - A Least Square line is calculated based on all the yellow points within the first pick zone, defined by the **Size** value (the default is 4) and the selected start point. This is repeated for the second pick zone, defined by the **Size** value and the selected end point. The centroid of the first Least

Square line is projected onto the measurement zone line. This is repeated for the centroid of the second Least Square line. The distance is between these two projected points.

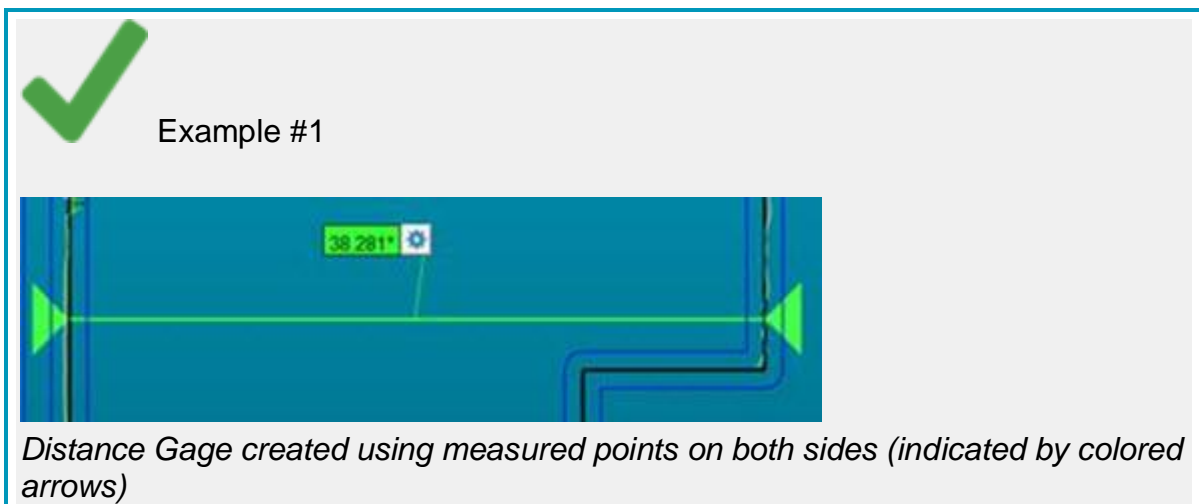
- **Max Fit** - Defined by the point farthest out in the first pick zone, defined by the **Size** value and the selected start point, and by the point farthest out in the second pick zone, defined by the **Size** value and the selected end point. The Max Fit points are projected onto the measurement zone line. The Max distance is between these two projected points.
- **Min Fit** - Defined by the closest point in the first pick zone, defined by the **Size** value and the selected start point, and in the second pick zone, defined by the **Size** value and the selected end point. The Min Fit points are projected onto the measurement zone line. The Min distance is between these two projected points.

If the **Type** option is changed, the measured distance is automatically recalculated, and the updated value is displayed based on the option selected.

Constraint - Select **None** (default) if you do not want to constrain it to any axis. Select the appropriate option to constrain the Distance Gage to the **X**, **Y**, or **Z** axis, or **Parallel** to calculate the distance parallel to the first selected side.

Creating a Distance Gage with and without Measured Points

You can create a distance gage with or without measured points on either side of the gage.





Example #2



Distance Gage created using measured points only on one side

In this case, PC-DMIS precedes the distance value with an asterisk. This indicates that one or more sides are not measured. The value shows the distance between the nominal (gray arrow side) and the measured side.



Example #3



Distance Gage created with no measured points on either side (gray arrows)

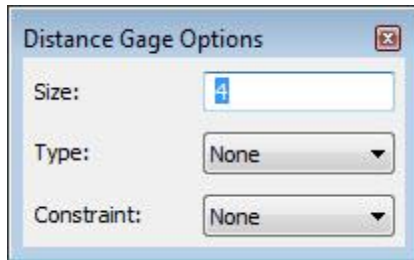
In this case, the distance gage shows the nominal value.

Creating a 3D Distance Gage

To create a 3D Distance Gage that is not constrained to any axis:

1. Press and hold the Ctrl key, hover over the cross section in the Graphic Display window, and click and drag to display the start point.
2. Continue to drag the cursor with the Ctrl key pressed to the end point location.
3. Click to select the end point and display the Distance Gage and its associated label.

The same functionality is available as described earlier for 2D Distance Gages. Click the **Distance Gage Options** button to view the **Distance Gage Options** dialog box. The **Constraint** option is set to **None**.

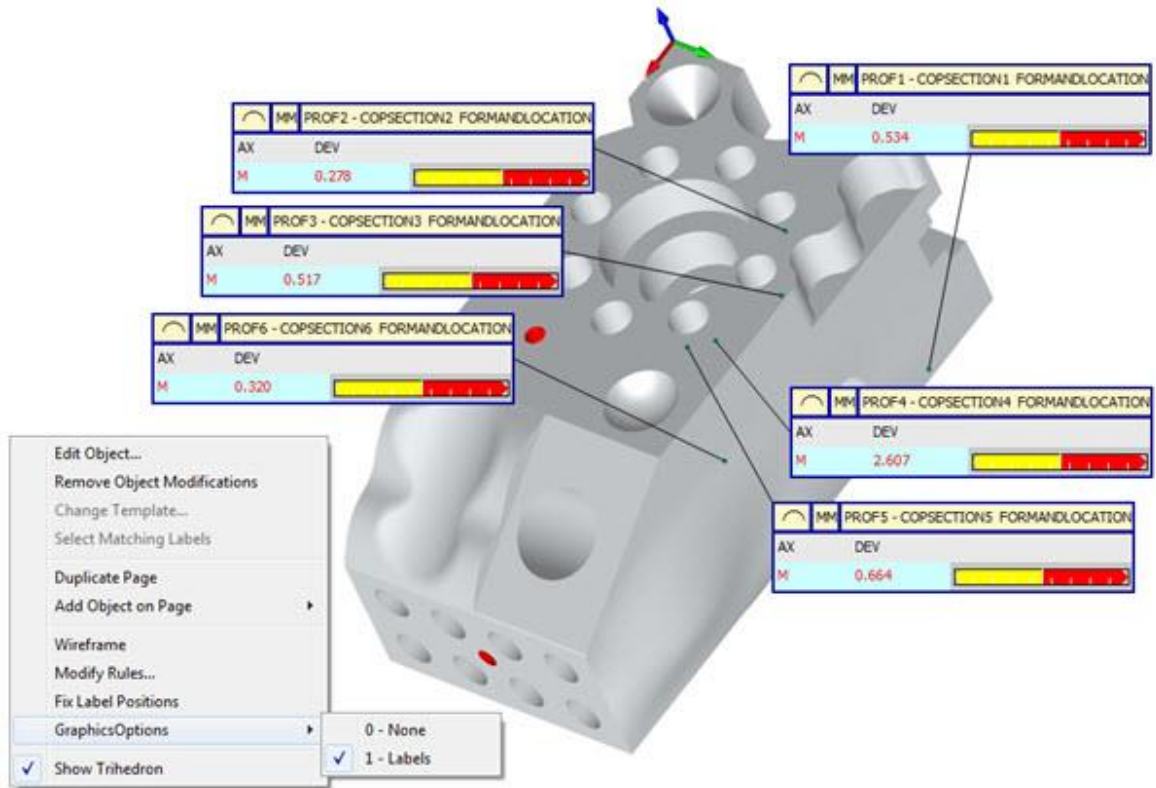


Viewing Cross Section Labels in Reports

You can view cross section Annotation and Distance Gage labels in reports in two ways:

Viewing labels from a report template that has a graphic image

1. From any report template that has a graphic image, right-click the image to display a pop-up menu.

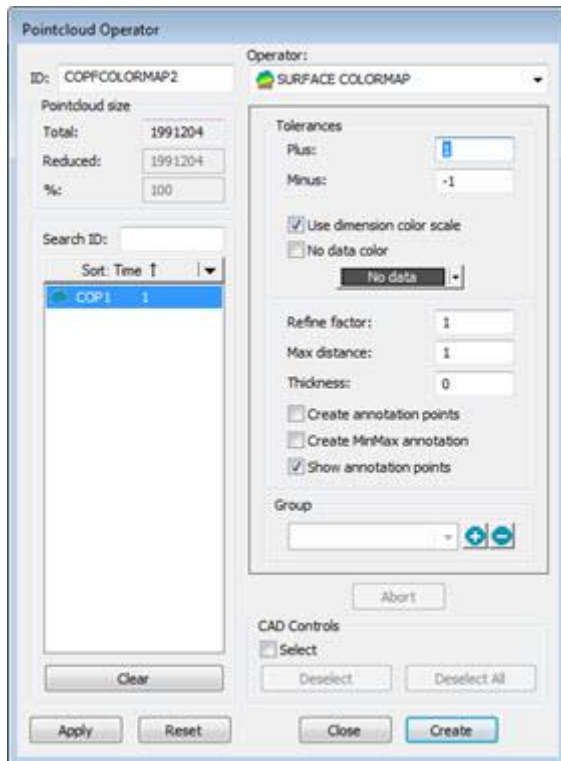


2. Click **GraphicOptions** and then **1 - Labels** to display all the labels in your report. Click **0 - None** to hide all the labels.

Viewing labels in the Report Graphical Analysis template from the Cross Section dialog box

1. Create the **Annotations** and **Distance Gage** items for your cross sections. For details on creating **Annotations**, see the "Cross Section" help topic. For details on creating **Distance Gage** items, see the "Measuring Cross Section Distance" help topic.
2. Create the Analysis View. For details on the [Analysis View](#) command, see the "Analysis View" description in the "Cross Section" help topic.
3. Click the **Graphical Analysis** option in the Report window (**View | Report**). The annotation and gage labels are automatically visible.

SURFACE COLORMAP




Pointcloud Operator dialog box - SURFACE COLORMAP Operator

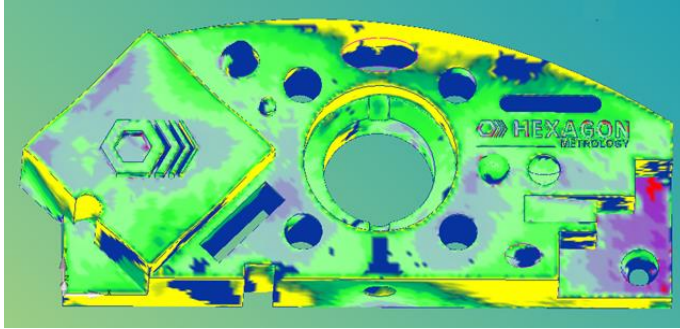
The SURFACE COLORMAP operation applies a colored shading to the CAD model. The model is shaded according to the deviations of the cloud of points compared to CAD. The model uses the colors defined in the **Edit Dimension Color** dialog box and the tolerance limits specified in the **Upper tolerance** and **Lower tolerance** boxes discussed below.

The colors used for the colormap are defined in **Edit Dimension Color** dialog box (**Edit | Graphic Display Window | Dimension Color**).

Select **View | Other Windows | Dimensions Colors** to view the color scale from the Dimensions Color Bar.

To apply the SURFACE COLORMAP operation to a Pointcloud, click the **Pointcloud**

Surface Colormap button () on the **Pointcloud** toolbar (**View | Toolbars | Pointcloud**), or select **Insert | Pointcloud | Surface Colormap**.



Example of a Surface COLORMAP applied to selected CAD elements

The SURFACE COLORMAP operator has these options:

Tolerances - Use this option to set the upper (Plus) and lower (Minus) tolerance values:

Plus - The upper tolerance value.

Minus - The lower tolerance value.

Use dimension color scale check box - When you select this check box, the software defines the color bar used for the Surface Colormap color properties by the **Dimensions Color Bar**. For details on the **Dimensions Color Bar**, see "Using the Dimensions Colors Window (Dimensions Color Bar)" in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.



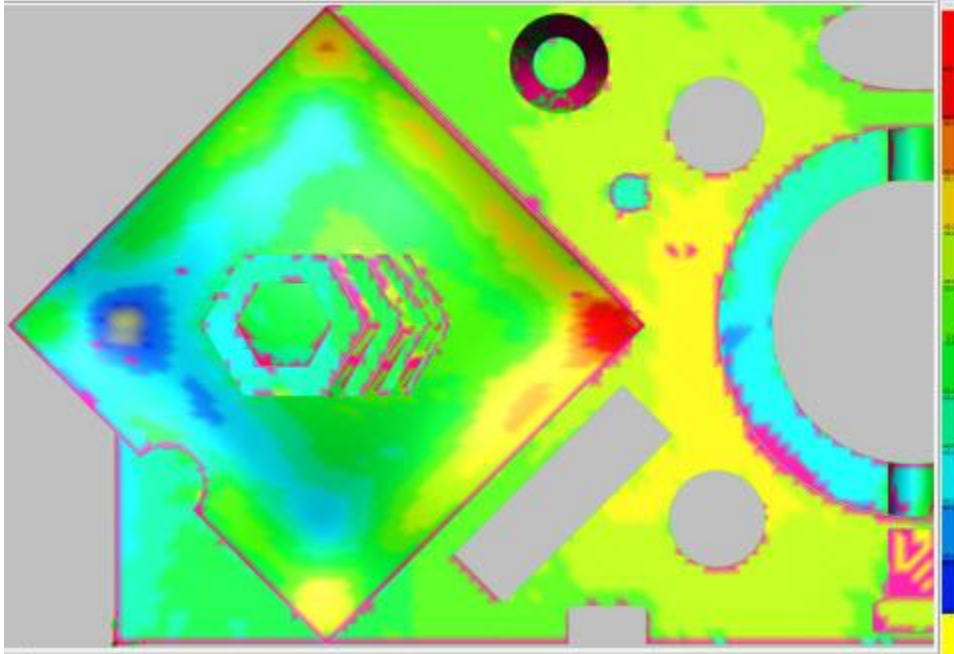
Edit Color Scale - When the **Use dimension color scale** check box is cleared, PC-DMIS enables the **Edit Color Scale** button. When you click this button, the functionality to dynamically change the color, scale, and threshold of the surface and point colormap properties becomes available through the **Color Scale Editor** dialog box. For details, see the "Edit the Color Scale" topic.

No data color check box - When you select this check box, the software maps the selected color to areas on the selected surfaces where no data is found.

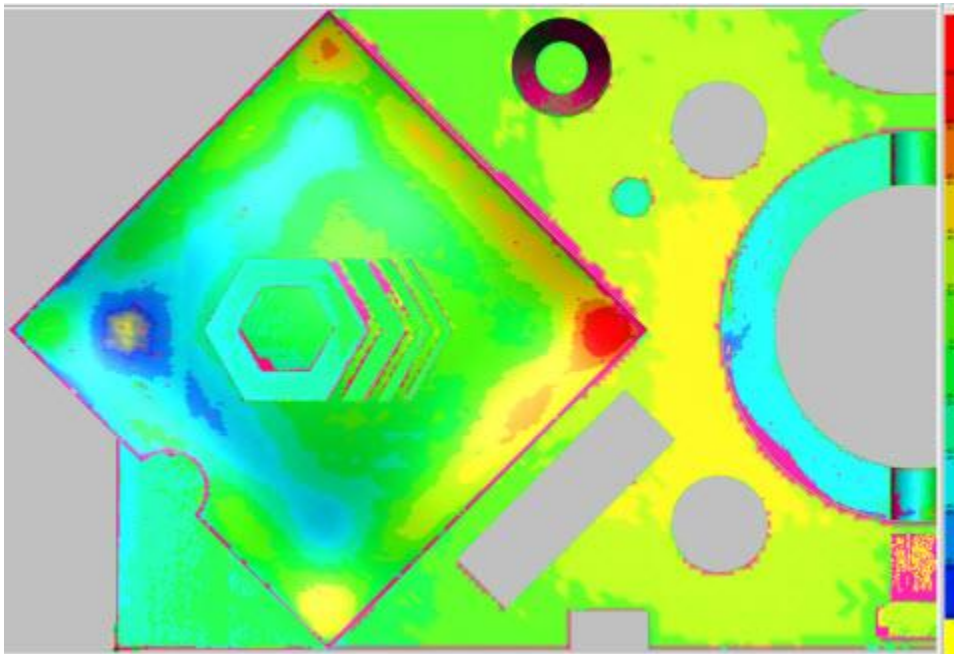
Refine Factor - This option adjusts the accuracy of the Surface colormap. If you change this value, PC-DMIS draws a new and changed colormap. The underlying measured data does not change. The colormap tessellates the CAD model with an overlay of colored triangles. The vertices of each triangle are colored with the color that corresponds to its deviation from the pointcloud. The colors are taken from the dimensions color scale discussed above. By using a smaller or larger refine factor value, you can generate a finer or coarser tessellation, respectively. You may want to decrease the refine factor to obtain a smoothly shaded CAD with a more accurate deviation representation. However, setting a smaller refine value

results in a larger number of triangles, thereby increasing the computation time and the size of the CAD model. For comparison, note that the number of triangles for a refine factor of 0.5, compared to a refine factor of 1.0, is about 4 times more; whereas a refine factor of 0.1 compared to 1.0, is about 100 times more.

Pointcloud COLORMAP example with a Refine Factor of 1:



Pointcloud COLORMAP example with a Refine Factor of 0.1:



Max distance - The software only includes points that fall within the **Max distance** value as part of the colormap. Note that if this value is too small, you may not see all the expected colored deviations. A good rule of thumb is to set this value slightly larger (10%, for example) than the largest deviation.

Thickness - This option adds a thickness value to deviations on the colormap. This is useful if you want to add a material thickness to a CAD surface model.

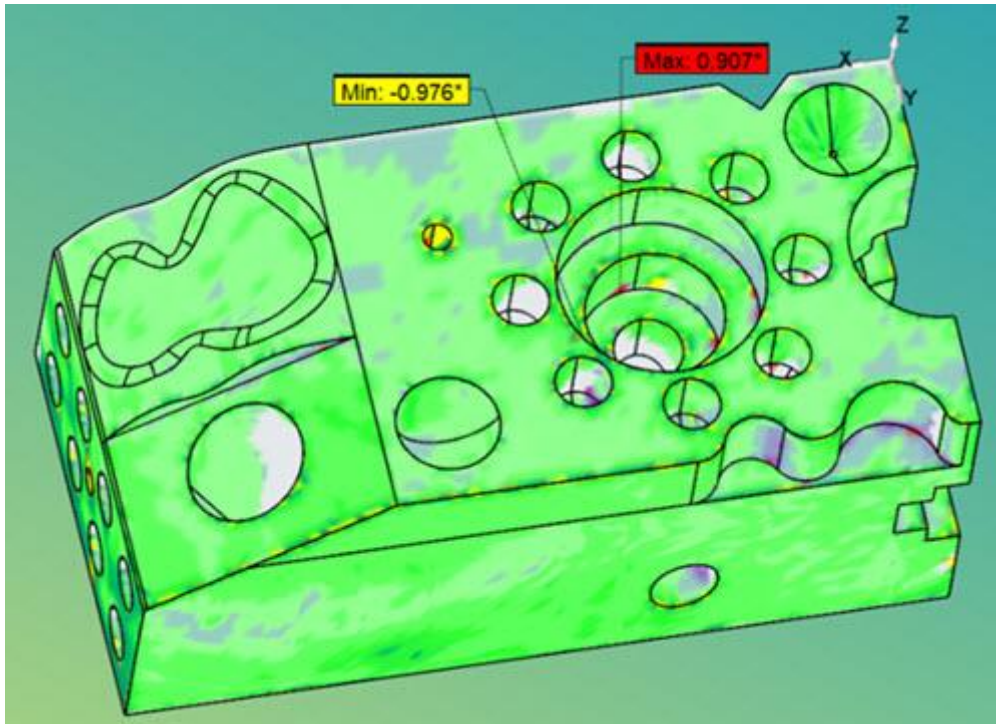
Create annotation points check box - Annotations are a way to display the deviation for a specific location on a surface colormap with its associated color. To create an annotation:

1. Click the **Create annotation points** check box to select it. This clears the **Select** check box in the **CAD Controls** area and disables most of the options on the right side of the dialog box.
2. Select a point on the CAD surface in the Graphic Display window. PC-DMIS evaluates and creates an annotation label in the same background color as the COP deviation point with the deviation value. You can move the label around in the Graphic Display window as any other label.



Once created, the annotation labels remain in the same position and have the same characteristics if you restart the measurement routine, or if you restart PC-DMIS and you reload the same measurement routine.

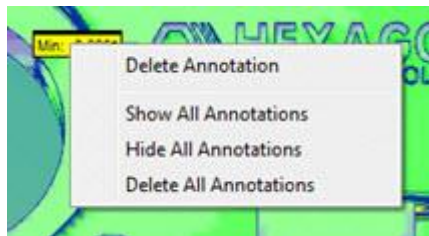
Create MinMax annotations check box - When you select this check box, the software creates the minimum and maximum values and displays them as annotation labels for the active COP Surface Colormap.



PC-DMIS calculates the minimum and maximum points each time you execute the measurement routine.

Show, Hide, or Delete Annotation Labels

To show, hide, or delete annotation labels, right-click one to display the pop-up menu and then select the appropriate option.



Delete Annotation - The software deletes the selected annotation label.

Show All Annotations - The software displays all annotation labels.

Hide All Annotations - The software hides all annotation labels.

Delete All Annotations - The software deletes all annotation labels.

Show annotation points check box - When you select this check box, the software displays all annotation points.

Group - You can use this option to create, modify, or identify Surface Colormap groups. For details, see "Method 2" in the "Apply COLORMAP to a CAD model with Multiple Surface Profile Tolerances" topic.

Click **Abort** to undo any calculations that were generated after you click the **Apply** button.

CAD Controls - This option lets you apply the operation to CAD elements you select. See the scan area of the "CAD Controls" topic for details.

Click **Create** to insert a `COP/OPER, SURFACE COLORMAP` command into the Edit window.



For example:

```
COPFCOLMAP2=COP/OPER, SURFACE COLORMAP, PLUS  
TOLERANCE=0.25, MINUS TOLERANCE=-0.25, THICKNESS=0  
  
REF, COP1, ,
```

Colormaps in the Report

For information on how the software shows colormaps in the report, see "Colormaps and the CadReportObject" in the "Reporting Measurement Results" chapter of the PC-DMIS Core documentation.

Apply COLORMAP to a CAD model with Multiple Surface Profile Tolerances

There are two methods to apply a Surface Colormap when the CAD model has multiple surface-profile tolerances.

Method 1

Create multiple Surface Colormaps, one for each tolerance or Surface Profile.

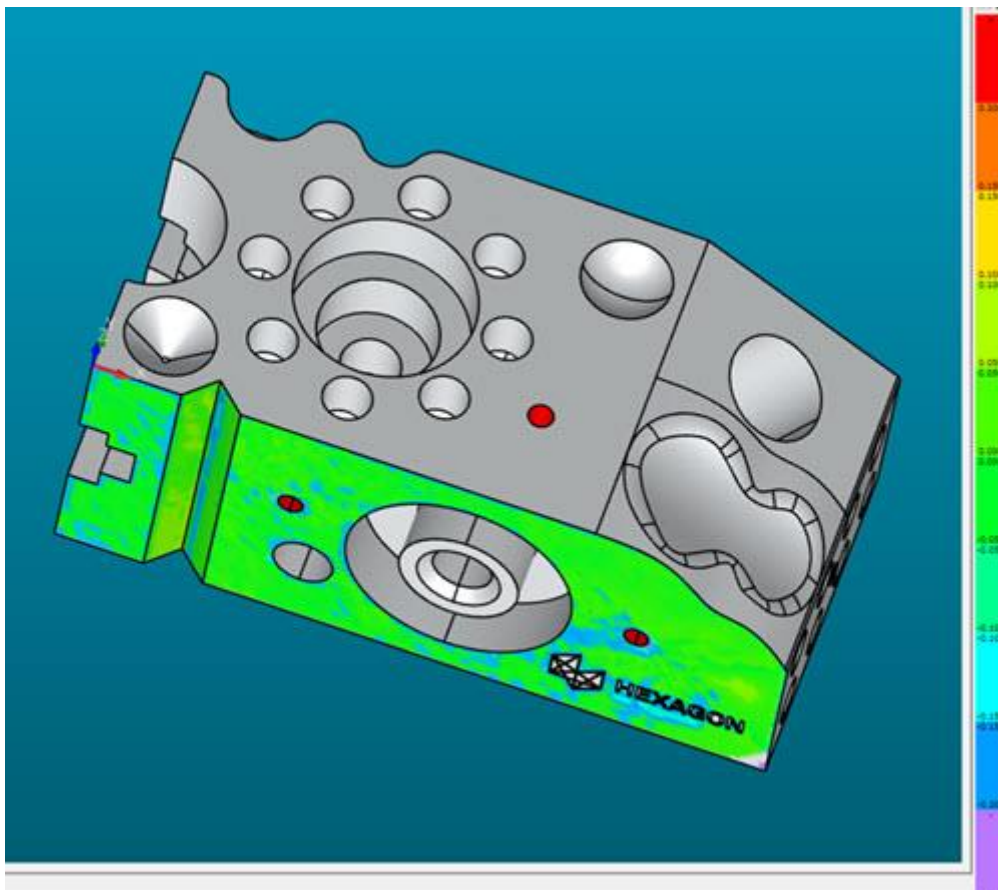
To create multiple Surface Colormaps, do the following:

1. From the **Pointcloud** toolbar, select the **Pointcloud Surface Colormap** button



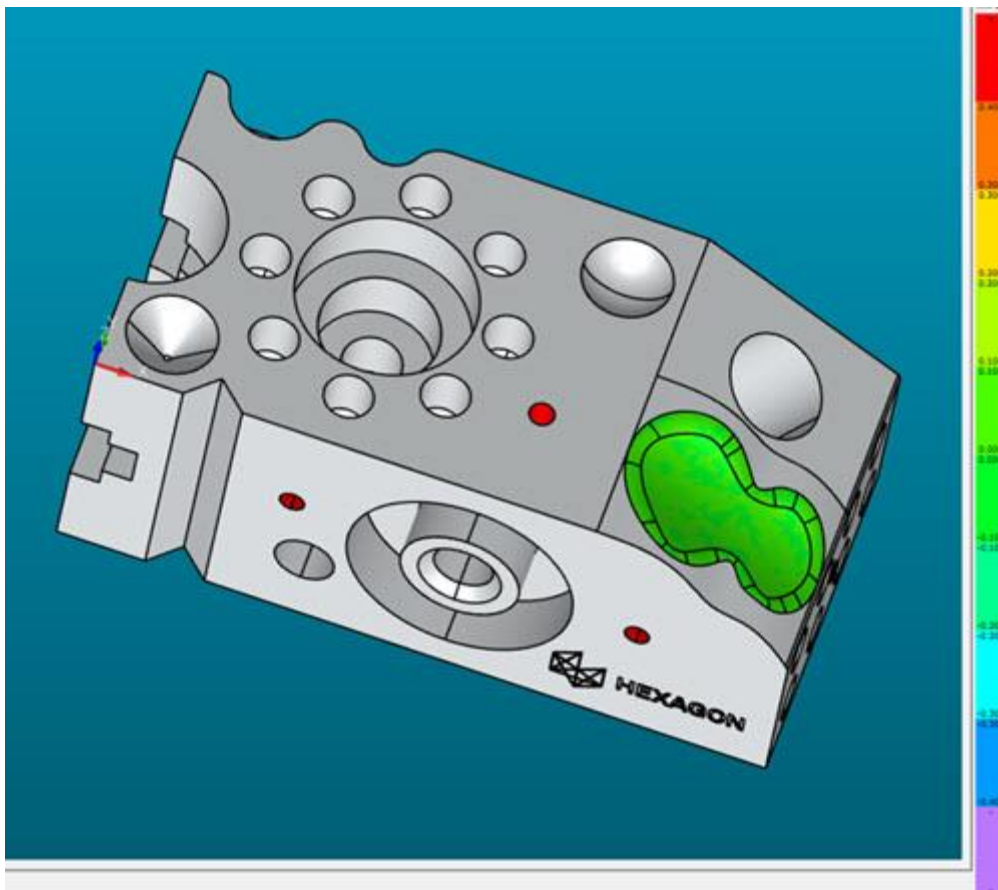
(). The **Pointcloud Operator** dialog box for the Surface Colormap appears.

2. Enter the tolerances.
3. Select the specific CAD surfaces. For details on selecting CAD surfaces, see "Working with CAD Surfaces" in the "Scanning Your Part" chapter in the PC-DMIS Core documentation.
4. Click **Apply** to apply the Surface Colormap to the selected CAD surface.



Example of the Surface Colormap applied to the first-selected CAD surfaces

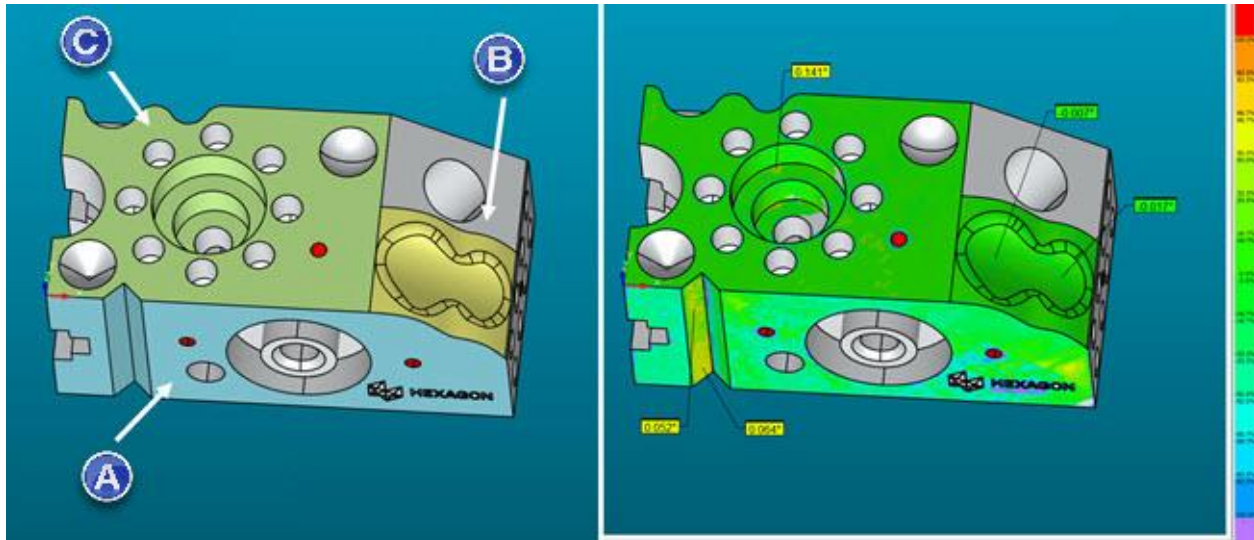
5. Click **Create** to add the Surface Colormap in the Edit window.
6. Create a second Surface Colormap in the same manner for the next Surface Profile.



Example of the second Surface Colormap applied to the selected CAD surfaces

Method 2

You can create groups of selected CAD surfaces within a single colormap. Each group can have different tolerances and Surface Colormap parameters (Refine factor, Max distance, and Thickness). If the Surface Colormap has two or more groups, the software displays the color scale with percentages.

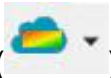


Examples:

Grouped CAD surfaces (left): (A) - Group01 TOL +/-0.1mm (B) - Group02 TOL +/-0.2mm (C) - Group03 TOL +/-

Surface Colormap applied to grouped CAD surfaces (right): The colormap image on the right represents the deviations in each group using the percentage of tolerances.

To create groups and apply different tolerances to the selected CAD surfaces within one colormap, do the following:

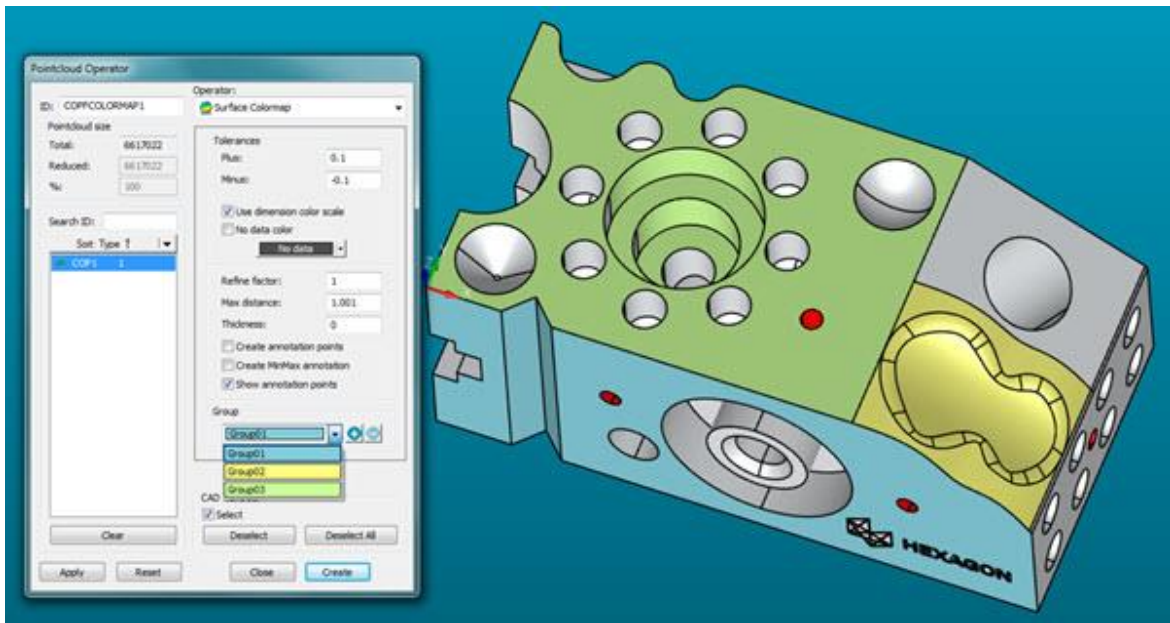
1. From the **Pointcloud** toolbar, select the **Pointcloud Surface Colormap** button
). The **Pointcloud Operator** dialog box for the Surface Colormap appears.
2. Enter the tolerance values and colormap parameters (**Refine factor**, **Max distance**, and other parameters).
3. From the **Pointcloud Operator** dialog box, select the **Select** check box in the **CAD Controls** area.
4. Click each of the CAD surfaces to be grouped. The surfaces highlight with the group color as you click them. Click the **Deselect** button to remove the last highlighted surface from the group.
5. To group the selected (highlighted) surfaces, click the **Add a new data group (+)** button located to the right of the **Group** list.

This group remains the active group until a new group is created. Any changes made to tolerances or COLORMAP parameters are applied to the active group. Also, if you select additional surfaces, they are added to the active group.

To identify which surface belongs to which group, the selected CAD surfaces are highlighted with the group color. To identify which group a grouped surface belongs to, press and hold the Shift key and then left-click the surface. The **Group** list updates to show the group to which it's assigned.

If you click a CAD surface that is not in the active group, it is removed from the group it is currently assigned to, and it is added to the active group.

6. To create another group, click the **Add a new data group (+)** button again, click the surfaces on the CAD, and update the tolerances and any COLORMAP parameters as necessary. Repeat to create more groups.



Example of grouped CAD surfaces

7. To make changes to a group, select it from the **Group** list and make the necessary changes.
8. To delete a group, select it from the **Group** list and click the **Remove the current data group (-)** button.



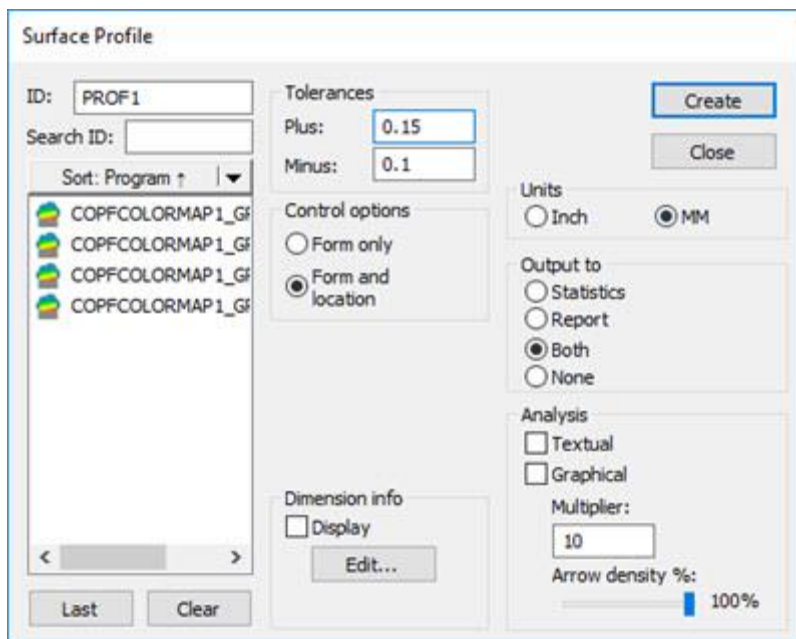
When a COLORMAP contains two or more groups with different tolerances, the color scale is automatically set to show the deviations with percentages.

Dimensioning Surface Profile using Pointcloud Colormap with Groups

You can use Pointcloud COLORMAP Groups to dimension Surface Profiles.

1. Create the Pointcloud COLORMAP Groups as described in Method 2.
2. For legacy dimensions, do this:

Click the **Profile Surface Dimension** option from the **Dimension** toolbar (**View | Toolbars | Dimension**). The software displays the **Surface Profile** dialog box for legacy dimensions:

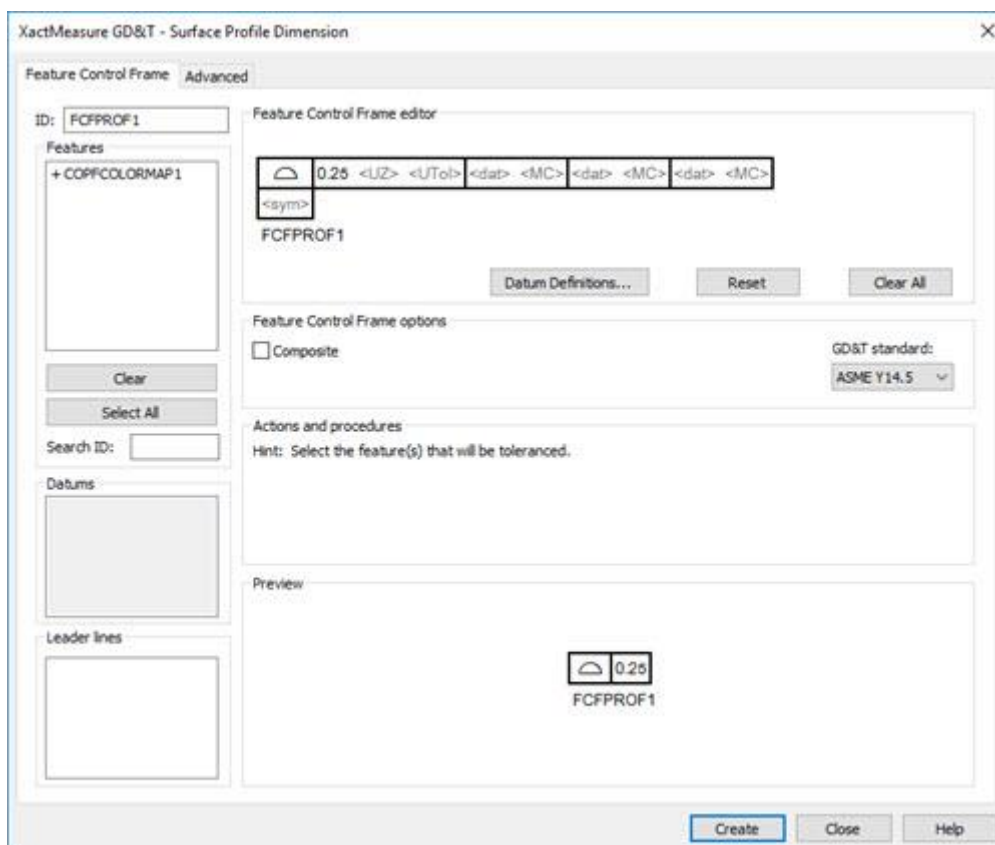


Surface Profile legacy dialog box for Pointcloud Colormap with Groups

For XactMeasure dimensions, do this:

Ensure that the **Use Legacy Dimensions** option (**Insert | Dimension | Use Legacy Dimensions**) is not marked.

Click the **Profile Surface Dimension** option from the **Dimension** toolbar. The software displays the **XactMeasure GD&T - Surface Profile Dimension** dialog box:



XactMeasure GD&T - Surface Profile Dimension dialog box for Pointcloud Colormap with Groups

Click the **+** sign to the left of the COPFCOLORMAP in the **Features** list to show any COLORMAP groups.

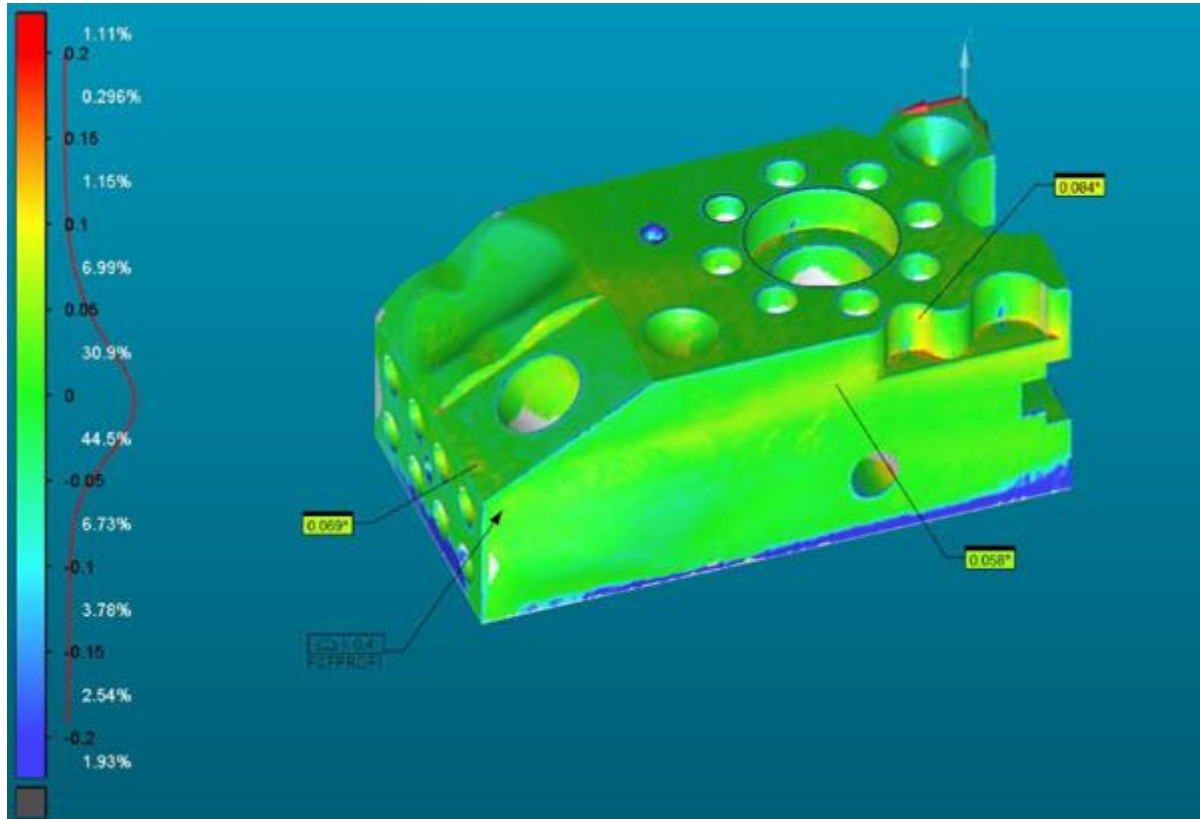


3. Select the desired COLORMAP groups and features to dimension from the **Features** list. If you select a datum feature, it must be a plane.
4. Set the other options as needed.

For details on creating a legacy Surface Profile, see "To Dimension a Feature Using the Surface Profile Option" in the "Using Legacy Dimensions" chapter of the PC-DMIS Core documentation.

Dimensioning Surface Profile Using the Pointcloud Surface COLORMAP

You can use a Pointcloud Surface COLORMAP to create a Dimension Surface Profile.



Example of a Dimension Surface Profile created using a Pointcloud Surface COLORMAP

To create a Dimension Surface Profile from a Pointcloud Surface COLORMAP:

1. Create a Pointcloud Surface COLORMAP. For details, see "POINT COLORMAP".
2. Use one of these dimensioning methods to create the Dimension Surface Profile:

Legacy Dimension

To create the Dimension Surface Profile for legacy dimensions:

- a. Make sure you have the **Use Legacy Dimensions** option selected (**Insert | Dimension | Use Legacy Dimensions**).

- b. Click the **Profile Surface Dimension** option from the **Dimension** toolbar (**View | Toolbars | Dimension**), or select it from the menu (**Insert | Dimension | Profile | Surface**). The **Surface Profile** dialog box opens.

The **Surface Profile** dialog box is shown with the following settings:

- ID:** PROF1
- Search ID:** (empty)
- Sort:** Program ↑
- Feature List:** CIR1, CIR2, PNT1, PNT2, **COPFCOLORMAP1 1** (selected)
- Tolerances:** Plus: 0.2, Minus: 0.2
- Control options:** ☐ Form only, ☒ Form and location
- Units:** ☐ Inch, ☒ MM
- Output to:** ☐ Statistics, ☐ Report, ☒ Both, ☐ None
- Analysis:** ☐ Textual, ☐ Graphical
- Multiplier:** 10
- Arrow density %:** 100%
- Dimension info:** ☐ Display, **Edit...** button
- Buttons:** Last, Clear, Create, Close

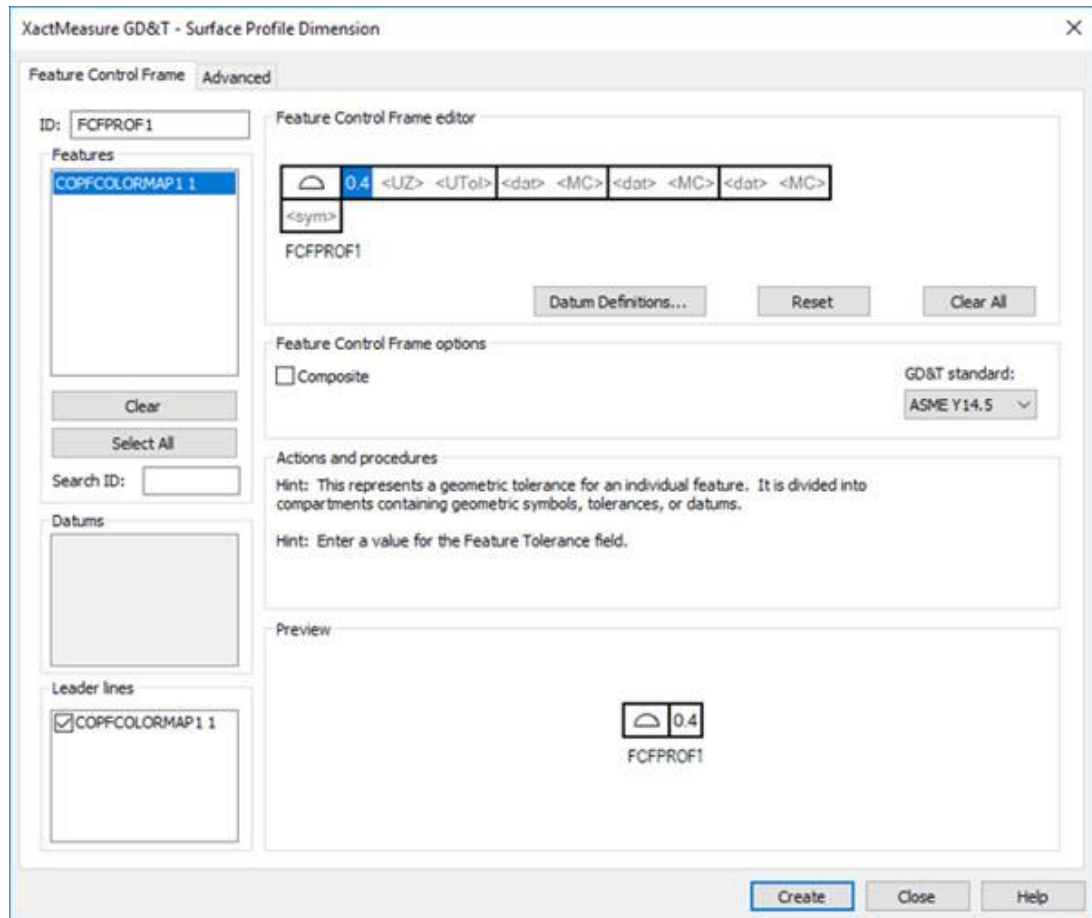
Surface Profile legacy dialog box for Pointcloud Surface COLORMAP

For details on creating a legacy Surface Profile, see "To Dimension a Feature Using the Surface Profile Option" in the "Using Legacy Dimensions" chapter of the PC-DMIS Core documentation.

XactMeasure Dimension

To create the Dimension Surface Profile for XactMeasure dimensions:

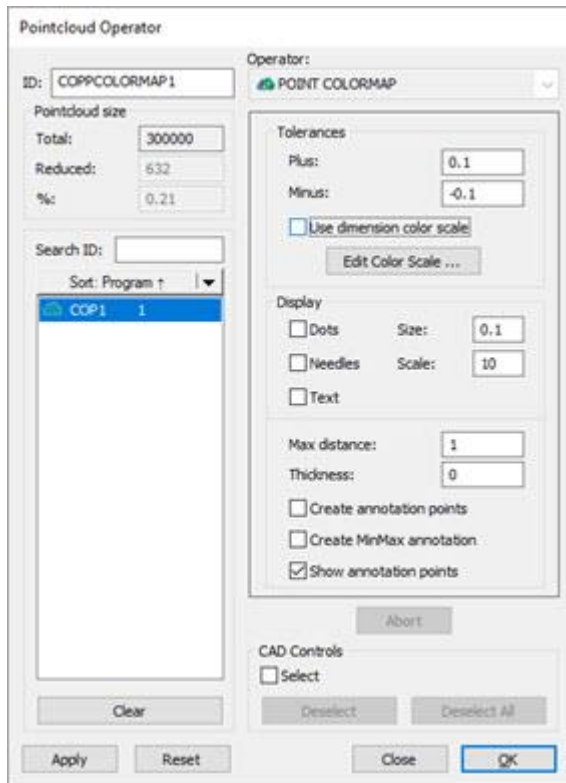
- a. Make sure the **Use Legacy Dimensions** option is NOT selected (**Insert | Dimension | Use Legacy Dimensions**).
- b. Click the **Profile Surface Dimension** option from the **Dimension** toolbar (**View | Toolbars | Dimension**), or select it from the menu (**Insert | Dimension | Profile | Surface**). The **XactMeasure GD&T - Surface Profile Dimension** dialog box opens.



XactMeasure GD&T - Surface Profile Dimension dialog box for Pointcloud Surface COLORMAP

3. Select the desired Pointcloud Surface COLORMAP from the **Features** list box.
4. Set the other options as needed.

POINT COLORMAP



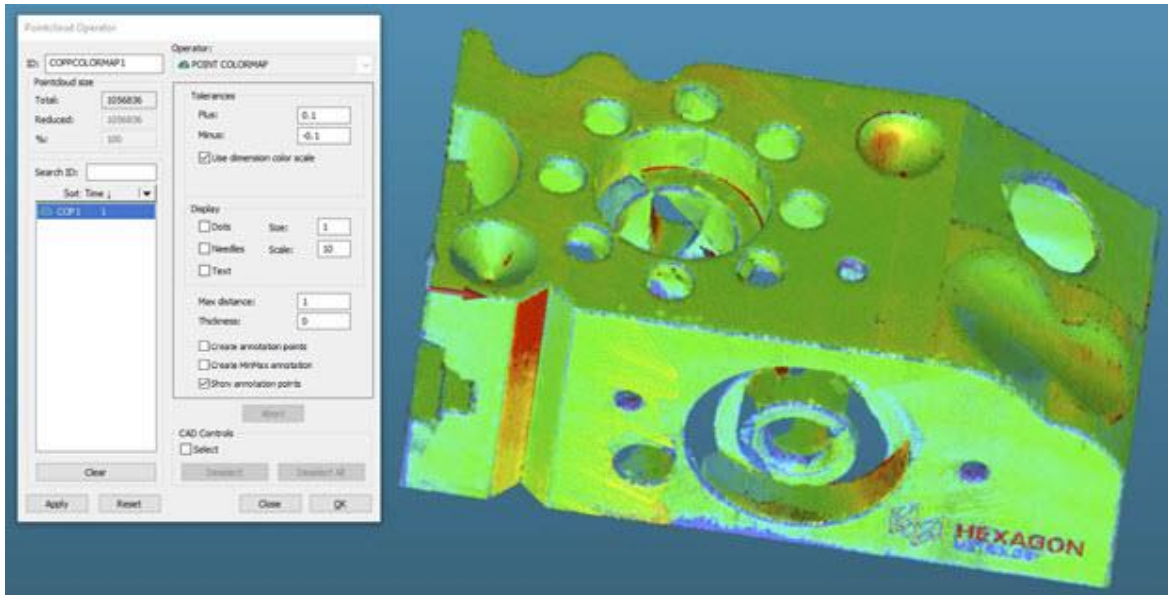
Pointcloud Operator dialog box - POINT COLORMAP Operator

The Point Colormap operation evaluates the deviations of the data points contained in a COP command compared to a CAD object. You can use Method 1 to color the points in the COP, or you can choose to represent the deviations by colored dots, colored needles that show the actual deviations, or as numerical values of the deviations with Method 2. You need to specify the plus and minus tolerances, and the scale to use.

You can create a Pointcloud Colormap in two ways:

Method 1: Uncheck all three of the check boxes (**Dots**, **Needles** and **Text**) in the **Display** area of the **Pointcloud Operator** dialog box.

With all three of the **Display** check boxes unchecked, PC-DMIS projects the points onto the tessellated CAD model. The software computes the deviations, and then colors the Pointcloud accordingly.



Example of a Point Colormap using Method 1 (CAD model hidden)

This method allows you to create annotation points as well. For details on the annotation-related check boxes in the **Pointcloud Operator** dialog box, see the appropriate description in the "Surface Colormap" help topic starting with the "Create annotation points" check box description.

Method 2: Select any of the three check boxes in the **Display** area of the **Pointcloud Operator** dialog box.

When you select any or all of the **Display** check boxes, PC-DMIS projects the points onto the actual CAD model. The software computes the deviations, and then colors the Pointcloud accordingly. This process is more time-consuming and more accurate since the software projects the points onto the actual CAD model instead of on the tessellated CAD model. Because this operation is more time-consuming, it is best to first filter the pointcloud, or confine it to selected CAD faces.

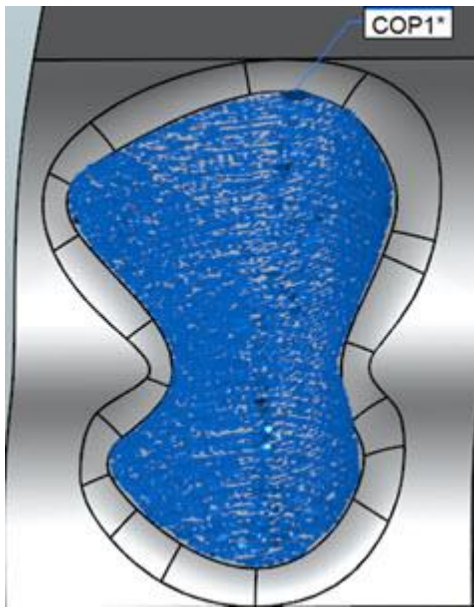
To apply the POINT COLORMAP operation to a pointcloud, click **Pointcloud Point**



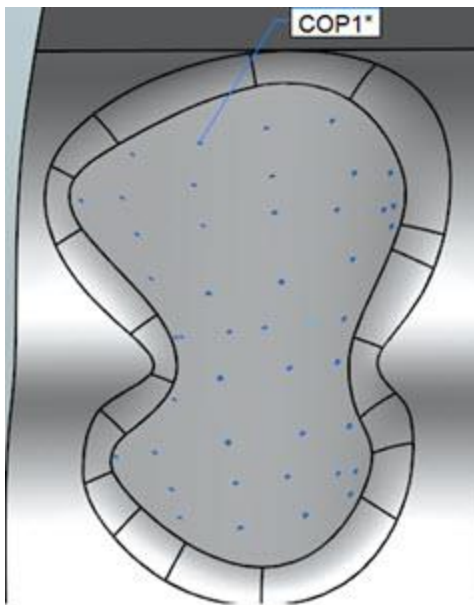
Colormap () on the **Pointcloud** toolbar, or select **Insert | Pointcloud | Point Colormap** from the menu.

The recommended process when creating a point colormap using dots, needles and/or text (Method 2) is:

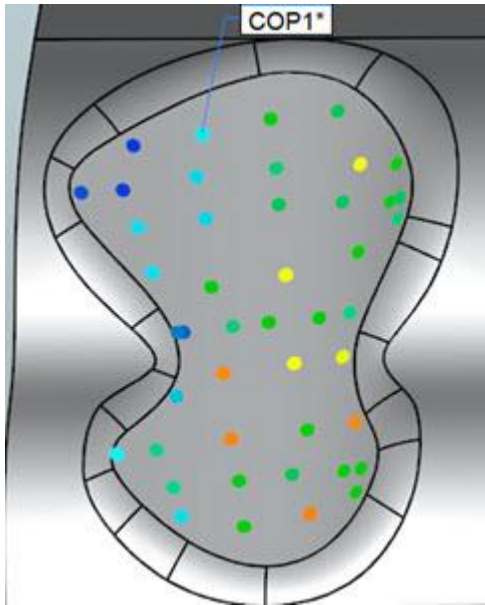
1. Clean the data or select just those surfaces where you want the point colormap.



2. Use the **DISTANCE** type setting from the **Filter** COP Operator to filter the data.



3. Create the point colormap.



Example of the recommended steps to apply a point colormap

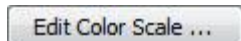
The Point Colormap operator has these properties:

Tolerances - Use this property to set the upper (plus) and lower (minus) tolerance values:

Plus - The upper tolerance value

Minus - The lower tolerance value

Use dimension color scale check box - When you select this check box, the software uses the **Dimensions Color Bar** to define the color bar for the Point Colormap color properties. For details on the **Dimensions Color Bar**, see "Using the Dimensions Colors Window (Dimensions Color Bar)" in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.



Edit Color Bar - If you do not select the **Use dimension color scale** check box, the software enables the **Edit Color Scale** button. When you click this button, the functionality to dynamically change the color, scale and threshold of the surface and point colormap properties becomes available through the **Color Scale Editor** dialog box. For details, see the "Edit the Color Scale" topic.

Dots - This option allows the use of colored dots.

Size - This option defines the size of the dots.

Needles - This option allows the use of scaled deviations (using the **Scale** value below) as a colored line segment normal to CAD.

Scale - This option defines the scale value PC-DMIS uses for the needle representation.

Text - This option defines the numerical value of the deviation.

Max distance - The software only includes points that fall within the **Max distance** value as part of the colormap. Note that if this value is too small, you may not see all the expected colored deviations. A good rule of thumb is to set this value slightly larger (10%, for example) than the largest deviation.

Thickness - This allows you to add a thickness value to deviations on the colormap. This is useful if you want to add a material thickness to a CAD surface model.

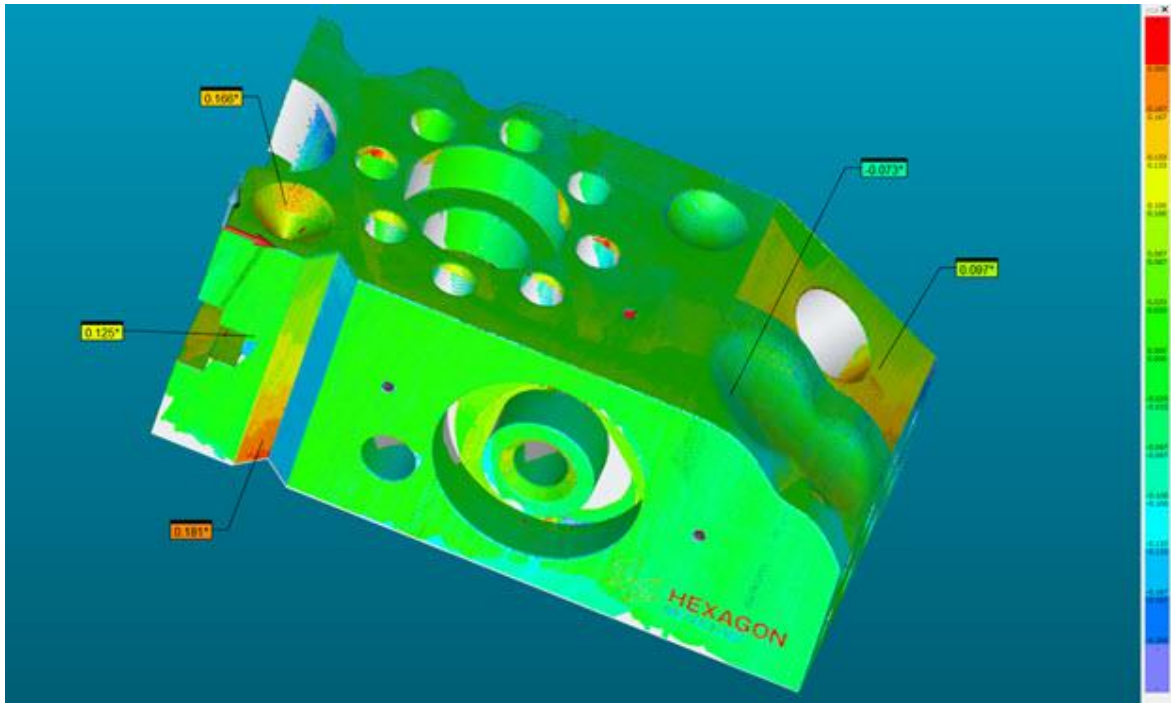


The following three annotation point check box options are only available if you DO NOT select the **Dots**, **Needles** and **Text** check boxes. Also, the CAD model must be visible in order to create annotation points.

Create annotation points check box - For details on this check box, see the "Create annotation points" description in the "Surface Colormap" help topic.

Create MinMax annotations check box - For details on this check box, see the "Create MinMax annotations" description in the "Surface Colormap" help topic.

Show annotation points check box - When you select this check box, the software displays all annotation points.



Example of Point Colormap with annotations

Click **Create** to insert a `COP/OPER, POINT COLORMAP` command into the Edit window.



For example:

```
COPPCOLMAP1=COP/OPER, POINT COLORMAP, PLUS
TOLERANCE=0.0394, MINUS TOLERANCE=-0.0394, THICKNESS=0,

SHOW DOTS=YES, DOT SIZE=0.0787, SHOW NEEDLES=YES, NEEDLE
SCALE=10, SHOW LABELS=YES,

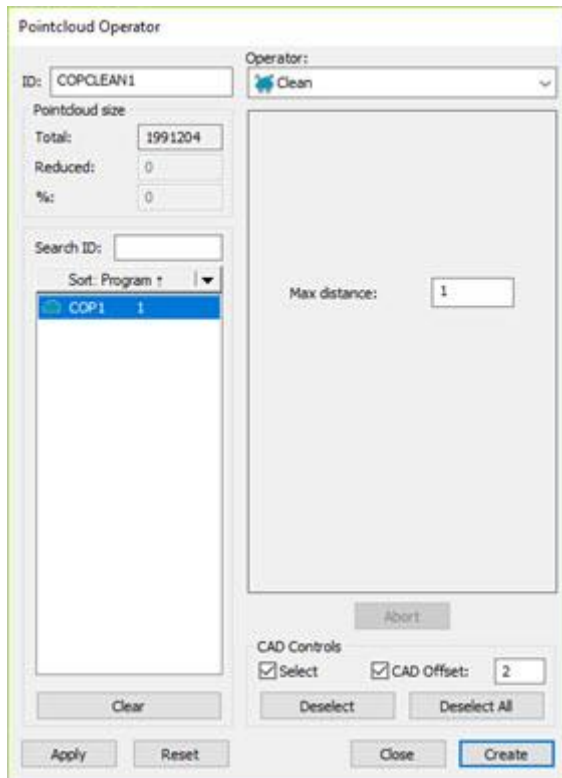
SIZE=50023

REF, COP2, ,
```

Colormaps in the Report


For information on how the software shows colormaps in the report, see "Colormaps and the CadReportObject" in the "Reporting Measurement Results" chapter of the PC-DMIS Core documentation.

CLEAN



Pointcloud Operator dialog box - Clean Operator

The Clean operation eliminates outliers by using the distance of the points to the CAD model of the part. If the distance of a point is greater than the value of **Max distance**, PC-DMIS considers the point an outlier or not belonging to the part. To use this operation, you must establish at least a rough alignment (see "Creating a Pointcloud/CAD Alignment").

To apply the Clean operation to a Pointcloud, click **Clean Pointcloud** () on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Clean**. This immediately cleans the pointcloud.

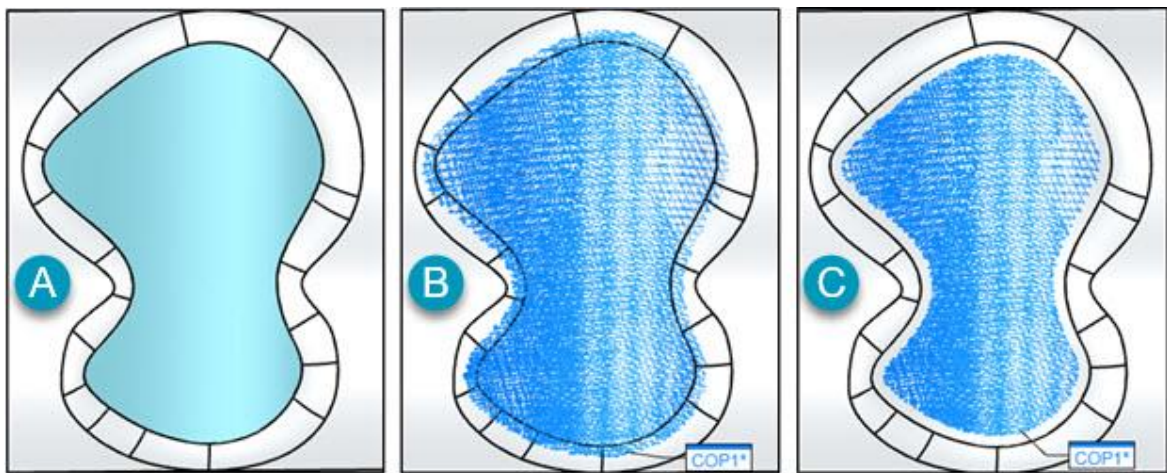
Select **Insert | Pointcloud | Operator** to open the **Pointcloud Operator** dialog box. Select **Clean** from the **Operator** list.

The dialog box for the **Clean** operator contains these options:

Max distance - Enter a value to indicate the maximum distance of a point to the CAD model that you want the software to consider the point an outlier.

CAD Controls - If you select the **Select** check box, you can click specific CAD surfaces in the Graphic Display window to apply the Clean operation. The software highlights the selected surfaces in red. The operation affects the entire cloud of points on the selected surfaces. PC-DMIS discards any point located at a distance greater than the specified **Max distance** from all selected surfaces. For example, suppose you select a single surface and enter a value of 10. This means that the software cleans any point in the COP located 10 or more units away from the selected surface. Any point in the COP within 10 units of the selected surface remain.

With the **Select** check box selected, the **CAD Offset** check box becomes available. Select the **CAD Offset** check box to enable the **CAD Offset** input box. Enter a value that PC-DMIS uses to "shrink away" from the CAD edges. This allows you to isolate points relative to specific CAD faces, and ignore points along the edge within this fixed offset distance.



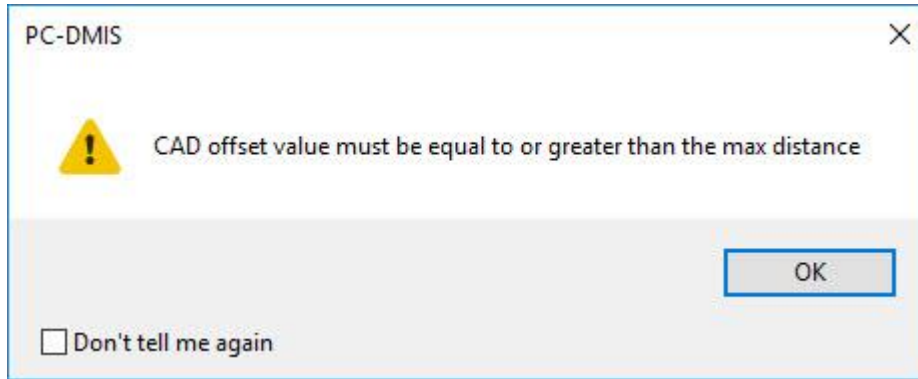
Example using the Clean operator with the Max Distance and CAD Offset options

A - CAD Surface selected in the Graphic Display window

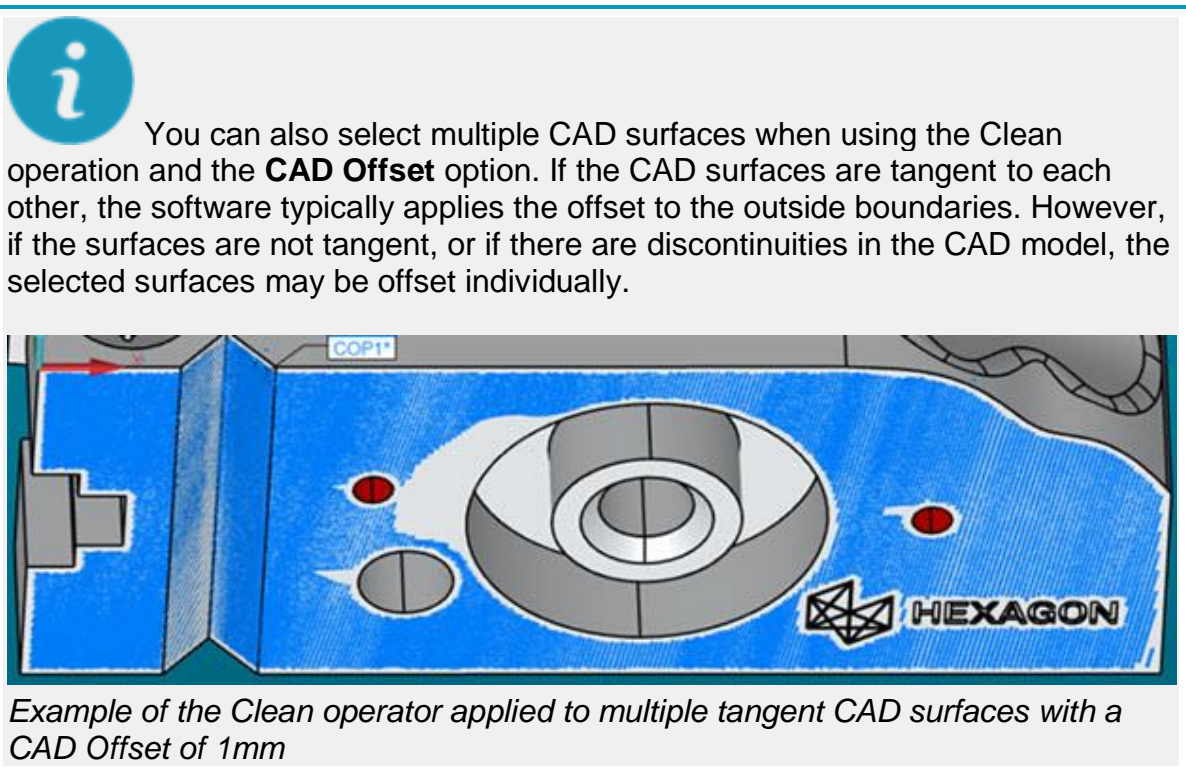
B - Clean operation applied with a Max Distance of 1mm

C - Clean operation applied with a Max Distance of 1mm, and a CAD Offset of 1mm

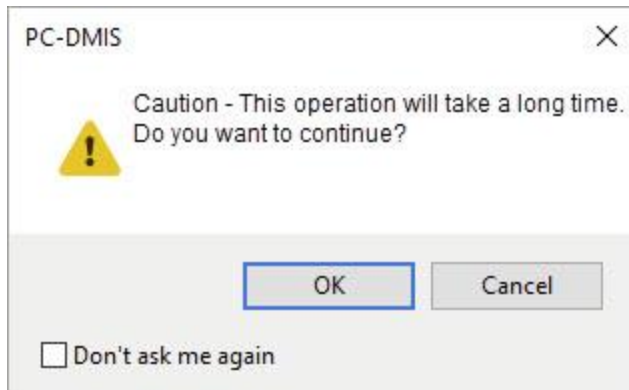
The **CAD Offset** value must be greater than or equal to the **Max distance** value. If the **CAD Offset** value is less than the **Max distance** value, PC-DMIS displays this message:



When you click **OK**, PC-DMIS resets the **CAD Offset** value to the current **Max distance** value.



With the COPCLEAN function, if you select a large number of CAD faces, and you marked the **Select** option and entered a **CAD Offset** value in the **CAD Controls** area of the **Pointcloud Operator** dialog box, PC-DMIS displays this message:

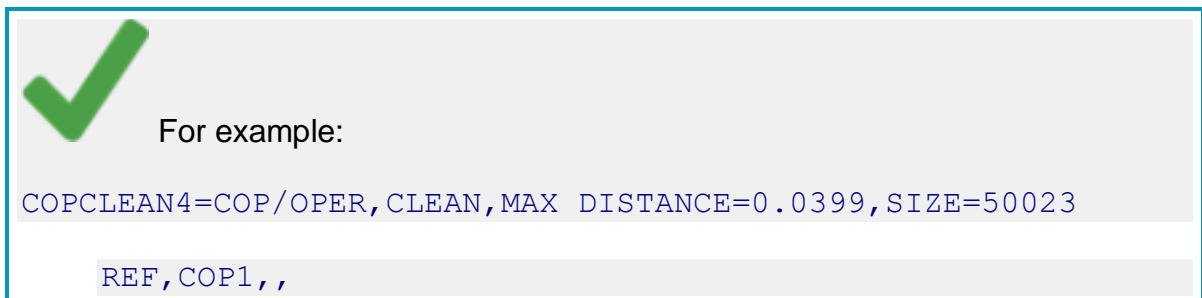


The message indicates that if you click **OK** to perform the CAD Offset operation with the current settings, it will take PC-DMIS a very long time to process due to the large number of CAD faces you picked.

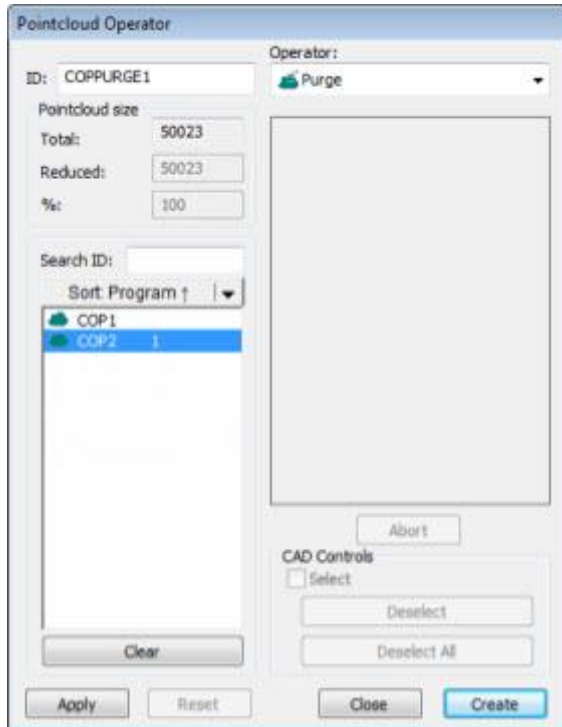
Select the **Don't ask me again** check box to prevent the message from appearing again.

Click **OK** to continue the operation, or click **Cancel** and the software will not execute the operation.

When you finish the updates to the dialog box, click **Create** to insert a `COP/OPER, CLEAN` command into the Edit window.



PURGE



Pointcloud Operator dialog box - PURGE Operator

From the COP command referenced by this operator, the PURGE operation removes all data points that do not belong to this operator. The PURGE operation is irreversible and affects all other operator commands that refer to the same COP container, so use the operation with caution.

To apply the PURGE operation to a pointcloud, click **Purge Pointcloud** () on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Purge**.

Clicking **Create** inserts a `COP/OPER, PURGE` command into the Edit window like the following examples:

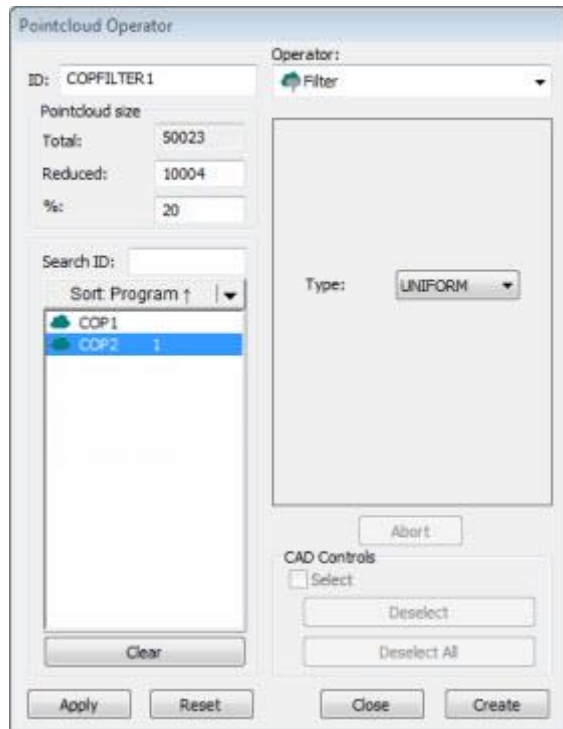
```
COPPURGE1=COP/OPER, PURGE, SIZE=0
```

```
REF, COPSECTION1, ,
```



Once you apply this command to a COP, there is no way to restore the COP data that was removed. You cannot click **Undo** to restore the data.

FILTER



Pointcloud Operator dialog box - FILTER Operator

The FILTER operation filters data to a smaller subset of points.

To apply the FILTER operation to a pointcloud, click **Filter Pointcloud** () on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Filter**.

The FILTER operator has these options:

Type - Indicates the type of filter operator to apply:

UNIFORM - This option generates a subset of points distributed evenly in the X, Y, and Z directions. It produces the same effect as a regular grid in 2D, but in this case the effect is a 3D grid.

CURVATURE - This option generates a subset of points with the highest estimated curvatures, mainly around edges, vertices, and highly curved areas of the surface.

RANDOM - This option generates a subset of points randomly distributed in the pointcloud.

DISTANCE - This option generates a subset of points where points are at least the specified **Distance** value apart from each other.

Distance - When you select **DISTANCE**, the value you enter in this box specifies the distance for the distance filter.

INCIDENCE ANGLE - This option generates a subset of points which excludes (filters out) points that have a normal vector orientation that fall outside the specified angle, relative to the laser sensor orientation. This filter allows you to remove laser points caused by secondary reflections or "noise". You can see the effect of this filter after you click the **Apply** button in the dialog box.

A valid value is any real number from 10 to 90, inclusive.

To use this filter, the pointcloud data must have vector information.

To filter COP data:

1. From the **Type** list, select a filter type.
2. From the list of commands, select the Pointcloud command to which you want to apply the filter.
3. Specify the number of points or percentage of points to keep after applying the filter in the **Reduced** or **%** boxes. This does not apply to the **Distance** filter.
4. Click the **Apply** button.

PC-DMIS filters the data, and the Graphic Display window shows the result. The size of the filtered data may differ slightly from the value that you specified. It is even more noticeable when you execute the measurement routine and the software collects the data from the scan commands. It is generally impossible to get the same number of points from a laser sensor that you repeatedly scan the same entity.

5. When the results are acceptable, click the **Create** button. PC-DMIS adds a **COPFILTER** command to the measurement routine that contains all the information related to the applied filter.

When you click **Create**, PC-DMIS inserts a **COP/OPER, FILTER** command into the Edit window similar to this:

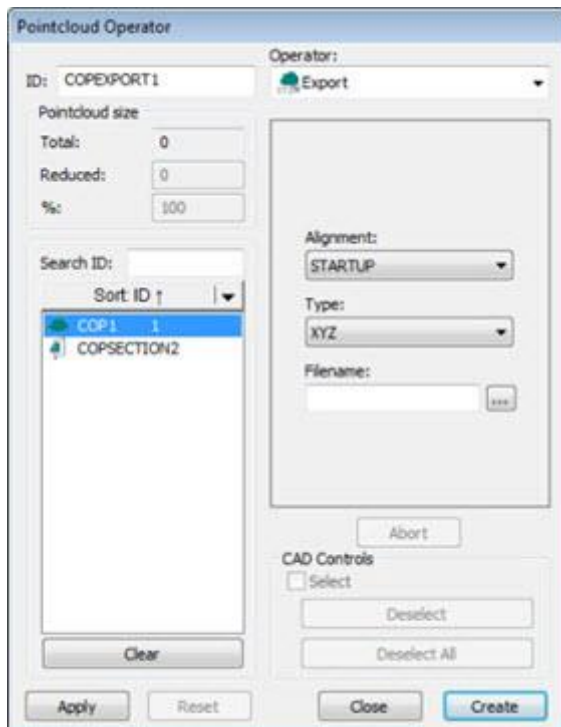
```
COPFILTER3=COP/OPER, FILTER, UNIFORM, SIZE=3000  
REF, COP1, ,
```

In the example above, if the initial size of COP1 was 10,000 points, the filter replaces the 10,000 points held in COP1 with the filtered 3,000 points. In this case, COP1 now holds the filtered 3,000 points for its cloud of points. PC-DMIS flags the 7000 points it didn't use, so that you can undo the filter operation with the RESET operation. You can permanently delete the 7000 points PC-DMIS did not use with the PURGE operation. For more information, see "RESET" and "PURGE".

Pointcloud EXPORT






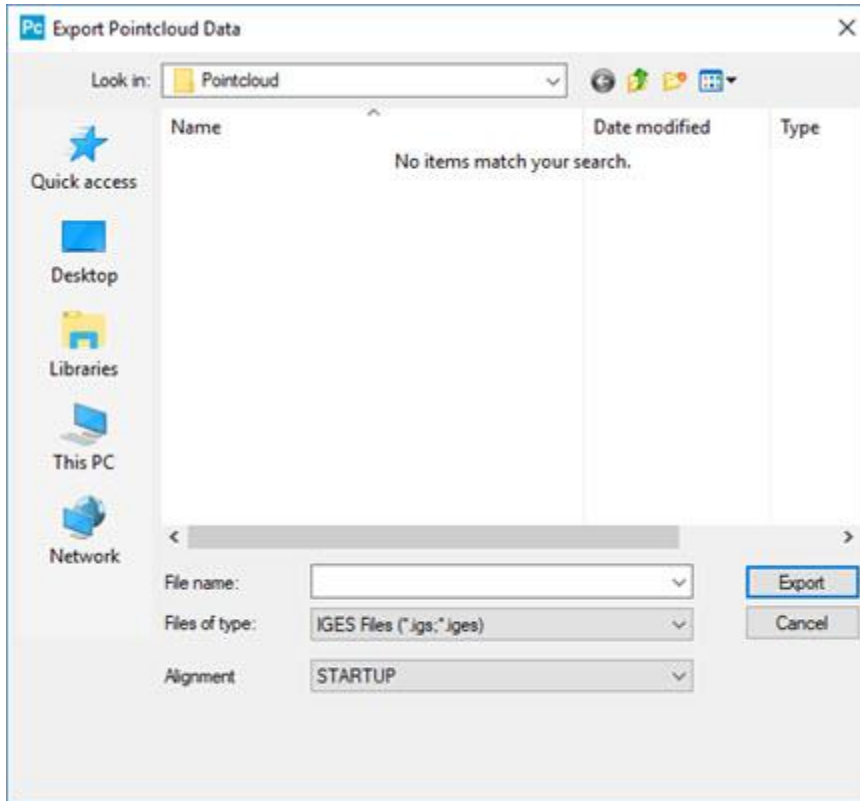
If you choose the EXPORT COPOPER with a cross section using IGES type, PC-DMIS exports the selected cross sections as a B-spline curve using IGES Type 126.



Pointcloud Operator dialog box - Pointcloud EXPORT Operator

The Pointcloud EXPORT operation exports the data in a COP or operator command in a specified format to an external file. The dialog box of this operation is similar to the Pointcloud IMPORT operator.

To apply the **Pointcloud EXPORT** operation to a pointcloud, click **XYZ** () , **IGS** () , or **PSL** () on the **Pointcloud** toolbar, or select a menu option **File | Export | Pointcloud**. The software opens the **Export Pointcloud Data** dialog box.



Export Pointcloud Data dialog box

The **Pointcloud EXPORT** operator uses these options:

File name - This option indicates the name of the export file.

Files of type - This option indicates the data format for the export operation. It can be **XYZ**, **IGES**, or **PSL** (Polyworks).



For exporting XYZ file types, you can define the separator character to use. For details, see "ExportXYZSeparator" in the "PointcloudOperator" section of the PC-DMIS Settings Editor documentation.

Alignment - This option indicates the type of alignment to include when you export the data.

Click **Create** to insert a `COP/OPER,EXPORT` command into the **Edit** window.



For example:

```
COPEXPORT1=COP/OPER,EXPORT,FORMAT=IGES,FILENAME=D:/Dataout.IGS,SIZE=1623201
```

```
REF,COP1,,
```

Specify the format in `FORMAT` and the output file name and path in `FILENAME`, and then reference the COP command holding the data. If you applied a filter to the COP command, then you should reference the `COPFILTER` command for export rather than the original COP command.

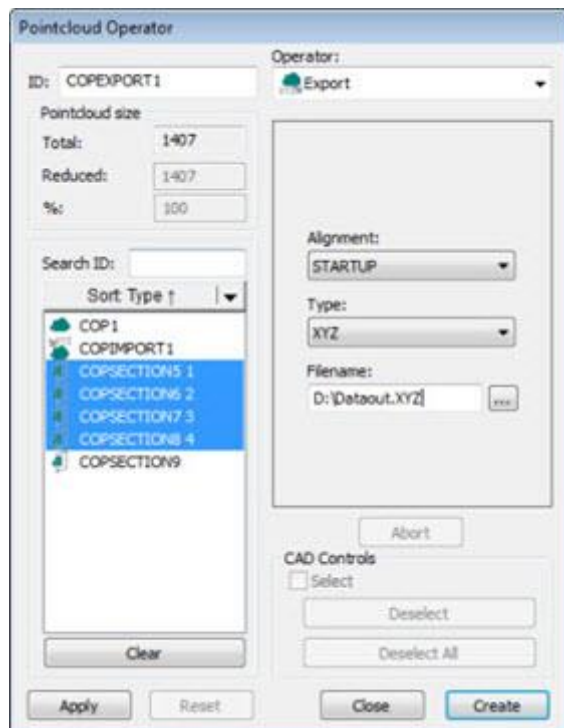


For example, `REF, COPFILTER1,` rather than `REF, COP1,`. This ensures that the exported file reflects the filter set.

```
COPEXPORT2=COP/OPER,EXPORT,FORMAT=IGES,FILENAME=D:/Dataout.IGS,SIZE=0
```

```
REF,COPFILTER1,,
```

It is also possible to select more than one command in the list of commands to export them in a single operation:



Pointcloud Operator dialog box with multiple commands selected

In this case, PC-DMIS inserts the command into the **Edit** window.

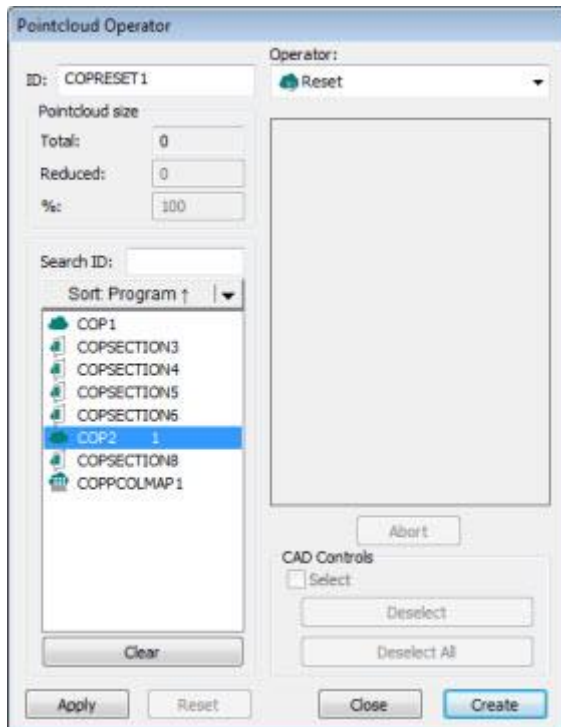


For example:

```
COPEXPORT1=COP/OPER,EXPORT,FORMAT=XYZ,FILENAME=D:/Dataout.XYZ,SIZE=1246
```

```
REF,COPECTION1,COPECTION2,COPECTION3,COPECTION4,,
```

RESET



Pointcloud Operator dialog box - Reset Operator

The RESET operation has a behavior similar to Undo. It resets the data referred to in a previous operator command so that the new operator command represents all the data of the referred COP command and not only a subset.

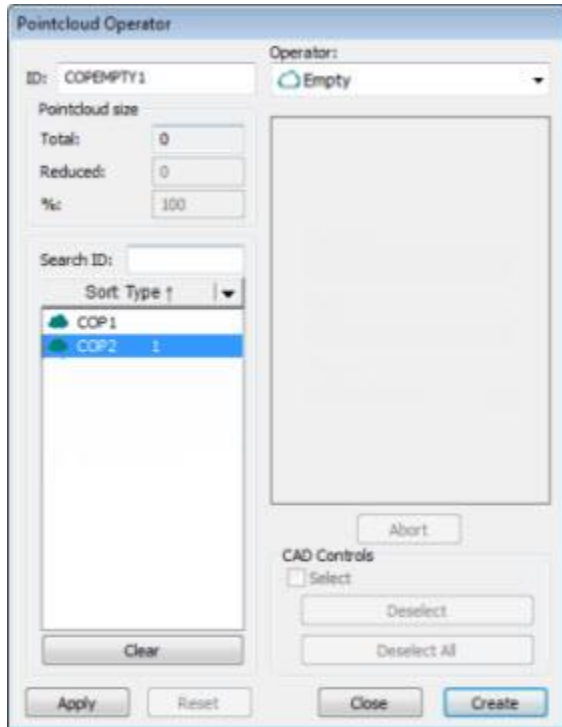
To apply the RESET operation, click **Reset Pointcloud** () on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Reset**.

Clicking **Create** inserts a `COP/OPER, RESET` command into the Edit window like the following examples:

```
COPRESET7=COP/OPER, RESET, SIZE=0
```

```
REF, COPFILTER 2, ,
```


EMPTY



Pointcloud Operator dialog box - EMPTY Operator

This operation deletes all of the data contained in a selected COP or operator command. When you execute this command, PC-DMIS removes the data of the associated COP.

To apply the EMPTY Pointcloud operation to a pointcloud:

1. If you have more than one pointcloud defined, position your cursor at the location of the pointcloud you want to empty. If you only have one pointcloud defined, place the cursor on or above it.
2. Click **Empty Pointcloud** () on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Empty**.
3. Click **Create** to insert a `COP/OPER, EMPTY` command into the Edit window.



For example:

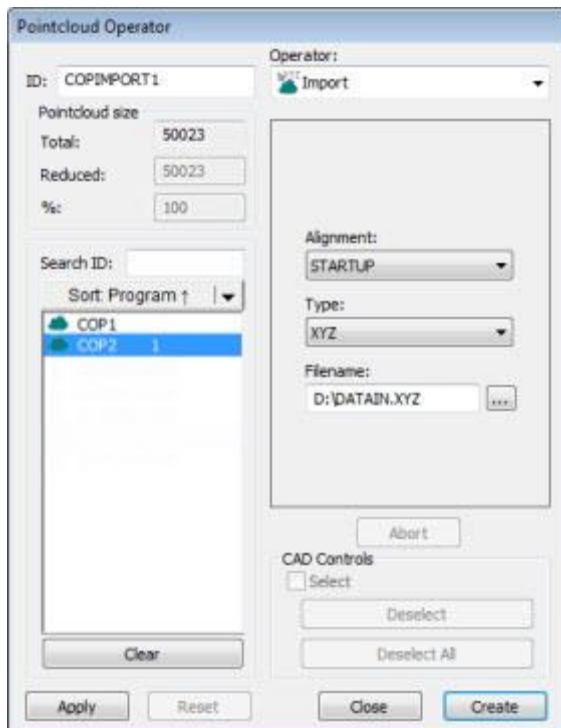
```
COPEMPTY2 =COP/OPER, EMPTY, SIZE=0
```

```
REF, COP2, ,
```






Once you apply this command to a COP, there is no way to restore the COP data. You cannot click **Undo** to restore the data.

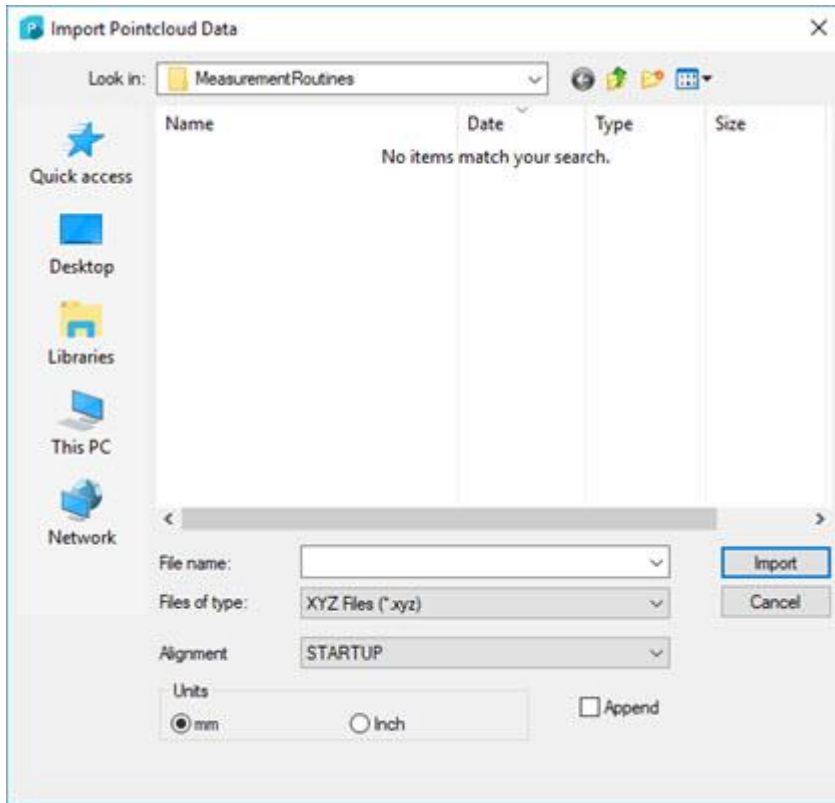
Pointcloud IMPORT



Pointcloud Operator dialog box - Pointcloud IMPORT Operator

The **Pointcloud IMPORT** operation imports data from an external file into a COP command in the specified format. The dialog box for this operation is similar to the dialog box for the Pointcloud EXPORT operation.

To apply the **Pointcloud IMPORT** operation to a pointcloud, click **XYZ** () , **PSL** () , or **STL** () on the **Pointcloud** toolbar, or select a menu option on the **File | Import | Pointcloud** menu. The software opens the **Import Pointcloud Data** dialog box.



Import Pointcloud Data dialog box

Navigate to the pointcloud data file, and click **Import**.

1. From the Edit window, select the COP that you want to add the new data to.
2. Click the **Import** option from the menu or toolbar as described above.
3. From the **Import Pointcloud Data** dialog box, mark the **Append** check box if you want to add the new COP data onto existing COP data.
4. Click **Import**.

The **Pointcloud IMPORT** operator uses these options:

Alignment - This option indicates the type of alignment to include when importing.

Type - This option indicates the type of format from which data is imported. It can be **XYZ**, **PSL** (Polyworks), or **STL** type.

Filename - This option indicates the name of the import file.

Units - Use this option to select the units of the imported COP data.

Append - Select this check box if you want to append the imported data into the existing COP. If you do not select this option, then PC-DMIS empties the first COP

it finds after the current cursor position in the **Edit** window, and replaces it with the imported COP data.

Click **Create** to insert a `COP/OPER, IMPORT` command into the **Edit** window.

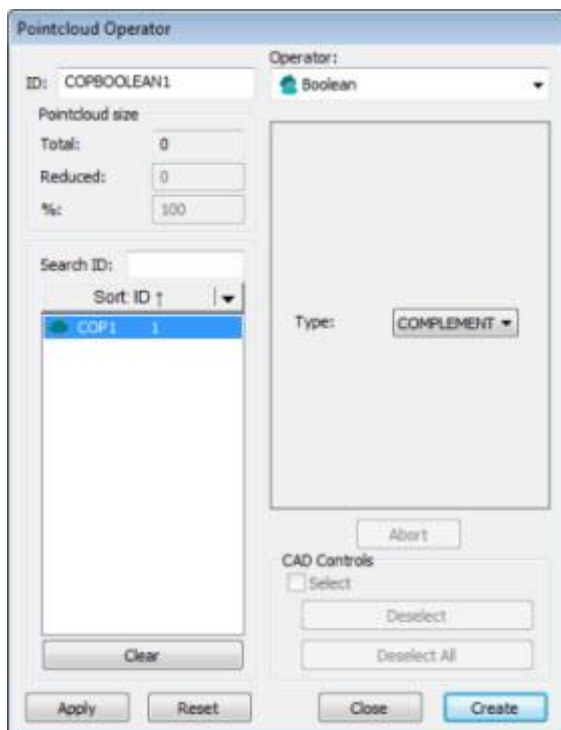


For example:

```
COPIMPORT1=COP/OPER, IMPORT, FORMAT=XYZ,
FILENAME=D:/DATAIN.XYZ, SIZE=0
```

```
REF, COP1,
```


BOOLEAN



Pointcloud Operator dialog box - BOOLEAN Operator

The software applies this operation on one or two operators or COP commands you select.

To apply the BOOLEAN operation to a pointcloud, click **Pointcloud Boolean**

Operation () on the **Pointcloud** toolbar.

The BOOLEAN operator uses the following option:

Type - This option indicates the type of Boolean operator to apply:


COMPLEMENT - This type generates the points that are not visible in a single-selected command.

UNITE - When applied to the two selected commands, this type generates a set of data points that contains all of the points in those commands.

INTERSECT - This type generates the set of data points that have the same locations in two selected commands.

DIFFERENCE - This type removes, from the first selected command, all of the points that are in common with the second selected command.

Click **Create** after you edit the command to insert a `COP/OPER, BOOLEAN` command into the Edit window.



For example:

```
COPBOOLEAN1=COP/OPER, BOOLEAN, UNITE, SIZE=0  
REF, COOPER2, COOPER3, ,
```

Gages

PC-DMIS gages are quick-check tools designed to measure lengths along an axis or a direction (Caliper), or a radius on a pointcloud cross section (2D Radius Gage).

PC-DMIS also provides a Temperature Gage and a Thickness Gage. This section describes only the gages that you can use with pointclouds and meshes.

For details on the Temperature Gage, see "Temperature Gage" in the PC-DMIS Core documentation.

For details on the Thickness Gage, see "Thickness Gage" in the PC-DMIS Core documentation.

Caliper Overview




This option is only available if your PC-DMIS license includes the Small COP or Big COP option.

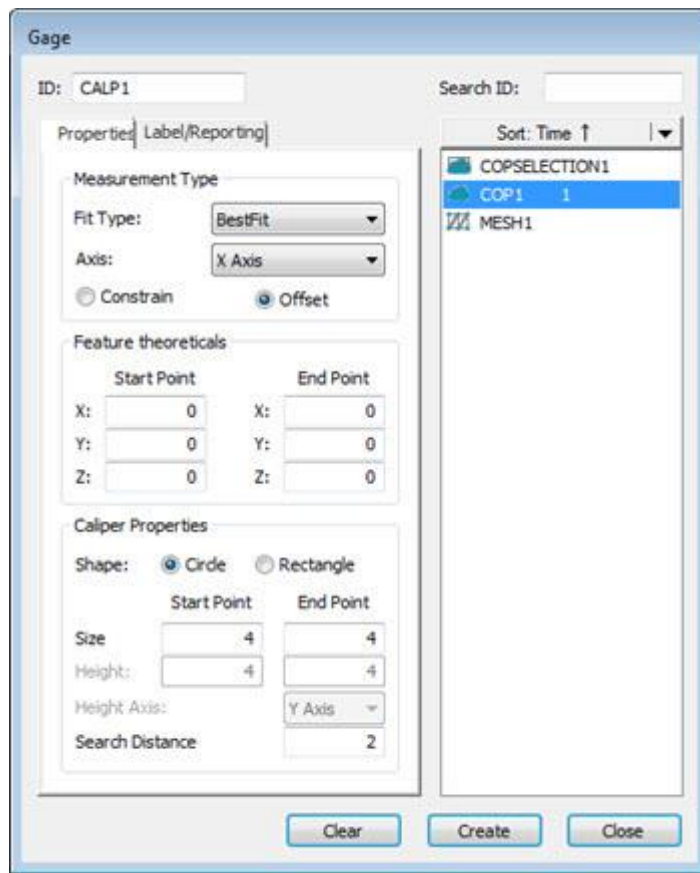
The Caliper is a quick-check tool that works similar to a physical caliper. It provides a local two-point size check on the Pointcloud (COP), Mesh, or COOPER (such as the COPSELECT, COPCLEAN, or COPFILTER) object. The Caliper shows the measured length along the selected axis or direction.

Select the **Caliper** option from the **Insert | Gage** menu.



You can also access the **Gage** dialog box in these ways:

- Click the **Caliper** button () from the **QuickCloud** toolbar.
- From the **QuickMeasure** toolbar, click the **Gage** drop-down arrow, and then click the **Caliper** button.



Gage dialog box

A caliper has two tips, which are used to measure the distance between two opposing sides. The caliper's tip has a user-defined size. Click in the Graphic Display window to select the start and end points. Using the data within the tip size, the caliper end points stop at the high points on the selected data (or optionally, on the calculated best fit points). The software performs a search distance along the caliper axis to determine the relevant points.

The **Gage** dialog box has these tabs:

Gage dialog box - Properties tab

Properties | Label/Reporting

Measurement Type

Fit Type: BestFit

Axis: Y Axis

☐ Constrain ☒ Offset

Feature theoreticals

Start Point		End Point	
X:	0	X:	0
Y:	0	Y:	0
Z:	0	Z:	0

Caliper Properties

Shape: ☒ Circle ☐ Rectangle

	Start Point	End Point
Size	2	6
Height	4	4
Height Axis:		X Axis
Search Distance		2

Gage dialog box - Properties tab

The **Properties** tab of the **Gage** dialog box has these sections:

Measurement Type

Fit Type: Click the drop-down arrow to display these options:

MaxFit: This setting is the default. Using the tip size and search distance, the caliper end points stop at the high points on the selected surfaces. A search distance along the caliper axis is used to determine the relevant points.

BestFit: A best fit least square fit is applied to all data points that fall within the caliper tip size and search distance. The resulting best fit points are used to determine the caliper length. This alternative method may be used if the scan data contains "noise", but can result in the caliper shown inside the pointcloud or mesh.

Axis: The Caliper can be constructed along the X, Y, or Z axis. Select **Parallel** to construct it Normal to the first surface picked. Select **None** to apply no constraint (3D distance between two points).

Constrain: Select this option to make the two end points exactly opposite one another along the selected axis.

Offset: Select this option to allow the two end points to be offset from each other in position. The measured length remains along the selected axis.

Feature theoreticals

Start Point: This option is the XYZ coordinate location of where the Caliper begins.

End Point: This option is the XYZ coordinate location of where the Caliper stops.

Caliper Properties

Shape: Select the appropriate tip shape, **Circle** (default) or **Rectangle**. If you select **Rectangle**, the **Height** and **Height Axis** options are enabled.



The **Rectangle** option is only enabled when you select the **X Axis**, **Y Axis**, or **Z Axis** option from the **Measurement Type** section. If you select **Parallel** or **None**, the **Rectangle** option is disabled.

Size / Width: The Caliper can have different size start and end tips. Enter the **Size Start Point** and **End Point** values for the circle tip, or the **Width Start Point** and **End Point** values for a rectangle tip. When the distance is then computed, the tip stops at the high point the way a caliper would.

Height: These values define the height of a rectangular tip's **Start Point** and **End Point**. The height size runs along the selected axis. This option is only enabled for rectangular calipers.

Height Axis: Select the option from the list to set the axis used to control the rotation of the rectangle. This option is only enabled for rectangular calipers.

Search Distance: This value defines the length, from nominal, on either side of the picked point. The search distance along with the caliper tip shape creates a cylindrical zone. All data within this zone is evaluated to determine the caliper high point.

For details, see the "Creating a Caliper" topic.

Gage dialog box - Label/Reporting tab

Gage dialog box - Label/Reporting tab

The **Label/Reporting** tab of the **Gage** dialog box has these sections:

Tolerances section



The default Caliper tolerances are defined by the Dimension Color scale. For details, see "Editing Dimension Colors" in the PC-DMIS Core documentation.

The **Tolerances** section allows you to type plus and minus tolerances for the Caliper length.

To enter the plus, minus, and nominal tolerances:

1. Type the plus tolerance value in the **Plus** box.
2. Type the minus tolerance value in the **Minus** box.

If a CAD model is used, the nominal (theoretical) caliper length is determined from the CAD. If no CAD model is used, the nominal value is updated with the initial measured value. The Nominal value can be edited.

Label section

Show check box: When this check box is selected, the Caliper label and graphic are displayed in the Graphic Display window.

Show heading check box: Toggles the display of the row and column headings in the Caliper label. When this check box is selected, the label's row and column headings are displayed.

Contents area



The order in which you select the following check boxes defines the order in which they appear in the label. The order number appears to the left of each selected item. When you clear a check box, the software reorders the order numbers of the remaining selected check boxes accordingly.

Measured check box: When this check box is selected, the measured data is displayed in the label.

Nominal check box: When this check box is selected, the nominal data is displayed in the label.

Tolerance check box: When this check box is selected, the tolerance data appears in the label.

Deviation check box: When this check box is selected, the deviation data between the measured and nominal values appears in the label.

OutTol check box: When this check box is selected, the out-of-tolerance data appears in the label.

Default button: Click to set the current selection of check boxes as the default.

Reset button: Click to clear all check boxes in the **Contents** area. The software then resets the section to the auto-setting configuration showing the measured value.

Report and Statistics section

From this section, you can use the options to control the output results:

STATS - sends output to statistical files

REPORT - sends output to inspection report

BOTH - sends output to both inspection report and statistical files

NONE - doesn't send output anywhere

When PC-DMIS executes the command, the results are sent to the specified output.

If you choose Stats or Both, a preceding STATS/ON command must exist inside the Edit window to send the results to the stats file.

The items that appear in the text format output are defined by the dimension format command in your measurement routine. For details, see the topic "Dimension Format" in the PC-DMIS Core documentation.

Clear button: Click to reset the **Gage** dialog box to the auto settings configuration.

Create button: Click to create a new Caliper defined with the settings you made in the **Gage** dialog box. The software creates the Caliper.

Close button: Click to close the **Gage** dialog box without creating a Caliper.

Caliper line thickness



You can set the thickness of the Caliper line through the **OpenGL** tab of the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | OpenGL**). For details, see the topic "Changing OpenGL Options" in the "Setting Your Preferences" chapter of the PC-DMIS Core documentation.

Creating a Caliper

To create a Caliper feature:

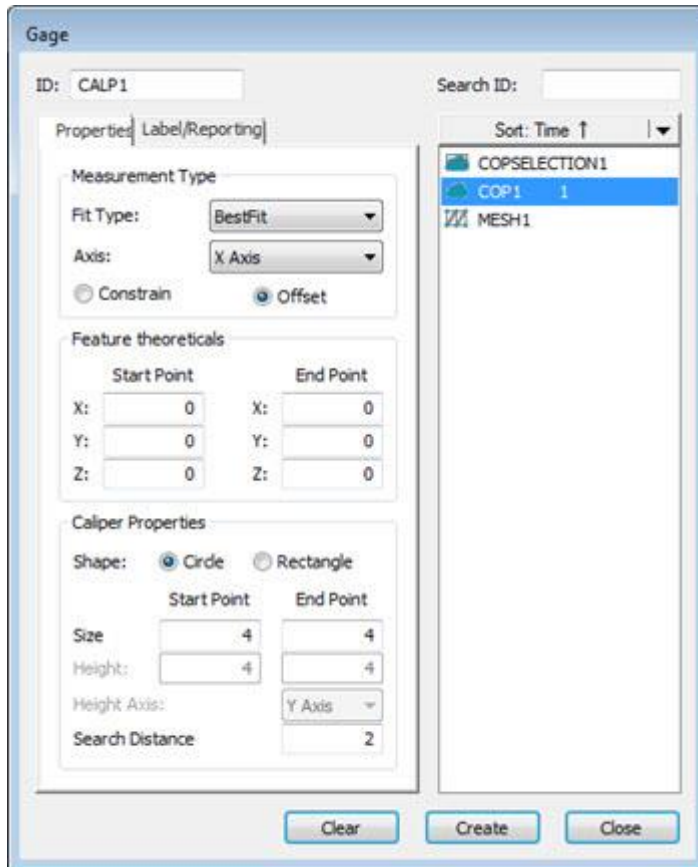
1. Select the **Caliper** option from the **Insert | Gage** menu. The **Gage** dialog box opens.



You can also access the **Gage** dialog box in these ways:

- Click the **Caliper** button () from the **QuickCloud** toolbar.

- From the **QuickMeasure** toolbar, click the **Gage** drop-down arrow, and then click the **Caliper** button.

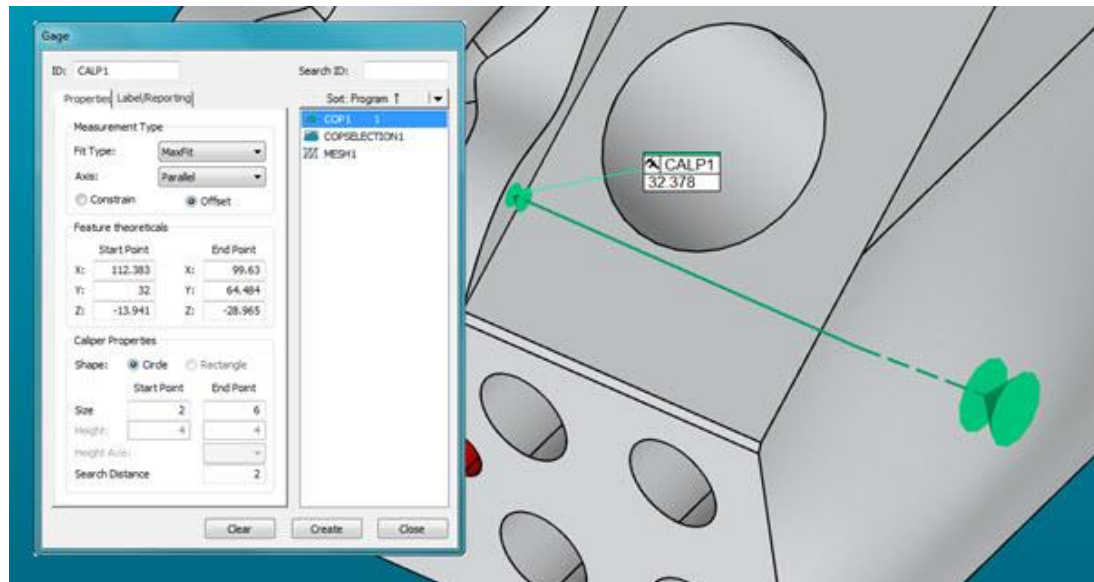


Gage dialog box

2. Select the COP, COPOPER, or Mesh data object to use.
3. In the **Measurement Type** area, select a type in the **Fit Type** list.
4. Select an axis in the **Axis** list, and then select the **Constrain** or **Offset** option.
5. In the **Caliper Properties** area, select the **Circle** or **Rectangle** shape option.
6. Edit the current value, or select the appropriate values for the following options:

Circle-shaped caliper tip options

- Size:** The default value is 4 mm for the **Start Point** and **End Point** options. You can set the start and end points of the caliper to different sizes depending on the CAD surfaces.



Example of a caliper created with different-sized start and end points



For non-planar surfaces, you should set the size to a larger value, such as 8-10 mm, to capture the high point. For planar surfaces, you can set it to a smaller value, such as 2 mm.

- **Search Distance:** The default value is 2 mm. This value defines the length, from nominal, on either side of the picked point. The search distance along with the caliper tip shape creates a cylindrical zone. All data within this zone is evaluated to determine the caliper high point.

Rectangle shaped caliper tip options

- **Width:** The default value is 4 mm for the **Start Point** and **End Point** options. The value sets the width of the caliper tip's start and end points.
- **Height:** The default value is 4 mm for both the **Start Point** and **End Point**. The value sets the height of the caliper tip's start and end points.



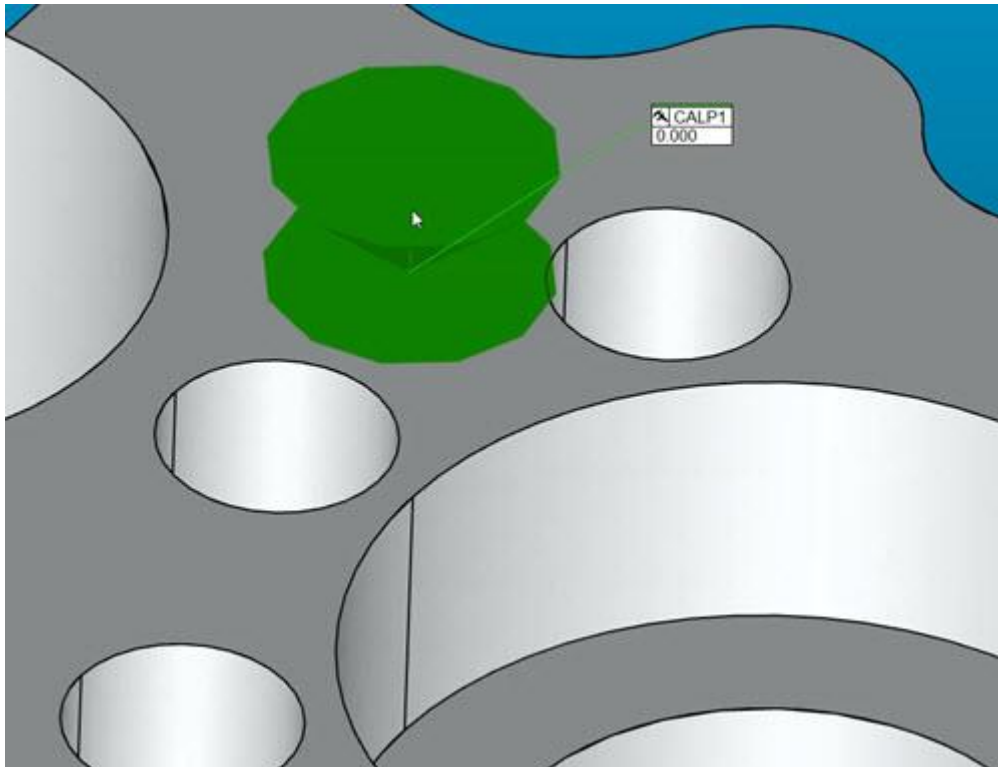
For non-planar surfaces, you should set the width and height to a larger value, such as 8-10 mm, to capture the high point. For planar surfaces, you can set the width and height to a smaller value, such as 2 mm.

- **Height Axis:** The default value depends on the **Axis** option you select in the **Measurement Type** area. From the list, select the option to define the axis that controls the rotation of the rectangle.
- **Search Distance:** See the description in the **Circle-shaped caliper tip options** section.



Changes to any of the **Gage** dialog box properties become the default values the next time the dialog box opens.

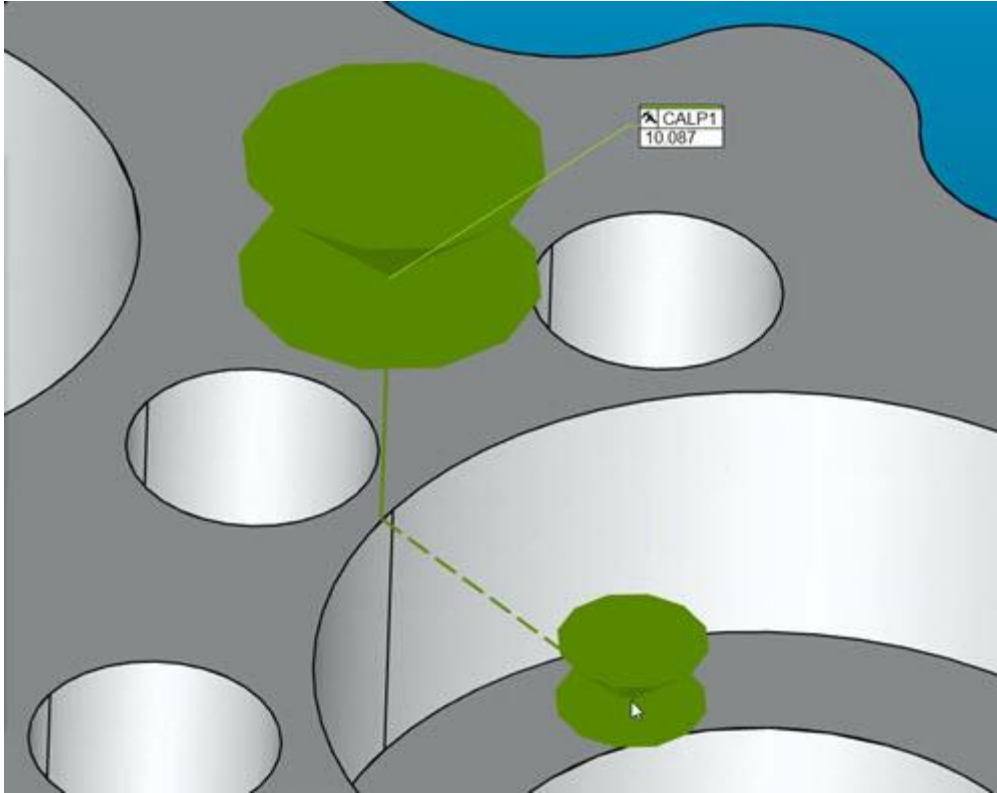
7. From the Graphic Display window, click to define the start point. To remove the first selected point, press the Delete key.



8. Move your cursor to the second location, and then click to define the end point. As you move the cursor, the length value updates in the Graphic Display window. If the selected object (COP or Mesh) contains data, the length that is shown is the measured value. If the selected object is empty, and a CAD model is used, the length value that is shown is the nominal value.



You can also enter the XYZ values for each in the **Start Point** and **End Point** XYZ boxes.

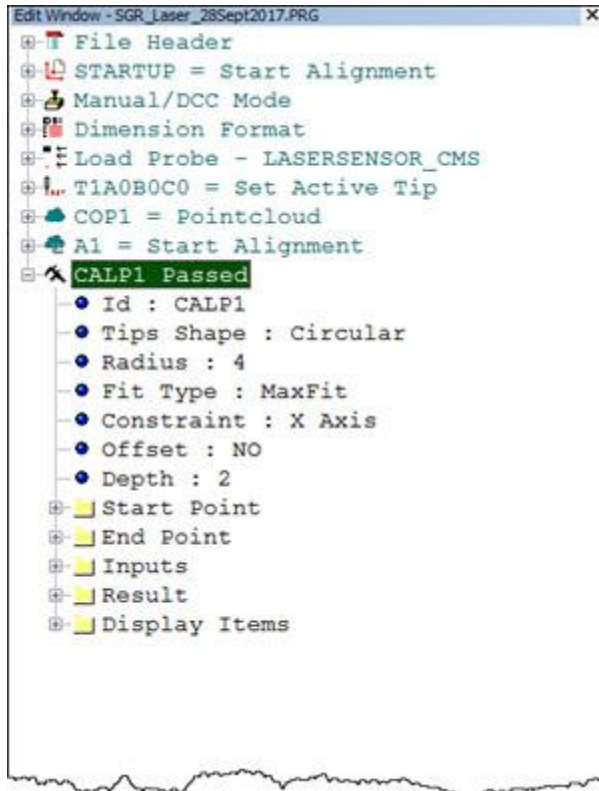


Caliper line thickness



You can set the thickness of the Caliper line through the **OpenGL** tab of the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | OpenGL**). For details, see the topic "Changing OpenGL Options" in the "Setting Your Preferences" chapter of the PC-DMIS Core documentation.

9. Click **Create** to define the caliper and add it to the commands in the Edit window.



Caliper Start Point, Mid Point, and End Point

The software extracts the nominal and measured start and end points of the Caliper gage when:

- You create the Caliper
- You execute the Caliper in the measurement routine

The software uses the start and end points to calculate the mid point. The mid point is then projected to the selected axis.

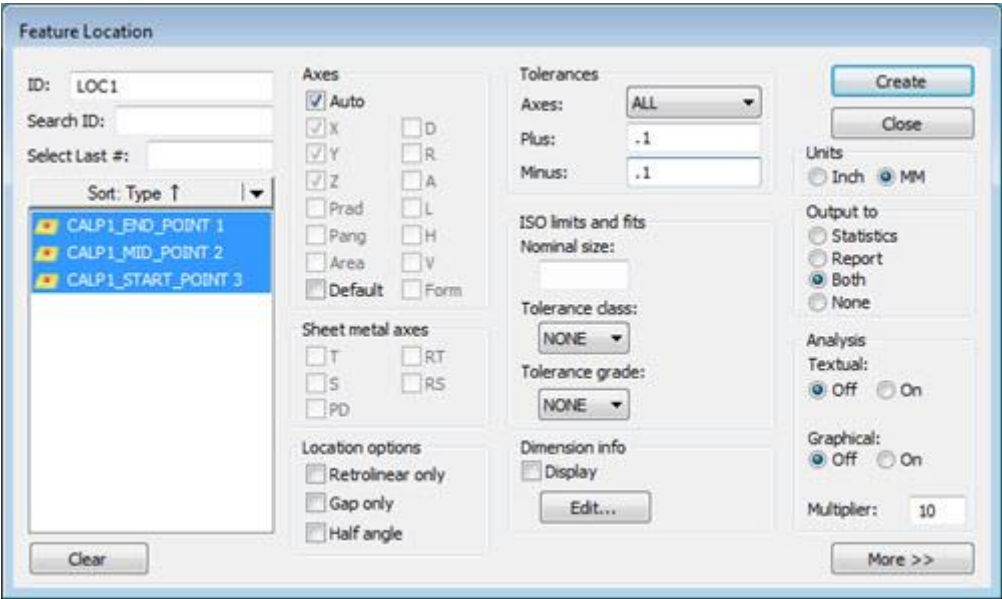
These points are not individual features in the Edit window. They are internal components of the Caliper gage.

The start point, mid point, and end point automatically appear as constructed offset points in the **Dimension**, **Construction**, and **Alignment** dialog boxes. You can dimension the points and use them in a best fit alignment, such as when you align a cast part that has excess material.

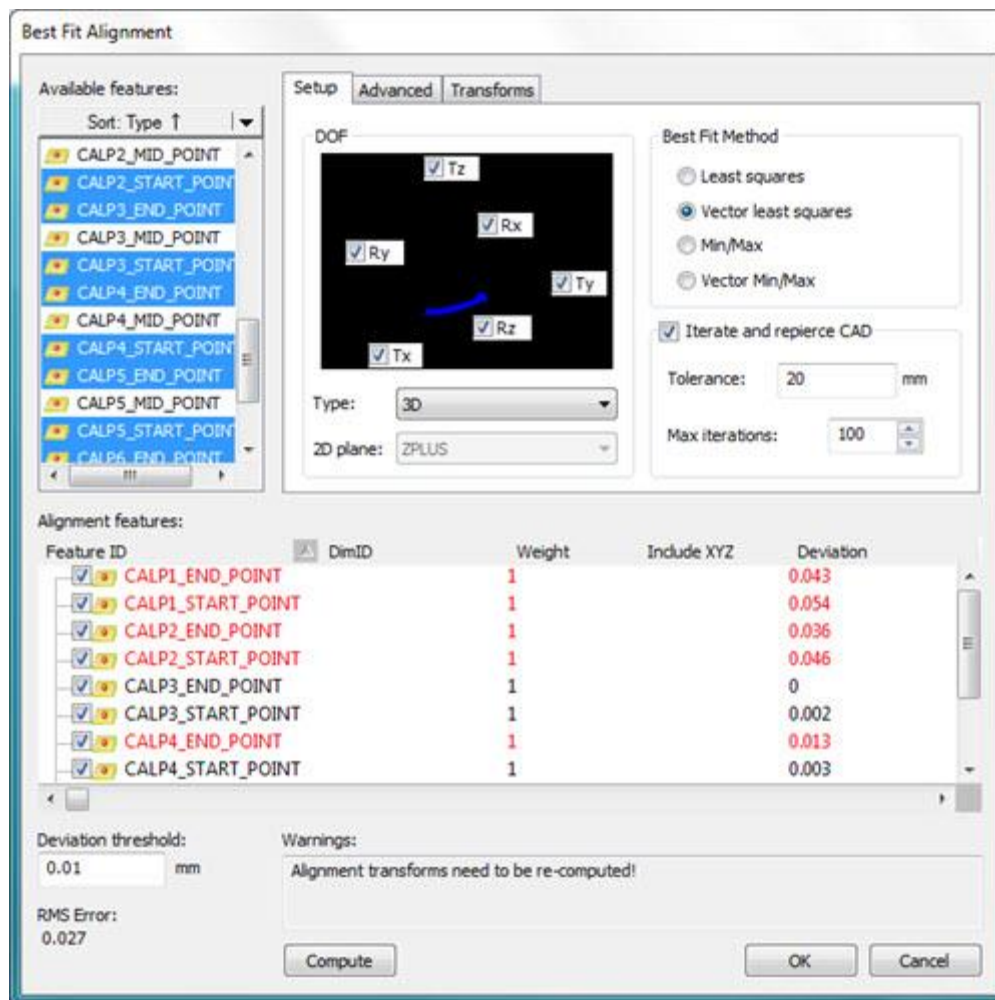
The following examples show various uses of the Caliper start, mid, and end points when you create features and alignments:



Example of start, mid, and end point options when creating a constructed feature

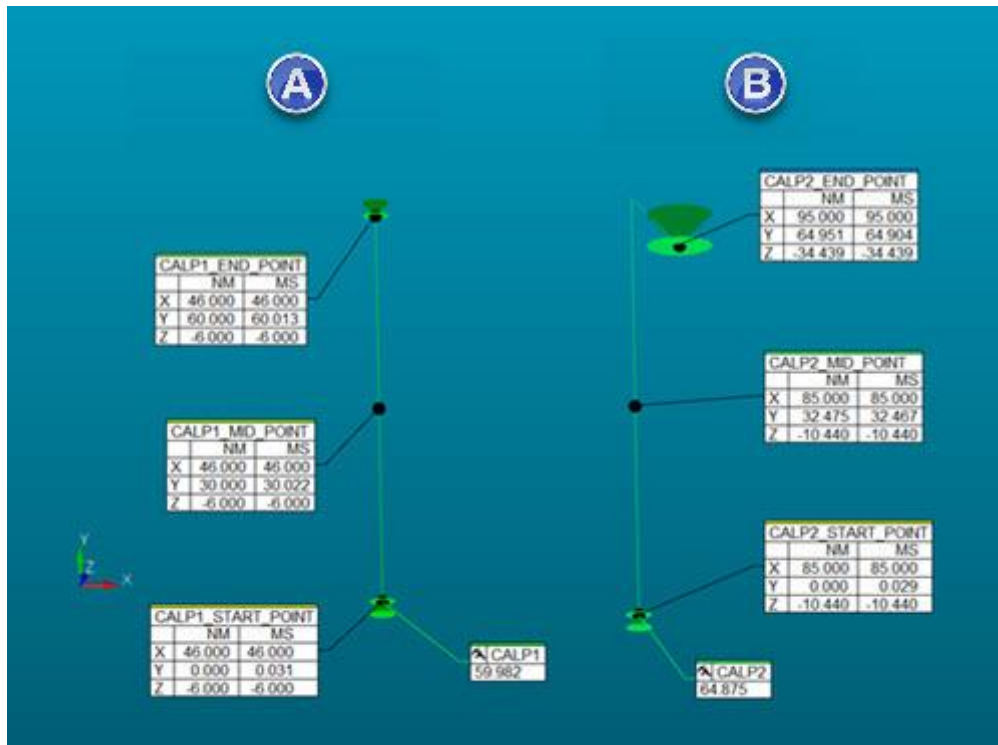


Example of start, mid, and end point options when creating a Feature Location Dimension



Example of start, mid, and end point options when creating an alignment

This example shows the use of the Constrain and Offset methods when you define a Caliper feature:



Examples of the Caliper points using the Constrained (left) and Offset methods (right)

(A) - Caliper1 end points constrained to Y axis


(B) - Caliper2 end points offset to Y axis

2D Radius Gage Overview

The 2D Radius Gage function is a quick-check tool that you can use to measure the radii on a pointcloud or mesh cross section.

You can create a 2D Radius Gage graphically on a cross section in the 2D Slide Show view.

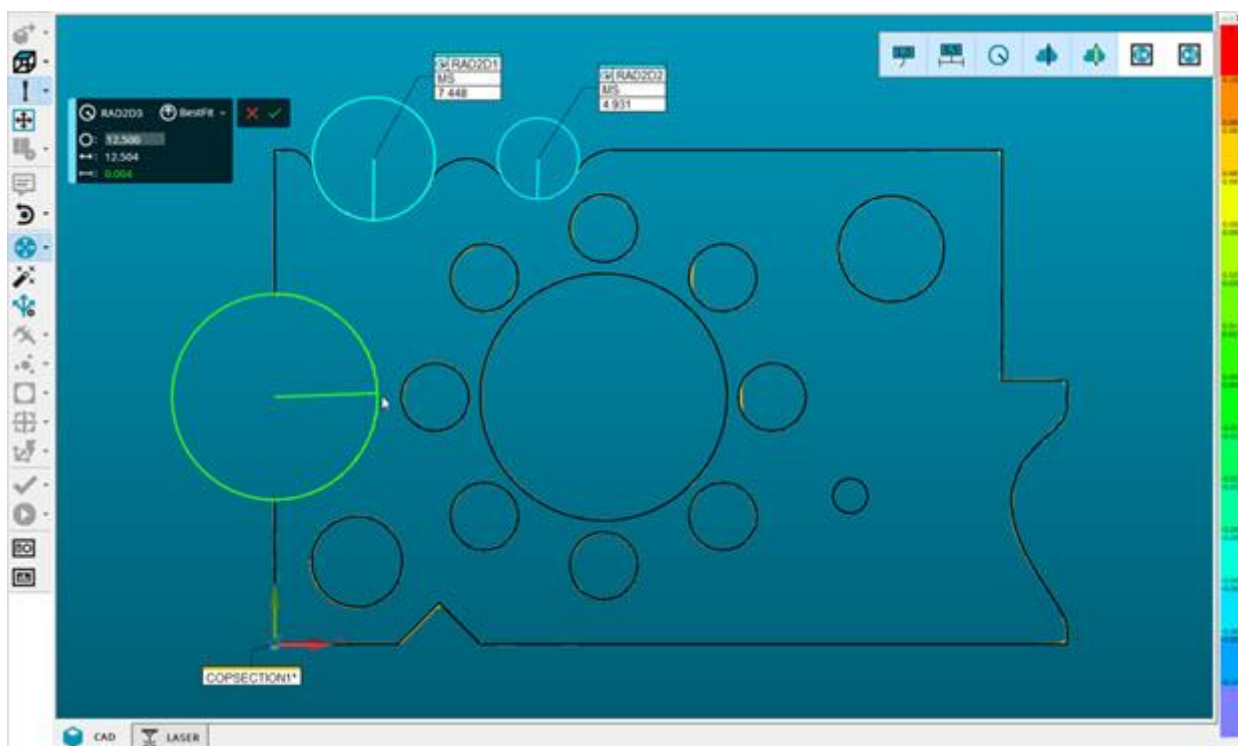
To create a 2D Radius Gage graphically, do the following:

1. After you create the cross sections, from the **Mesh**, **Pointcloud**, or **QuickCloud** toolbar (**View | Toolbars**), click the **Cross Section Slide Show** button () to display the cross sections in 2D view. For details, see the "Cross Section Slide Show" section of the "Show and Hide Cross Section Polylines" topic.
2. Hold down the Shift key, and move the mouse cursor onto the radius to view the nominal, measured, and deviation values in the display widget.

- Left-click to select the radius. You can create or cancel the radius gage from the widget dialog box.

The software uses a Least Square Best Fit algorithm to calculate the 2D Radius by default. The active tolerances are set on the Dimensions Color Bar. The radius gage graphic uses the color from the Dimensions Color Bar which corresponds to its deviation. For details on editing the dimension color scale, see "Editing Dimension Colors" in the PC-DMIS Core documentation.

You can change the tolerances for the gage from the Edit window, or press the F9 key to view the **2D Radius Gage** dialog box.



Examples of the 2D Radius Gage

By default, PC-DMIS automatically includes the 2D Radius Gage in the report.

	MM	RAD2D2 - COPSECTION1				
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
R	7.503	0.100	0.100	7.457	-0.046	0.000

Example of a 2D Radius Gage report

You can turn off showing the 2D Radius Gage in the report from the **Label/Reporting** tab of the **2D Radius Gage** dialog box. For details, see "2D Radius Gage Dialog Box".

Once you create a 2D Radius Gage, you can use it in Location and Distance Dimensions, and Constructions. For Location Dimension, Form is not supported.

2D Radius Gage Dialog Box

The **2D Radius Gage** dialog box has these tabs:

Properties tab

2D Radius Gage dialog box - Properties tab

The 2D Radius Gage is automatically linked to the cross section on which it was created. Since you created the 2D Radius Gage on the cross section, you cannot change the associated cross section.

The **Properties** tab of the **2D Radius Gage** dialog box has these areas:

Measurement type

- **Fit Type** - Click the drop-down arrow to display these options:

- **Best Fit** - The software applies a Best Fit Least Square fit to all data points that fall within the radius search zone.

Feature theoreticals - The software displays the XYZ centerpoint location and size of the nominal radius. You can edit the nominal values.

Feature actuals - The software displays the XYZ centerpoint and size of the measured radius. You cannot edit the actuals.

Label/Reporting tab

2D Radius Gage dialog box - Label/Reporting tab

The **Label/Reporting** tab of the **2D Radius Gage** dialog box has these areas:

Tolerances

The **Dimension Color Scale** defines the default 2D Radius Gage tolerances. For details, see "Editing Dimension Colors" in the PC-DMIS Core documentation.

The **Tolerances** section allows you to type the plus and minus tolerances for the radius.

To enter the plus, minus, and nominal tolerances, do the following:

1. Type the plus tolerance value in the **Plus** box.
2. Type the minus tolerance value in the **Minus** box.

If you are using a CAD model, the cross section nominal (black) polyline defines the nominal (theoretical) radius. If you are not using a CAD model, the software updates the nominal value with the initial measured value. You can edit the nominal value.

Label

Show check box - When you select this check box, the software displays the **2D Radius Gage** label and graphic in the Graphic Display window.

Show heading check box - This check box toggles the display of the row and column headings in the **2D Radius Gage** label. When you select this check box, the software displays the label's row and column headings.

Contents

The order in which you select the following check boxes defines the order in which they appear in the label. The order number appears to the left of each selected item. When you clear a check box, the software reorders the order numbers of the remaining selected check boxes accordingly.

Measured check box - When you select this check box, the software displays the measured data in the label.

Nominal check box - When you select this check box, the software displays the nominal data in the label.

Tolerances check box - When you select this check box, the software displays the tolerance data in the label.

Deviation check box - When you select this check box, the software displays the deviation data between the measured and nominal values in the label.

OutTol check box - When you select this check box, the software displays the out-of-tolerance data in the label.

Default button - Click to set the current selection of check boxes as the default.

Reset button - Click to clear all check box selections in the **Contents** area. The software then resets the section to the auto-setting configuration showing the measured value.

Report and statistics

From this section, you can use the options to control the output results:

STATS - When you select this option, the software sends the output to statistical files.

REPORT - When you select this option, the software sends the output to the inspection report.

BOTH - When you select this option, the software sends the output to the inspection report and the statistical files.


NONE - When you select this option, the software doesn't send output anywhere.

When PC-DMIS executes the command, the software sends the results to the specified output.

If you choose **STATS** or **BOTH**, a preceding STATS/ON command must exist inside the Edit window in order to send the results to the statistical file.

Creating a 2D Radius Gage

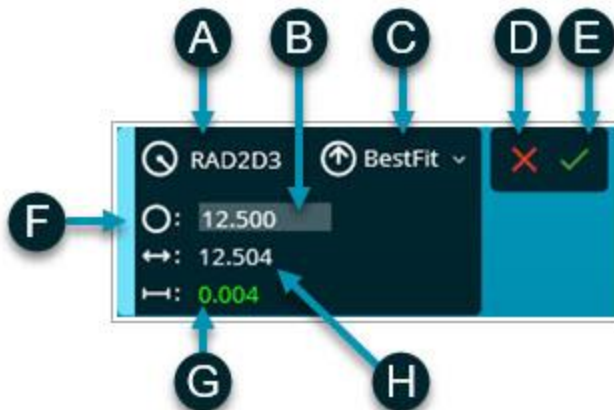
To create a 2D Radius Gage with a Cross Section:

1. Create the cross section. For details on creating a pointcloud cross section, see "CROSS SECTION". For details on creating a mesh cross section, see "Mesh CROSS SECTION Operator".
2. Select the **Cross Section Slide Show** button  from the **Pointcloud** toolbar (**View | Toolbars | Pointcloud**) to view the cross section in a 2D view.
3. Hold the Shift key down and move the mouse pointer onto the desired radius. A display widget appears. The display widget shows the nominal, measured and deviation values for the radius.



2D Radius Gage display widget showing the nominal, measured and deviation values for the radius

- Click the Left mouse button to select the radius. A widget dialog box appears.



- A** - 2D Radius Gage ID
- B** - Radius Nominal value
- C** - Algorithm used to calculate the radius
- D** - Cancel button
- E** - Create button
- F** - Use the bar to move the widget dialog box
- G** - Radius Deviation value
- H** - Radius Measured value

2D Radius Gage widget dialog box

From the widget dialog box, you can do the following:

- Change the 2D Radius Gage ID (**A**) and the nominal value (**B**).
 - From the list (**C**), select the algorithm the software uses to calculate the radius.
 - Click the **Create** button (**E**) to create the radius gage, or the **Cancel** button (**D**) to close the widget dialog box without creating the radius gage.
 - Position the mouse cursor over the bar on the left side of the widget (**F**). Click and hold the left mouse button, then drag the widget in the Graphic Display window to reposition it. Release the mouse button when the widget is in the desired location.
5. When you create the 2D Radius Gage, its associated command is created in the Edit window. You can create additional radius gages as needed.

Once you create a 2D Radius Gage, you can use it in Location and Distance Dimensions, and Constructions. For Location Dimension, Form is not supported.

To change the radius settings:

- Edit them directly in the Edit window.
- Click the radius gage command in the Edit window, and then press F9 to open the **2D Radius Gage** dialog box to make your changes.

How the 2D Radius Gage is Calculated

- When the cross section has both nominal (black polyline) and measured (yellow polyline) data:

Calculating the Nominal 2D Radius

Starting from the initial picked measured point, the nominal radius is found on the nearest black polyline. The software calculates the nominal (theoretical) radius for a least square best fit circle, using all the nominal points that are within 0.005 mm standard deviation.

Calculating the measured 2D Radius

The software calculates a least square best fit circle using the actual points on the yellow polyline that are associated with the nominal points.

- When the cross section has only nominal data (black polyline):

Starting from the initial picked nominal point, the software finds the radius on the nearest black polyline. The software calculates the nominal (theoretical) radius for a least square best fit circle, using all the nominal points that are within 0.005 mm standard deviation.

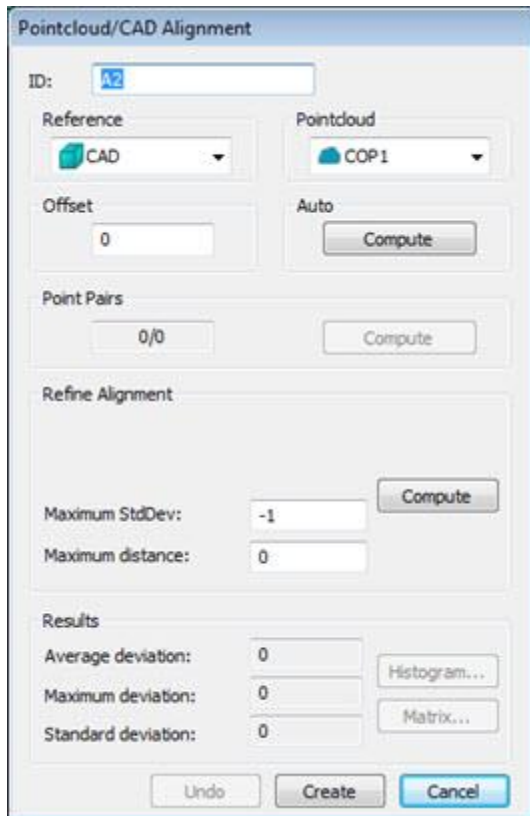
- When the cross section has only measured data (yellow polyline):

Starting from the initial picked measured point, the software calculates the radius for a least square best fit circle. The software uses all the measured points within 0.050 mm standard deviation and a search distance of 0.25 mm to find any additional segments that belong to the radius.

Pointcloud Alignments

In order to use the data you've collected in your pointclouds properly, you need to create an alignment between the pointclouds and the CAD data of your part model or between pointclouds. This is done using the **Pointcloud/CAD Alignment** dialog box.

Pointcloud/CAD Alignment Dialog Box Description



Default view of Pointcloud/CAD Alignment dialog box

The **Pointcloud/CAD Alignment** dialog box contains these options:

ID - This displays the identification label for the alignment.

Reference - Select the point of reference for your alignment, usually either from the CAD itself or a defined COP.

Pointcloud - This list lets you choose the cloud of points to use in the alignment.

Offset - This defines an offset value for a surface CAD model and is typically used with sheet metal parts. Applying an offset value essentially gives the surface CAD model a thickness so you can align the pointcloud data to a different face that isn't represented in the surface CAD model. For example, if you have a surface CAD model for the top of a part but you want to align to a corresponding bottom surface, you could apply an offset value of the part's thickness to align the scanned data to the bottom side. Use a positive value if you want to apply a thickness in the same direction as the surface normal vector; use a negative value if you want to apply a thickness opposite the surface normal. This option is available for Pointcloud to CAD alignments.

Auto - This area lets you automatically align the CAD with the cloud of points by using the **Compute** button. This option is available for Pointcloud to CAD alignments.

Point Pairs - This area lets you create a rough alignment based on selected points from the CAD that correspond to selected points from the pointcloud. Once you have the needed pairs selected, you can use the **Compute** button to perform the rough alignment.

Refine Alignment - This area allows for a more refined alignment. Only the **Maximum Distance** option is available for Pointcloud to Pointcloud alignments.

Depending on the alignment being made, the **Refine Alignment** area of the dialog box may consist of the following items:



The first two options (**Total points** and **Maximum iterations**) are only available if PC-DMIS IS NOT set up to use the Reshaper SDK for alignment computations. For details on using the SDK for alignment computations, see the topic "[UseSDKForCopCadAlignments](#)" in the PC-DMIS Settings Editor documentation.

Total points - This box defines the number of random sampled points used to refine the alignment. This number must be a value of at least 3. A good number is around 200 points.

Maximum iterations - This box defines the number of repetitions the process will make in order to refine the alignment.

Compute - This button begins the refined alignment process. A progress bar on the status bar shows the progress as the process moves through the alignment iterations.

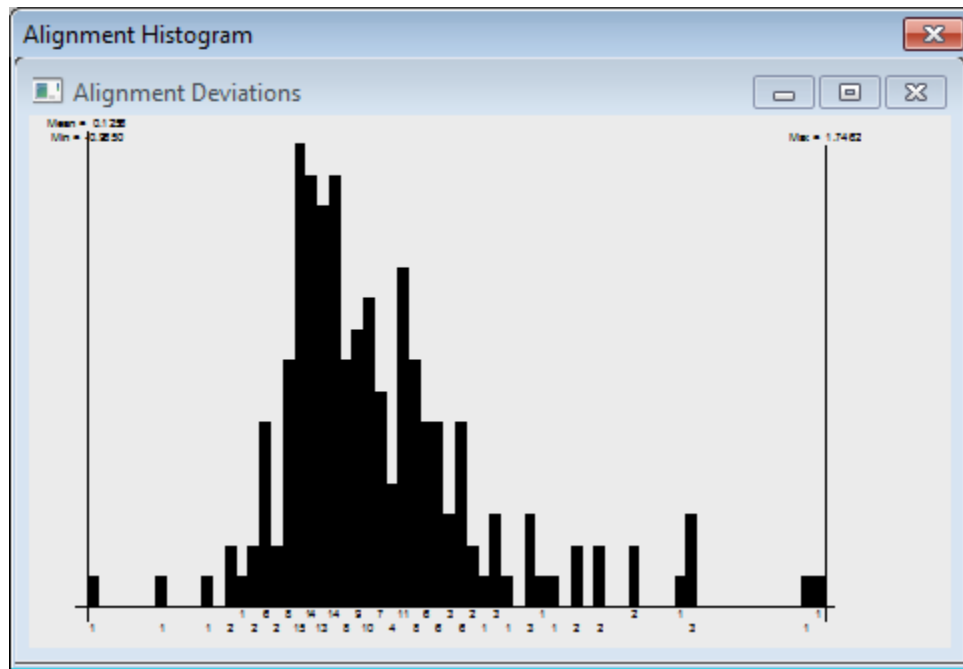
Maximum StdDev - Maximum StdDev is the maximum standard deviation used during the execution of an auto alignment. If the entered value is exceeded during the command execution, you are prompted to optionally pick point pairs on the CAD/Pointcloud. A value of -1 disables the Maximum StdDev functionality.

Maximum Distance - Defines the maximum distance PC-DMIS looks from the CAD for valid COP points. If no value is entered, the default value of 0 (zero) is used and the maximum distance becomes half the distance of the CAD bounding box.

Results - This area contains the following items:

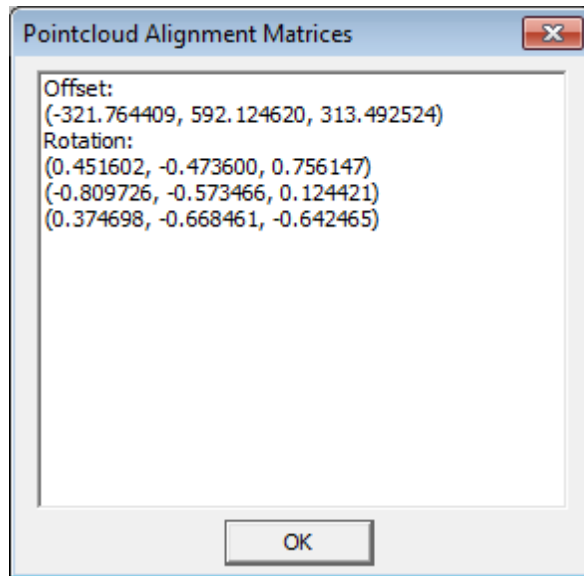
Information boxes showing the **Average Deviation**, **Maximum Deviation**, and **Standard Deviations** of the cloud of points in relation to the CAD model.

Histogram - This button takes a random sample of points from the pointcloud, projects them onto the CAD, and then shows the deviations for that sample in the **Pointcloud Alignment Histogram** dialog box.



Sample Pointcloud Alignment Histogram dialog box

Matrix - This button displays the **Alignment Matrices** dialog box for the pointcloud alignment. The numerical values of the alignment: the offset and the rotation matrix are listed.

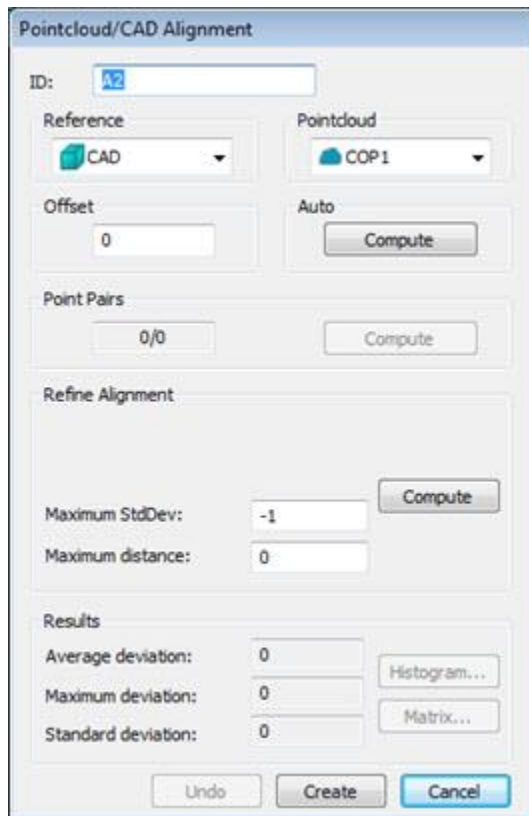


Sample Alignment Matrices dialog box for the alignment

Creating a Pointcloud/CAD Alignment

To create a Pointcloud to CAD alignment, do the following:

1. Ensure that you have an imported CAD model in the Graphic Display window and a [COP](#) command in the measurement routine. These elements are required to align pointclouds to the CAD.
2. Select the **Insert | Pointcloud | Alignment** menu option. You can also access this dialog box by typing the [COPCADBF](#) command in the Edit window's Command mode between the [ALIGNMENT/START](#) and [ALIGNMENT/END](#) commands. The dialog box appears:

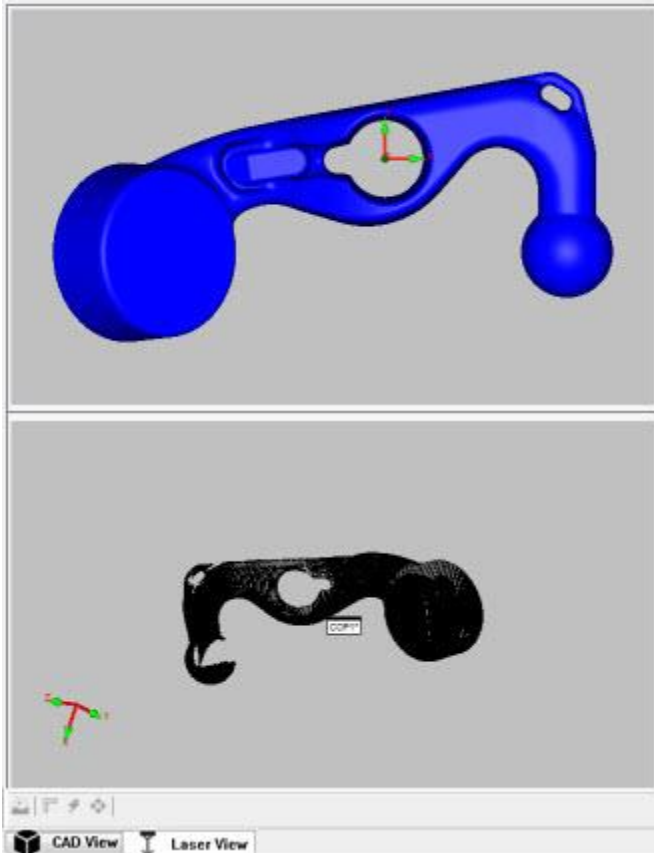


Pointcloud/CAD Alignment dialog box



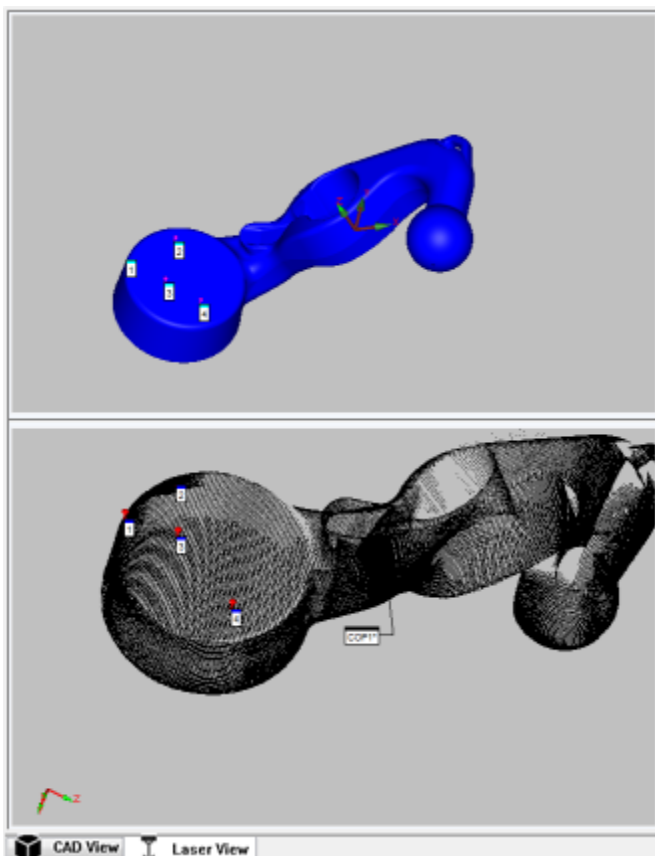
For a complete description of the **Alignment** dialog box, see the "Pointcloud/CAD Alignment Dialog Box Description" topic in the PC-DMIS Laser documentation.

3. A temporary and split-screen view of the CAD model and pointclouds appears in the Graphic Display window. You can use this split-screen view to visually see the alignment taking place. Select your point of reference from the **Reference** drop down list; usually, either the CAD model itself or a defined COP is available.



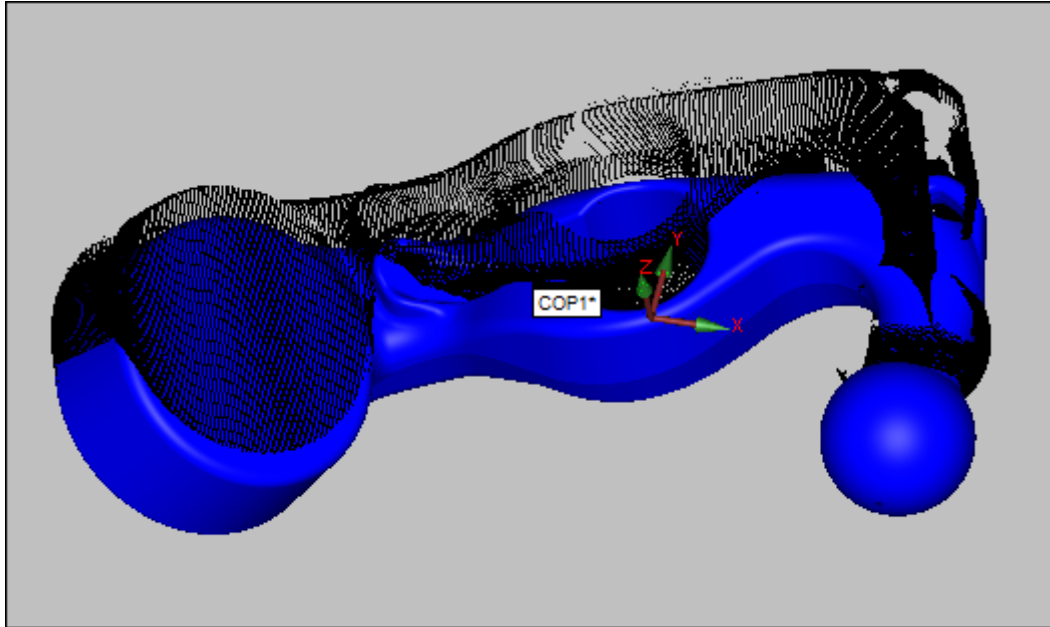
Split-screen view showing the CAD model on the top view and the pointcloud on the bottom view

4. If you have more than one pointcloud in your measurement routine, choose the pointclouds from the **Pointcloud** list.
5. Perform the alignment:
 - a. Click the **Compute** button in the **Auto** section. You should only use this when you have a full scan of the external faces of the part. This automatically performs an alignment of the Pointcloud to the CAD and also a refinement on the alignment as it's generating.
 - b. First, use the **Pointcloud/CAD Pairs** area to perform a rough alignment that brings the pointcloud close enough to the CAD (if not already close) to be able to refine the alignment further if needed. You should use this type of alignment if the pointcloud is not complete or it contains scanned data belonging to a fixture, the table, and so forth.
 - Click a desired number of points on the pointcloud.
 - Click corresponding locations on the CAD model. ⓘ



Split-screen view showing selected CAD points (top) and corresponding pointcloud points (bottom)

- The more points you take around the different areas of the model and pointcloud, the better the rough alignment.
 - Click **Compute** to create the rough alignment.
- c. Next, use the **Refine Alignment** area whenever you want to refine your alignment, thereby bringing the pointcloud closer to your CAD model. In order to be able to obtain a good refined alignment, the pointcloud points should be close enough to the CAD points through an initial rough alignment. ⓘ



A sample rough pointcloud to CAD alignment that needs a refinement

- Define the total number of random sample points to use in each iteration in the **Total Points**.
 - Define the number of iterations in the **Maximum Iterations** box.
 - Define the maximum standard deviation for the auto alignment execution between the points in the pointcloud and the CAD model using the **Maximum StdDev** box. When the auto alignment command is executed, if the standard deviation of the Pointcloud/CAD deviations is greater than the maximum value defined, you can select point pairs to get a better alignment. The default value of -1 is equivalent to an infinite allowed standard deviation.
 - Define the maximum distance of the points from the CAD for use in the best fit routines. The default value is 0. In this case, an internal max distance based on the size of the pointcloud is used.
 - Click **Compute** to refine the alignment.
6. If a portion of the pointcloud doesn't align nicely with the CAD, you can click the **Undo** button and recompute the alignment using the same type of alignment with additional parameters. Or, you can try a different alignment.
 7. If you have a surface model representing a sheet metal part, and you want to align to the offset faces, define an **Offset** value representing the constant thickness of the sheet metal part.
 8. Use the **Results** area to determine how well the pointcloud aligned with the CAD. Make any changes to the **Offset** or **Refine Alignment** values to improve the

alignment if necessary. If any changes are made, be sure to click the **Compute** button to regenerate the alignment with the new values.

9. Once you're satisfied with the alignment, click **Create**. PC-DMIS closes the temporary split-screen view and places the **COPCADB**F command in the Edit window. See the "COPCADB F Command Mode Text" topic.



If needed, you can adjust the `CadGridSizeForPointcloudCadAutoAlignment` registry entry to define the distance between the grid of points used to align the pointcloud to the CAD model.

COPCADB F Command Mode Text

The COPCADB F command allows you to perform a best fit alignment of the pointcloud data with the CAD data.

Below is an example code snippet for a COPCADB F alignment:

```
A1 =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
    COPCADB F/REFINE=n1,n2,n3,n4,n5,SHOWALLPARAMS=TOG1,
    ROUGH ALIGNPAIR/
        THEO/<x,y,z>,<i,j,k>,
        MEAS/<x1,y1,z1>
    REF,TOG2,,
ALIGNMENT/END
```

n1 represents the total number of sample points to use in the refinement.

n2 represents the maximum number of iterations.

n3 represents the offset value for applying a thickness.

n4 represents the maximum standard deviation value.

n5 represents the maximum distance value.

TOG1 lets you show or hide the parameters used for the rough alignment. It can be set to YES or NO.

```
ROUGH ALIGNPAIR/
```

```
THEO/x,y,z,i,j,k,
MEAS/x1,y1,z1
```

These rough alignment pairs of points are defined and selected using the Graphic Display window. The values next to **THEO/** represent the point on the CAD. The values next to **MEAS/** represent the corresponding point on the COP. These pairs are used to determine a rough transformation between the CAD and the COP which allows the COP to come close enough to the CAD to allow further refinements of the alignment.

TOG2 lets you choose the pointcloud to use for the alignment.

Creating a Pointcloud/Pointcloud Alignment

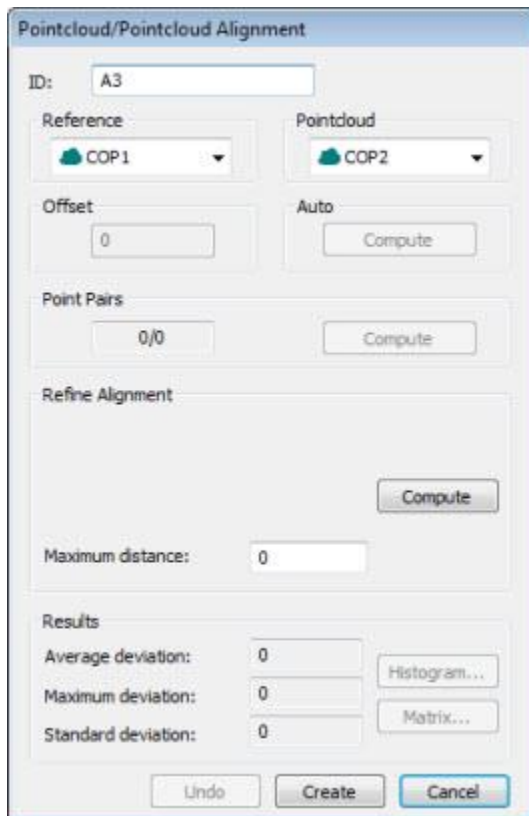
The Pointcloud to Pointcloud alignment functionality allows you to best fit align two pointclouds which have been collected in two different reference frames that have some overlap. A typical example is two scans in two pointcloud commands, representing areas of a part that cannot be scanned in the same part orientation.

The alignment is done in two steps:

- A rough alignment, where pairs of points in the overlapping area of the two pointclouds are selected.
- A refined bestfit, which tries to bring the second pointcloud as close as possible to the reference pointcloud.

To create a Pointcloud to Pointcloud alignment, do the following:

1. Ensure that you have two or more COP commands in the measurement routine that you are using to align. These elements are required to align two pointclouds.
2. Select the **Insert | Pointcloud | Alignment** menu option. You can also access this dialog box by typing the **COPCOPBF** command in the Edit window's Command mode between the **ALIGNMENT/START** and the **ALIGNMENT/END** commands. The dialog box appears:

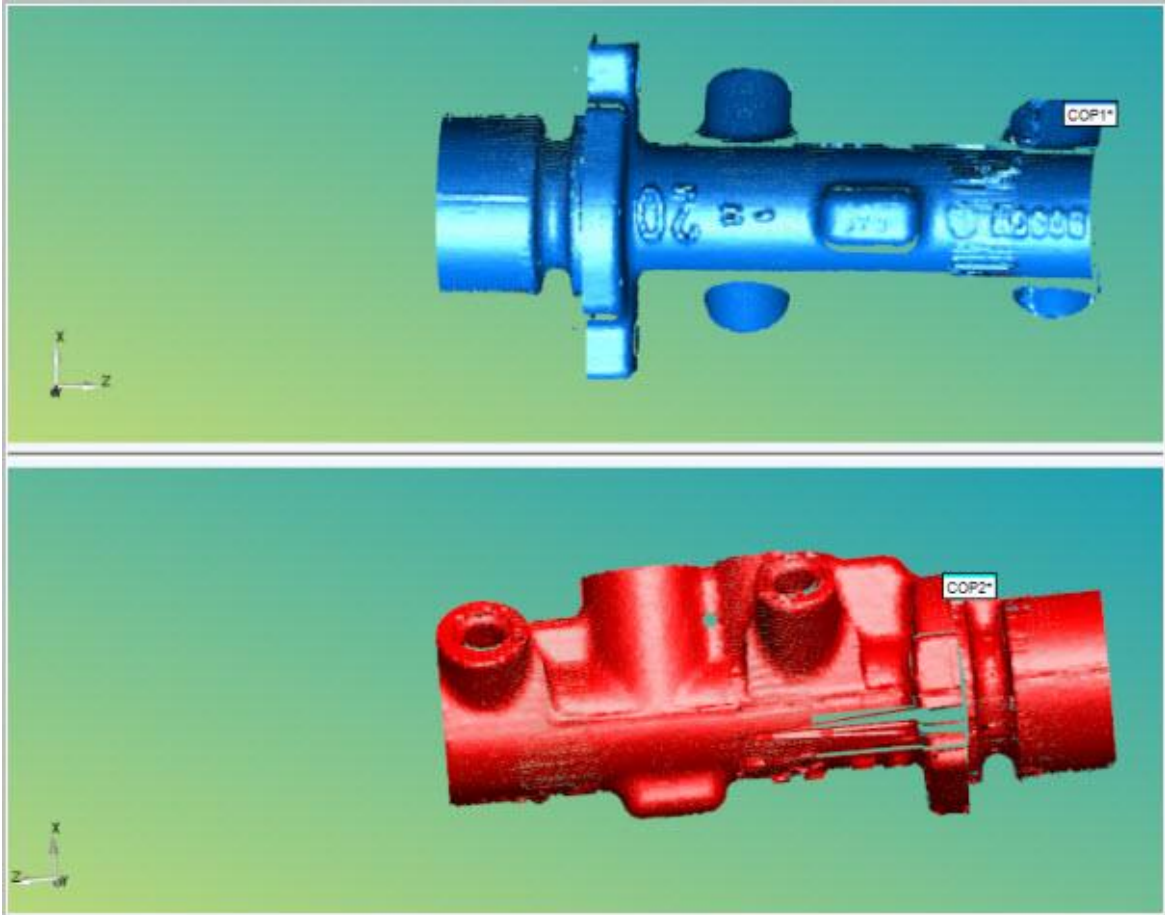


Pointcloud/Pointcloud Alignment dialog box




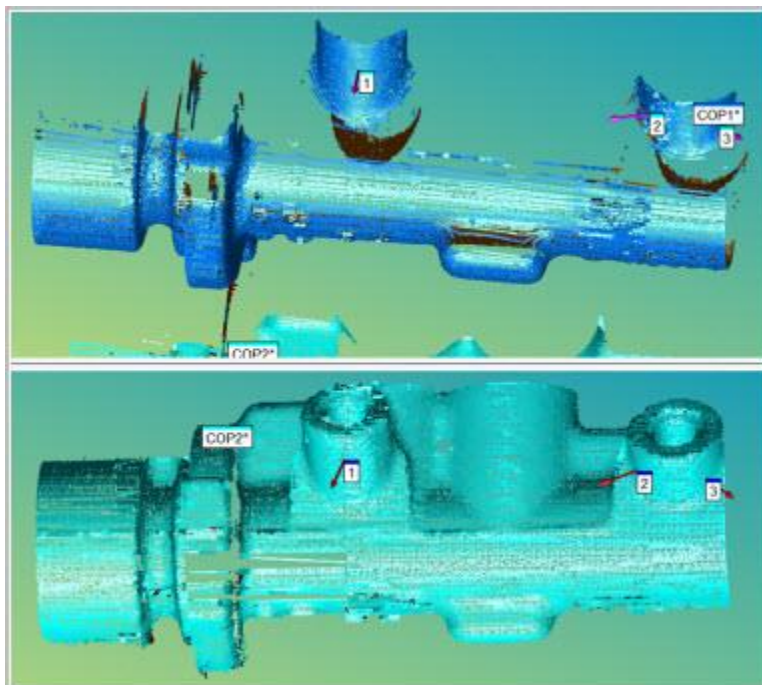
For a complete description of the dialog box, see the topic "Pointcloud/CAD Alignment Dialog Box Description".

3. A temporary split-screen view of the two pointclouds appears in the Graphic Display window. You can use this view to visually see the alignment taking place. Select the first COP to be used as a point of reference from the **Reference** drop down list.



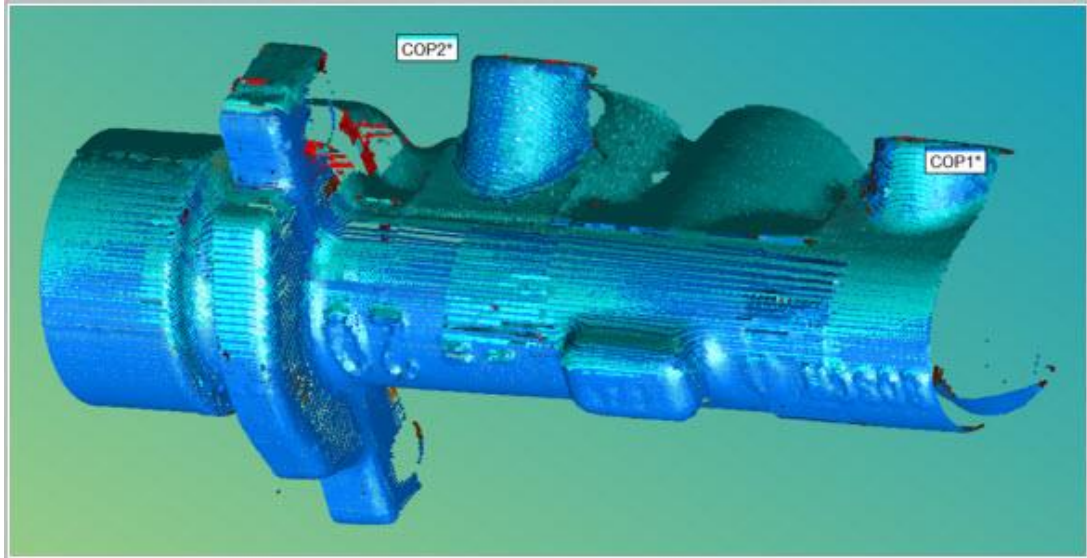
Split-screen view showing a pointcloud to pointcloud alignment

4. Use your mouse to manipulate and orient each view as needed to create the Point Pairs.
5. Perform the alignment:
 - First, use the **Point Pairs** area to perform a rough alignment that brings the pointclouds close enough to each other. This is a mandatory step.
 - Click a desired number of points (at least three pairs) on each of the pointclouds within the overlap area. ONLY click points on the overlap area of the two pointclouds. 



Split view showing selected COP1 and COP2 pointclouds

- The more points you take around the overlap area of the pointclouds results in an improved alignment. Click **Compute** to create the rough alignment.
- Next, use the **Refine Alignment** area whenever you want to refine your alignment, thereby bringing the two pointclouds closer to each other. In order to get a good refined alignment, the two pointcloud points should be close enough to each other through the initial rough alignment. ⓘ



A sample rough pointcloud to pointcloud alignment that needs a refinement

- Define the maximum distance between the points in the two pointclouds using the **Maximum Distance** box. The default value is 0 (zero). If the default value is used, PC-DMIS uses an internal default value related to the dimensions of the pointclouds.
 - Click **Compute** to refine the alignment.
6. If a portion of one pointcloud doesn't align nicely with the other, you can click the **Undo** button and recompute the alignment using the same type of alignment with additional parameters, or you can try a different alignment.
 7. Once you're satisfied with the alignment, click **Create**. PC-DMIS closes the temporary split-screen view and places the **COPCOPBF** command in the Edit window. For details on the COPCOPBF command, see the "COPCOPBF Command Mode Text" topic in the PC-DMIS Laser documentation.

COPCOPBF Command Mode Text

The COPCOPBF command allows you to perform a best fit alignment of the reference pointcloud with a second pointcloud.

Below is an example code snippet for a COPCOPBF alignment:

```
A1 =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
    COPCOPBF/REFINE,SHOWALLPARAMS=TOG1,
    ROUGH ALIGNPAIR/
        THEO/<x,y,z>,<i,j,k>,
```

```

    MEAS/<x1,y1,z1>
    REF,TOG2,TOG3,,
ALIGNMENT/END

```

TOG1 lets you show or hide the parameters used for the rough alignment. It can be set to YES or NO.

```

ROUGH ALIGNPAIR/
    THEO/x,y,z,i,j,k,
    MEAS/x1,y1,z1

```

These rough alignment pairs of points are defined and selected using the Graphic Display window. The values next to **THEO/** represent the point for the reference COP. The values next to **MEAS/** represent the corresponding point on the second COP. These pairs are used to determine a rough transformation between the reference COP and the second COP which allows the two pointclouds to come close to allow further refinements of the alignment.

TOG2 determines the reference COP used for aligning to the second COP.

TOG3 determines the second COP used for the alignment back to the reference COP.

TCP/IP Pointcloud Server

PC-DMIS has several options that use TCP/IP communication to watch or connect to a third-party client.

Generic OFFLINE TCP/IP Import Pointcloud Function

This OFFLINE function allows you to import a pointcloud from a client application into PC-DMIS (the Server application). When PC-DMIS receives the new pointcloud data, it automatically executes the inspection routine offline. See "Generic Import File Formats".

From the **Pointcloud** toolbar, click the **TCP/IP Pointcloud Server receive data** button



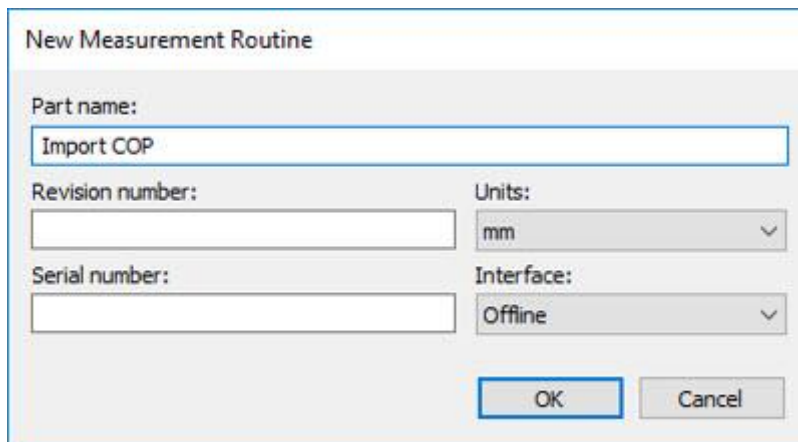
to put PC-DMIS in a "watch" state. When in this state, PC-DMIS is ready and waiting to receive a pointcloud file. The client application must initiate sending the pointcloud data. This button only appears when you run PC-DMIS in Offline mode. Click the button a second time to turn this function off.

When PC-DMIS detects a new pointcloud file:

- If the measurement routine already contains a COP (cloud of points), PC-DMIS replaces the COP with the received data and then executes the measurement routine.
- If the measurement routine does not contain a COP, PC-DMIS creates a COP feature, imports the data, and then executes the measurement routine.

To create the initial measurement routine for OFFLINE execution:

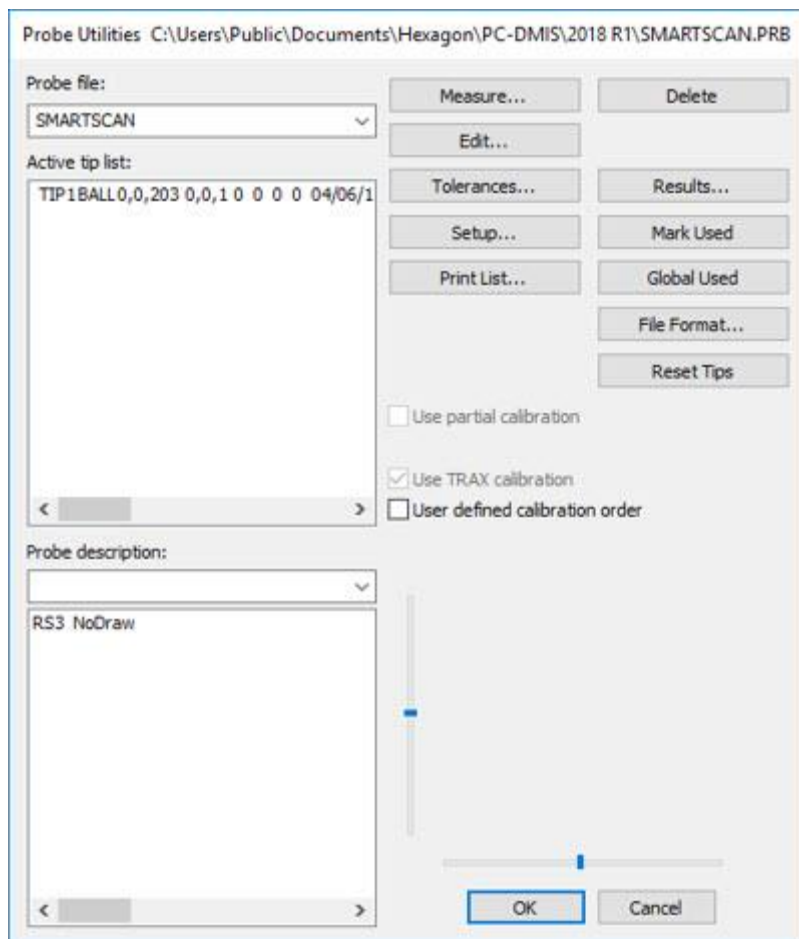
1. Create the PC-DMIS measurement routine with the offline interface.




The image shows a dialog box titled "New Measurement Routine". It contains the following fields and controls:

- Part name:** A text input field containing "Import COP".
- Revision number:** An empty text input field.
- Units:** A dropdown menu currently showing "mm".
- Serial number:** An empty text input field.
- Interface:** A dropdown menu currently showing "Offline".
- Buttons:** "OK" and "Cancel" buttons at the bottom right.

2. The software displays the **Probe Utilities** dialog box. Select SMARTSCAN as the active offline laser probe.

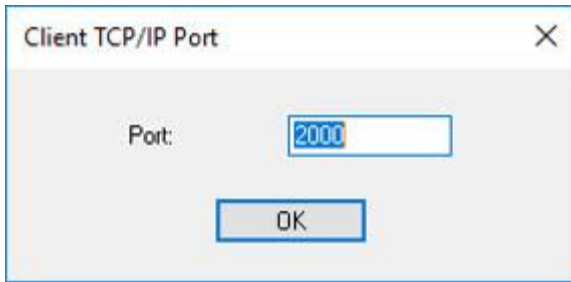


3. From the **Pointcloud** toolbar, select the **TCP/IP Operations** button and then the **TCP/IP Pointcloud Server receive data** button .



The **TCP/IP Pointcloud Server receive data** button is only available when PC-DMIS is run in Offline mode.

4. From the **Client TCP/IP Port** dialog box, enter the Port ID, and click **OK**. You can find the Port ID in the client application.



5. PC-DMIS begins to import the pointcloud data as soon as the client application initiates the Send function. The software displays the progress of the incoming data in the PC-DMIS status area, which is located in the lower-left corner.
6. Create any needed pointcloud commands (for example, Pointcloud Alignment, Pointcloud Surface Colormap, etc.), Auto Features, and Dimensions.
7. Save the measurement routine.

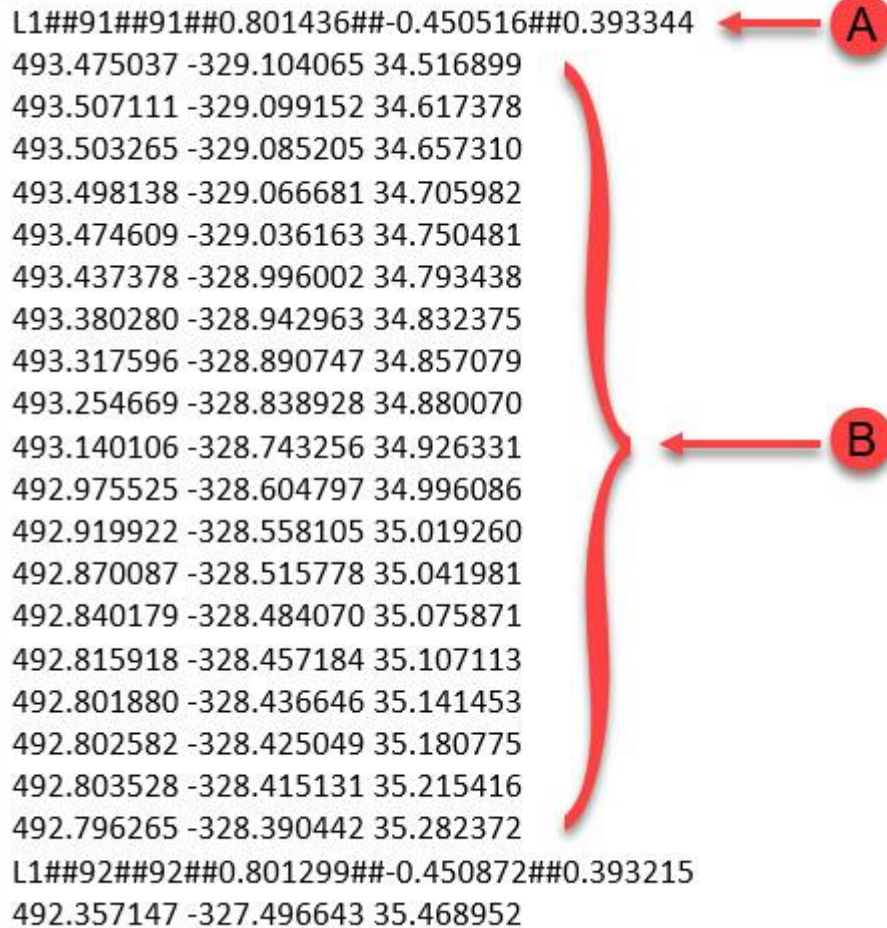
Generic Import File Formats

PC-DMIS allows the import of these pointcloud formats:

- Data per set of points



Data where each laser stripe, line or patch defines the vector (IJK) of the stripe, followed by XYZ value of the points on the stripe. Point XYZ values can be space-delimited or comma-delimited.



```

L1##91##91##0.801436##-0.450516##0.393344
493.475037 -329.104065 34.516899
493.507111 -329.099152 34.617378
493.503265 -329.085205 34.657310
493.498138 -329.066681 34.705982
493.474609 -329.036163 34.750481
493.437378 -328.996002 34.793438
493.380280 -328.942963 34.832375
493.317596 -328.890747 34.857079
493.254669 -328.838928 34.880070
493.140106 -328.743256 34.926331
492.975525 -328.604797 34.996086
492.919922 -328.558105 35.019260
492.870087 -328.515778 35.041981
492.840179 -328.484070 35.075871
492.815918 -328.457184 35.107113
492.801880 -328.436646 35.141453
492.802582 -328.425049 35.180775
492.803528 -328.415131 35.215416
492.796265 -328.390442 35.282372
L1##92##92##0.801299##-0.450872##0.393215
492.357147 -327.496643 35.468952
    
```

A - Line (laser stripe or patch) number unique identification number (optional) IJK of line (from sensor orientation)

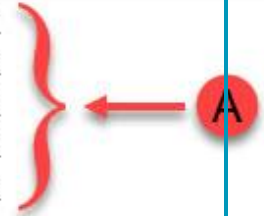
B - XYZ value of points on the line

- Data point



The data file defines the XYZ or XYZIJK value for each point. For these data types, XYZIJK is preferred as PC-DMIS uses the vector of the points in pointcloud operations such as Surface Colormaps and Feature Extraction. The following example shows points with XYZIJK values.

218.897448, 68.555506, -0.449651, -0.029287, -0.000550, 0.999571
218.534121, 68.249378, -0.460403, -0.029287, -0.000550, 0.999571
218.586008, 68.248738, -0.458884, -0.029287, -0.000550, 0.999571
218.638085, 68.558736, -0.456699, -0.029287, -0.000550, 0.999571
218.845633, 68.556175, -0.449459, -0.029287, -0.000550, 0.999571



A - XYZIJK value of each point

Generic ONLINE TCP/IP Export Pointcloud Functions

PC-DMIS can send your pointcloud data to a custom-built, third-party software application. It uses a TCP/IP communication protocol to do this. To establish the connection, your custom application must be able to load a dynamic link library (dll) file named PcDmisPointCloudClientDll.dll. You can request this file from Hexagon Technical Support.

Once your application loads the .dll file, click on one of these TCP/IP Pointcloud server icons available from the PC-DMIS **Pointcloud** toolbar to establish the connection:



TCP/IP Pointcloud Server Connection with Local Copy - Establishes the connection with the client and sends the pointcloud data directly to the client. When the scan finishes, the pointcloud data remains inside the measurement routine.



TCP/IP Pointcloud Server Connection without Local Copy - Establishes the connection with the client and sends the pointcloud data directly to the client. When the scan finishes, the pointcloud data is deleted from the measurement routine.

Extracting Auto Features from Pointclouds

Laser Auto Features can be extracted from scanned pointcloud data. Once the Auto Features are set up, you can simply scan the part and PC-DMIS extracts the Auto Feature information from the scan. You can include and extract multiple Auto Features from a single pointcloud.

Review the following topics to execute Auto Feature extraction from manual scans:

- Defining a Laser Auto Feature by Clicking on a Pointcloud
- Executing Scan-Extracted Auto Features
- Aligning Measured Auto Features to CAD

Defining a Laser Auto Feature by Clicking on a Pointcloud

Often, users will define Auto Features by clicking on the CAD. In the case where no CAD exists, you can perform a scan of the part, and then click on the individual Pointcloud points to define your Auto Feature; or you can box-select the feature from the point cloud.

To define an Auto Feature from Pointcloud points:

1. Scan the surface of the part in which the needed Auto Feature exists.
2. Click the needed Auto Feature from the **Auto Feature** toolbar or the **Insert | Feature | Auto** submenu. This opens the **Auto Feature** dialog box.
3. Either select points from the cloud of points that best define the feature's nominal position or drag a box directly on the cloud of points to have PC-DMIS extract the feature from the points within the dragged box. PC-DMIS will define the Auto Feature based on your selection.

Defining Features by Selecting Points

The following table shows the number of points that are needed to define an Auto Feature's location.

Feature	Points to Select
Surface Point	Select one point at the needed location within the measured surface area.
Edge Point	Select one point at the needed location along the measured edge.
Plane	Select at least three points that best define the needed plane's nominal position.
Circle	Select at least three points around the perimeter of the measured circle.
Round Slot	Select three points along one of the slot's arcs then select another three points along the other arc.

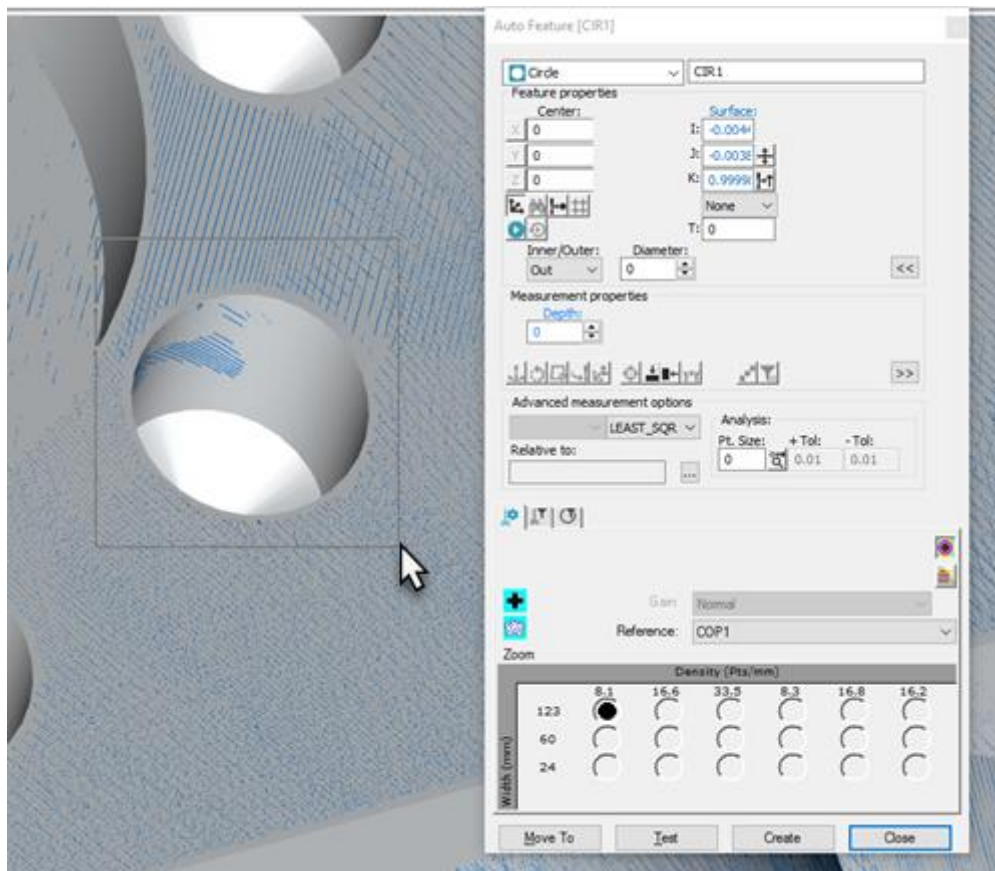
Square Slot	Type the slot's nominal Width in the Auto Feature dialog box. Select two points along a long side of the slot. Select one point on a short side of the slot. Select one point on the other long side of the slot. Finally, select one point on the other short side of the slot.
Flush and Gap	Select a point on each side of the gap.
Cylinder	Select three points for each of two circles that define the extents of the cylinder's form and length.
Sphere	Select at least five points around the surface of the measured sphere.

Defining Features by Box-Selecting

During learn mode, you can drag a box around the desired feature on the pointcloud to extract supported auto features using the selected data points.

This functionality has these limitations:

- PC-DMIS only calculates the surface vector. You may need to define the angle vector manually, such as for a polygon feature.
- If your box selection includes points at multiple depths in the Z axis, it may result in a poor feature extraction. You can avoid this by either clipping the acquisition or by using `COP/OPER, SELECT` to exclude those points prior to the box selection.



Example Circle Feature Creation by Box-Selecting

This works with these supported features:


- Surface Point
- Plane
- Circle
- Round Slot
- Square Slot
- Sphere
- Polygon

For all other auto features, you must use the point selection method.

Executing Scan-Extracted Auto Features

When executing manual scans from which Auto Features are extracted, you should do the following:

1. Scan the Auto Features in your measurement routine in any order. You can do this with one or more passes. After the first pass, if the scan's pointcloud points have changed for a feature, then the feature's measured values are recalculated.
2. When all the Auto Features associated with the scan have solved successfully, the command in the Edit window is highlighted in yellow.
3. When Auto Features have solved and have been reported correctly, the command in the Edit window is highlighted in green.
4. If additional scan data is taken for a feature that has already been solved, the feature's measured values are updated again with the new solution.
5. Once all of the included Auto Features have been solved, you may choose to continue scanning to further refine the measured results, or you can click the

Scan Done button () from the **Execution** dialog box. You can also finish the scan by pressing the done button on your measurement arm.



The **Scan Done** button is not available until all the included Auto Features are successfully measured.

See the "Using Pointclouds" topic.

Aligning Measured Auto Features to CAD

The procedure is only available when you measure Auto Features with a manual laser sensor (on a portable arm) and with imported CAD data. This allows you to select the *actual* measured features from the pointcloud that correspond to selected *nominal* features from the CAD.

To align measured auto features to CAD nominals:

1. Import CAD data.
2. Open the **Auto Feature** dialog box for a feature you want to include in the manual alignment.
3. Select the nominal location for the feature. Click on the CAD surface next to the feature to do this.
4. Change any auto feature parameters as needed and click **Create** to add the auto feature to the measurement routine.
5. Repeat steps 2 through 4 for each auto feature you want to include in the alignment.



PC-DMIS automatically adds a new extraction COP when you begin to create a new laser auto feature. You may include the features of the manual alignment in the same pointcloud. The Laser Probe Toolbox: Laser Scan Properties tab determines the COP from which the software extracts the laser auto features.

6. Execute the measurement routine. PC-DMIS prompts you to scan the laser auto features as part of a Portable Laser Alignment.
7. Scan the part to include the auto features for the manual alignment. You may need to perform more than one scan to adequately define each feature.
8. When you have completed scans, on your measurement arm, press the **Done** button.
9. PC-DMIS now prompts you to define the first manual alignment feature. Follow the instructions provided in the dialog box and status bar and then click **OK**. At the end of selection the software displays the preliminary form of the auto feature.
10. Repeat Step 9 for each of the manual alignment features.



PC-DMIS solves the laser auto feature with the theoretical values from the CAD and actual values from the measured pointcloud.

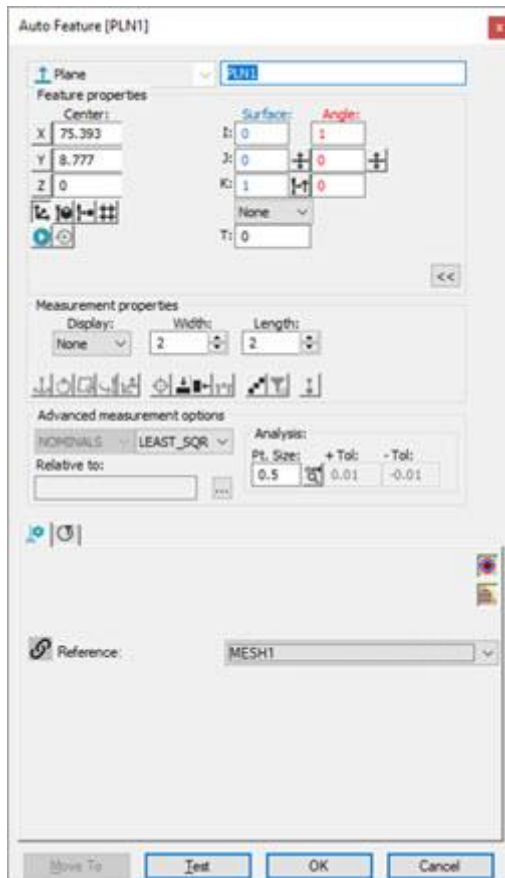
11. Select the **Insert | Alignment | New** menu item (Ctrl+Alt+A) to open the **Alignment Utilities** dialog box.
12. From the list box, select the alignment features, and click **Auto Align**. PC-DMIS aligns the defined features from pointcloud with the corresponding CAD nominals. This establishes the manual laser alignment.

Extracting Auto Features from a Mesh


You can extract a Laser Auto Feature from a mesh data object with the Laser **Auto Feature** dialog box.



Please see "Extracting a Laser Auto Surface Point from a Mesh" for details about extracting Laser Auto Surface Points from a Mesh data object.



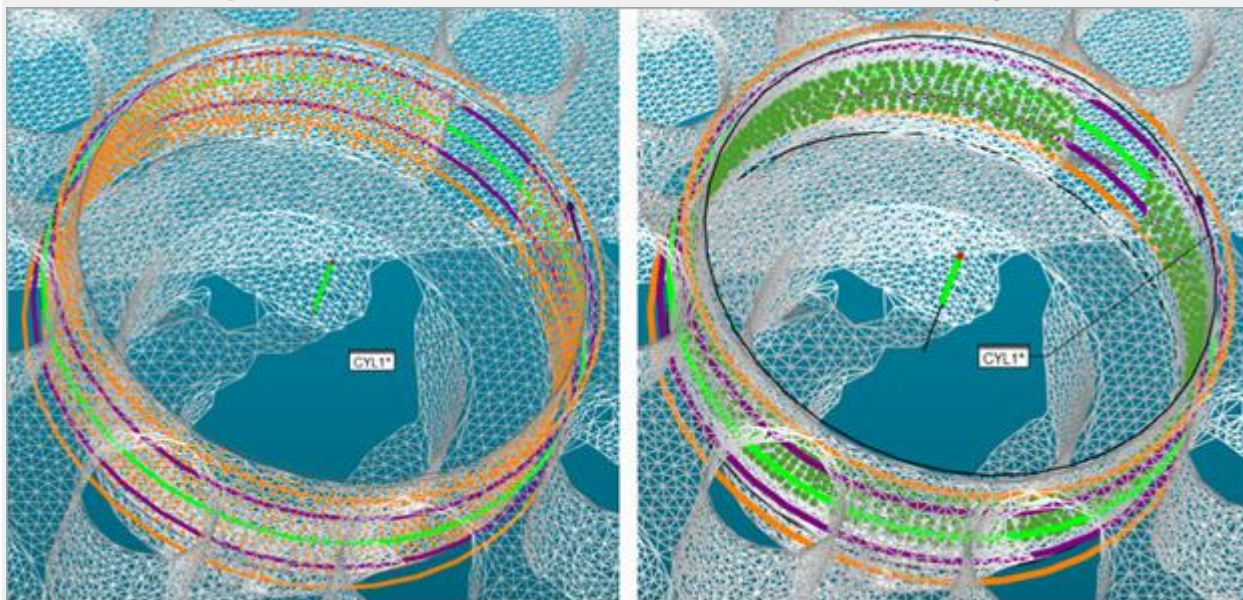
If there's only one Mesh in your measurement routine, PC-DMIS defaults to the Mesh as your reference data object. If there's a COP (or more than one COP) and one or more Mesh data objects, you need to select the correct reference data object from the **Reference** list of the **Feature Extraction** tab in the Probe Toolbox.

When extracting the Laser Auto Feature from a Mesh data object, all the triangle vertices inside the extraction zone, defined by the horizontal and vertical clipping, are first considered. To view the points which fall within the extraction zone, click the **Show/Hide segregated points** button () on the **Laser Scan Properties** tab.

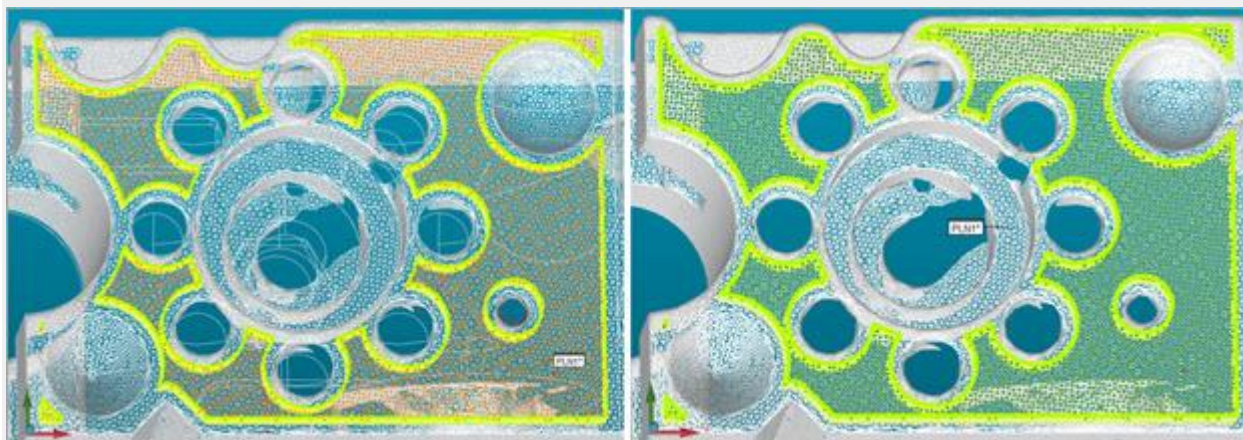
Click the **Test** button to measure the feature and view its measured points.



Examples of features extracted from a Mesh data object



Example of a Cylinder Auto Feature extracted from a Mesh data object



Example of a Plane Auto Feature extracted from a Mesh data object




Orange points show segregated points found within the extraction zone.

Green points show measured points after PC-DMIS performs the test operation when you click the **Test** button.

Extracting a Laser Auto Surface Point from a Mesh

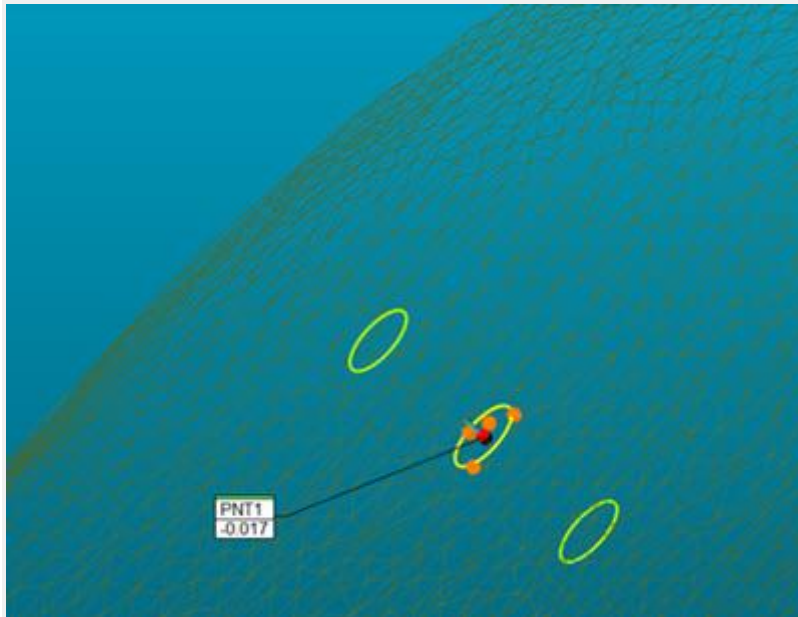
You can extract a Laser Auto Surface Point from a mesh data object with the **Laser Auto Surface Point** dialog box.

When you extract the Laser Auto Surface Point from a Mesh data object, all the triangle vertices inside the extraction zone, defined by the horizontal and vertical clipping, are first considered. To view the points that fall within the extraction zone, click the **Show/Hide segregated points** button () on the **Laser Scan Properties** tab.



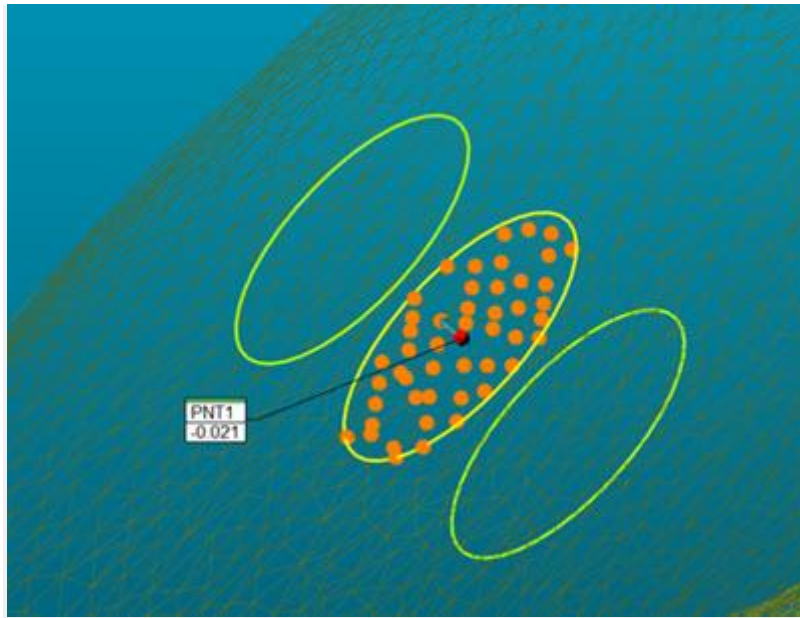
To get a more accurate result on a curved surface when you extract an Auto Surface Point from a Mesh, use a smaller horizontal clipping zone to limit the points (vertices) used to calculate the measured value.

For example, when using a small clipping zone, points close to the nominal location are used to calculate the deviation, resulting in a more accurate measurement on a curved surface:



Surface Point with small (0.25mm) horizontal clipping

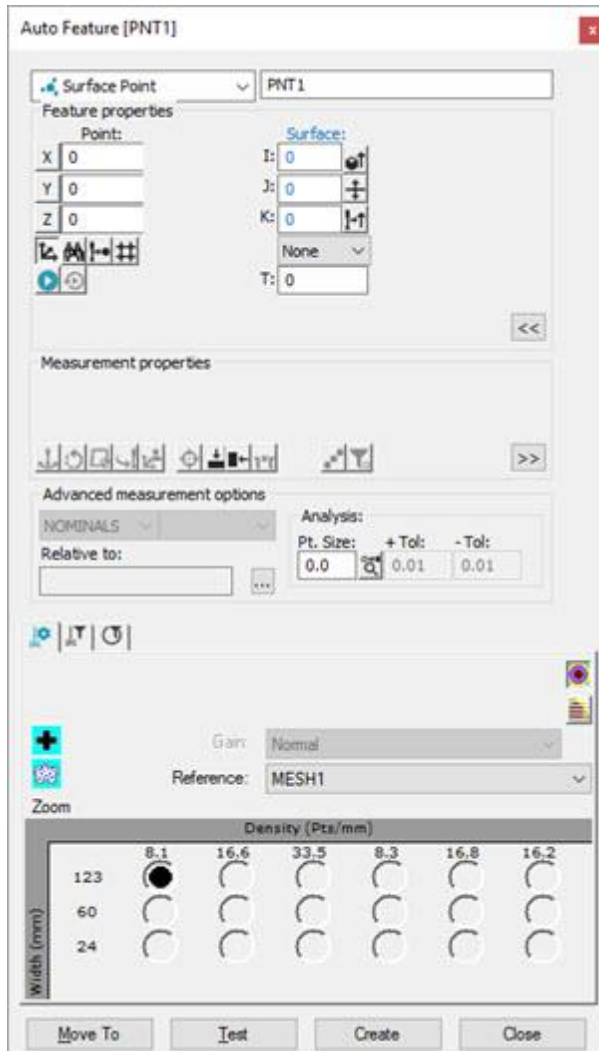
However, if a larger horizontal clipping zone is used, more points are used to calculate the deviation. This should be avoided when measuring points on a curved surface.



Surface Point with larger (1.0mm) horizontal clipping zone

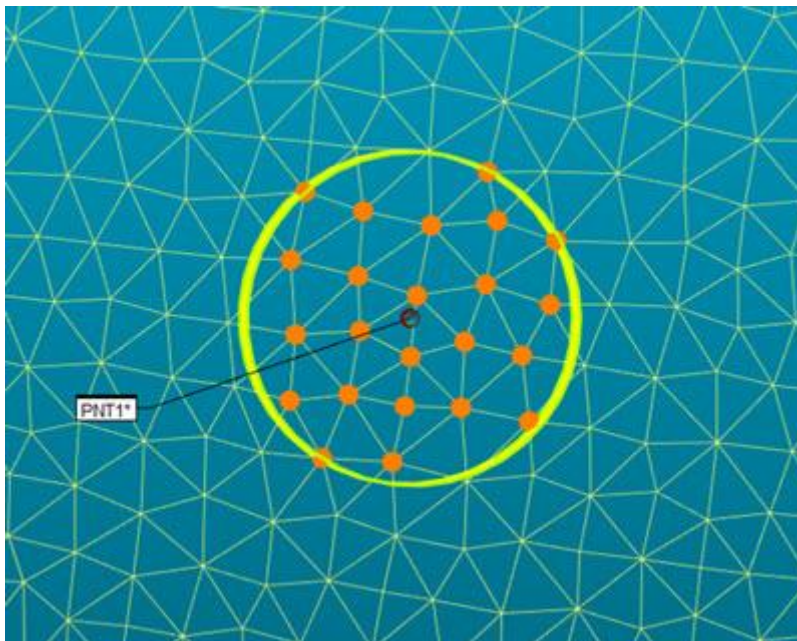
To extract a Surface Point from an existing Mesh:

1. Click the **Surface** menu option (**Insert | Feature | Auto | Point**). The **Auto Feature** dialog box appears. If the advanced options don't appear in the dialog box, click the **Show advanced measurement options** button.



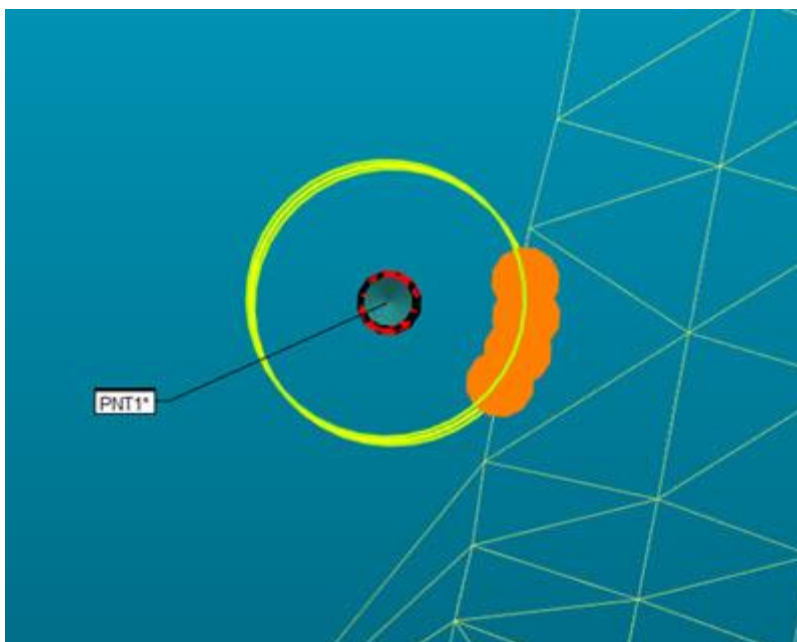
Auto Feature dialog box for Surface Point with the advanced measurement options

2. Select the **Mesh** reference for the surface point from the **Reference** list.
3. From the Graphic Display window, click the CAD to select the point nominal location and vector.
4. Click the **Show/Hide segregated points** button to view the points which fall within the extraction zone.



Example of the extracted points that fall within the extraction zone

If the number of vertices within the zone is less than three, then the clipping zone will intersect the Mesh and use the intersection points for the Auto Surface Point feature measurement.



Example of the extracted points that fall within the extraction zone with less than three vertices

5. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan Properties**, **Laser Filtering Properties**, and **Laser Clipping Region Properties** tabs to enter the information.



WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

6. If desired, click the **Test** button to test the feature.



WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

7. Click **Create** and then **Close**.

Creating Auto Features with a Laser Sensor

With PC-DMIS Laser, you can use your laser sensor to create these auto features:

- Laser Surface Point
- Laser Edge Point
- Laser Plane
- Laser Circle
- Laser Slot
- Laser Flush and Gap
- Laser Polygon
- Laser Cylinder
- Laser Cone
- Laser Sphere



This topic only discusses auto features as they pertain to laser sensor operations. For detailed information on auto features, see the "Creating Auto Features" chapter of the PC-DMIS Core documentation.

Implementation of QuickFeatures in PC-DMIS Laser

In order to cleanly implement the QuickFeature functionality, rules must be applied when switching between certain feature types that have the Inner/Outer options (Laser Circle, Laser Round Slot, Laser Square Slot, Laser Cylinder, Laser Cone, and Laser Sphere, for example).



This functionality is not available for Flush and Gap features since the hovering mouse functionality is not available for this feature type.

Since the **Inner** option enables LEAST_SQR and MAX_INSC and the **Outer** option enables LEAST_SQR and MIN_CIRCSC, the following rules apply:

- Whenever the **Inner/Outer** option that is selected in the dialog box as the default matches the Inner/Outer information that comes from the CAD quick selection, the best fitting algorithm default is kept in the created feature.
- When the **Inner/Outer** option that is selected in the dialog box as the default doesn't match the Inner/Outer information that comes from the CAD quick selection, the best fitting algorithm default is kept in the created feature only if LEAST_SQR was set as a default. In all the other cases, the created feature will have the Inner/Outer information coming from the CAD and the best fitting algorithm option set to LEAST_SQR.

For example, if you set Outer Circle as the default and MIN_CIRCSC as the best fitting algorithm, and then you quick select an inner circle, you'll get an Inner Circle with LEAST_SQR option as result.

For more information on ways to create QuickFeatures, see the "Quick Ways to Create Auto Features" topic in the "Creating Auto Features" chapter of the PC-DMIS Core documentation.

Common Laser Auto Feature Dialog Box Options

In PC-DMIS Laser, the **Auto Feature** dialog box works alongside the Probe Toolbox to create a complete laser auto feature command. To edit an auto feature, you can use the Edit window and modify the command there, or you can change parameters inside the **Auto Feature** dialog box and the Probe Toolbox. For information on the toolbox, see "Using the Probe Toolbox in PC-DMIS Laser".

The following **Auto Feature** dialog box options are common to all of the supported Laser Auto Feature types and are discussed briefly for each dialog box area.

- Feature Properties Area
- Measurement Properties Area
- Advanced Measurement Options Area
- Command Buttons
- Directly Measured Laser Auto Features

For additional information, see "The Auto Feature Dialog Box" in the "Creating Auto Features" chapter in the PC-DMIS Core documentation.


Options used for specific auto features are discussed in their respective sections.


Feature Properties Area


XYZ Center or Point - These boxes show the feature's XYZ center or point location in part coordinates.


IJK Surface, Edge, Slot, or Gap Dir (Vector) - These boxes allow you to set the surface normal vector, edge vector, slot vector, or gap direction of the feature.

IJK Angle Vector - These boxes allow you to define the feature's secondary vector. This helps control the orientation of the feature.

 **Polar/Cartesian** - This button toggles the display between Polar and Cartesian modes.


 **Find Nearest CAD** - When you select an axis (X, Y, or Z) from one of the **Center** boxes and click this button, PC-DMIS finds the closest CAD element in the Graphic Display window to that axis.

 **Point Read From Machine** - When you click this button, PC-DMIS uses the XYZ location of the machine for the feature's XYZ coordinates.


 **Find Vector(s)** - This button pierces all surfaces along the XYZ point and IJK vector looking for the closest point. The software displays the surface normal vector as the IJK NOM VEC but the XYZ values do not change.



This option is only available for Surface and Edge Point features.


 **Flip Vector** - This button flips the surface normal vector. For example, 0,0,1 would flip to 0,0,-1.


Thickness - This field (**T**) applies a thickness to the feature. You can specify whether to use actual or theoretical values and then enter the value for the thickness.

 **Swap Vectors** - Click this button to switch the current edge vector and surface vector with each other.



This option is only available for Edge Point features.

 **Measure Now** - This button determines whether or not PC-DMIS measures the feature when you click **Create**.

 **Re-Measure** - This button determines whether or not PC-DMIS automatically re-measures the feature a second time once you measure the feature. PC-DMIS uses the measured values from the first measurement as the target locations for the second measurement.




This is only available for Circle, Cylinder, Square Slot, Round Slot, and Notch features, and you must be in DCC Mode.


Measurement Properties Area


For information about the specific parameters configured in this section, see the following topics:

- Edge Point Specific Parameters
- Plane Specific Parameters

- Circle Specific Parameters
- Slot Specific Parameters
- Flush and Gap Specific Parameters
- Cylinder Specific Parameters
- Sphere Specific Parameters

 **Auto Wrist** - This button causes the probe orientation to move to a vector that closely corresponds to the surface vector of the Auto Feature.

 **View Normal** - Click this button to orient the CAD so that you look down on the feature.


 **View Perpendicular** - Click this button to orient the CAD so that you look at the side of the feature.

Advanced Measurement Options Area

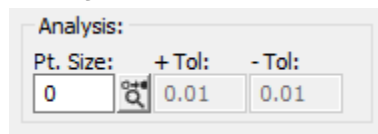
Best Fit Math Type

A Laser Auto Feature Circle also allows you to define the Best Fit Math Type. For details, see "Best Fit Type for Circle" in the "Constructing New Features from Existing Features" chapter in the PC-DMIS Core documentation. Valid options for the Perceptron system are Maximum Inscribed, Minimum Circumscribed, and Least Squares.

Relative to


This option allows you to keep the relative position and orientation between a given feature (or features) and the auto feature. Click the  button to open the **Relative Feature** dialog box to select the feature or features to which the auto feature is relative. Multiple features can be defined for each axis (XYZ) relative to your auto feature.

Analysis Area



The **Analysis** area allows you to determine how each measured hit/point is displayed.

Pt. Size - Determines how big the measured points are drawn in the **CAD** tab. This value specifies the diameter in the current units (mm or inch).

 **Graphic Analysis** button - When this button is switched on, PC-DMIS does a tolerance check on each point (how far from the calculated actual feature it is) and draws it in the appropriate color based on the current defined color range for the dimension.

+ Tol - This option provides the positive tolerance from the nominal. It is specified in the current measurement routine's units. Points that are greater than this value from the nominal are colored based on the standard PC-DMIS positive-tolerance color.

- Tol - This option provides the negative tolerance from the nominal. It is specified in the current measurement routine's units. Points that are less than this value from the nominal are colored based on the standard PC-DMIS negative-tolerance color.

For information on editing dimension colors for the positive and negative tolerances, see the "Editing Dimension Colors" topic in the "Editing the CAD Display" chapter in the PC-DMIS Core documentation.

Command Buttons

>> - This button expands the **Auto Feature** dialog box to show additional, more advanced, auto feature options.

<< - This button hides the more complex features of the **Auto Feature** dialog box.

Move To - This button moves the field of view of the Graphic Display window and centers it on the features XYZ location. If the feature is comprised of more than one point (such as a line), then clicking this button switches between the points making up the feature. For a laser slot auto feature the field of view moves to the center of the slot feature.


Test - This button tests the auto feature before PC-DMIS creates it. For laser features, the machine will scan over the feature and calculate the measured value for the feature.

Create - This button creates the auto feature and the **Auto Feature** dialog box remains open.

Close - This button closes the **Auto Feature** dialog box without creating a feature.

Directly Measured Laser Auto Features

The **Reference** parameter, found on the **Laser Scan Properties** tab of the **Laser Auto Feature** dialog box, defines the pointcloud or mesh from which PC-DMIS extracts the auto feature. If you select the **Disabled** option from the list, you can scan the feature directly. The software stores the scanned stripes in an internal COP. This is referred to as a "Directly Measured Laser Auto Feature".

When you run PC-DMIS in either Online or Offline mode, the internal scanned stripes are visible in the Graphic Display window only while the laser **Auto Feature** dialog box is open and you enable the **Show/Hide Stripes** button . When you close the dialog box, the scan stripes are no longer visible. After you create the auto feature and press F9 to edit the Directly Measured Laser Auto Feature, the stripes become visible again.



You can only use the **Disabled** parameter in DCC mode.

Online

When you run PC-DMIS in Online mode with the CMM, you can directly measure a laser auto feature. To do this, you must set the **Reference** parameter to **Disabled**.




WARNING – When you select the **Disabled** parameter with the machine online, and the **Measure Now** button is selected, the machine moves to the feature and begins the scan using the selected settings as soon as you click the **Create** or **OK** button.

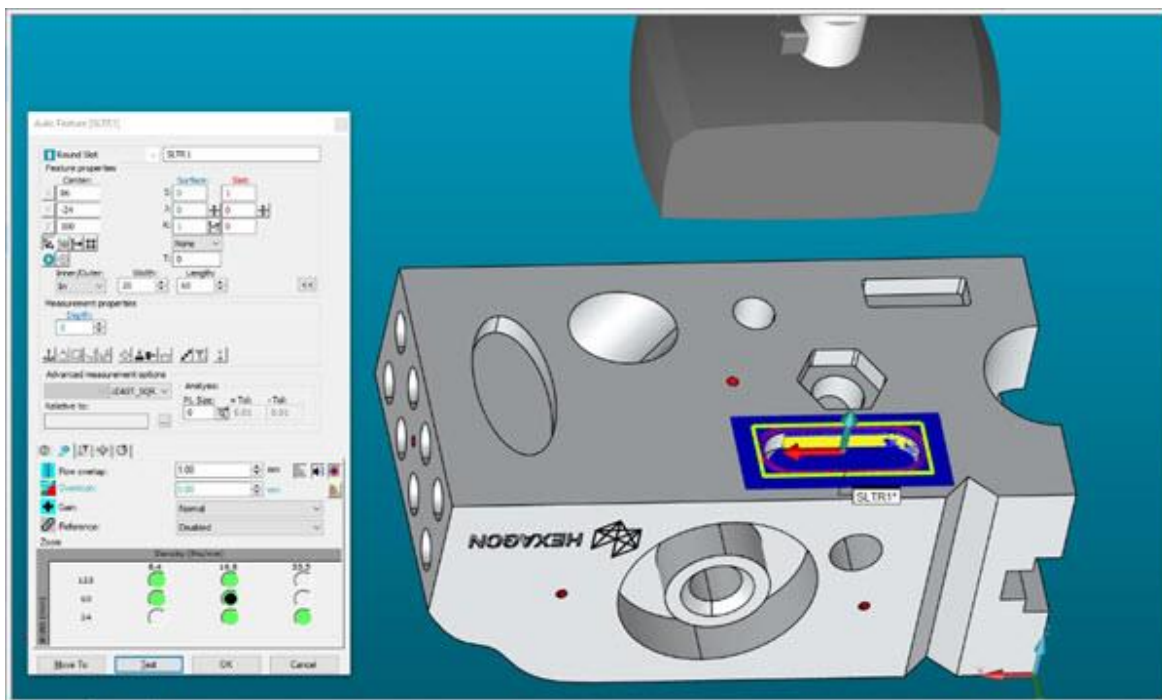
When online with the CMM and you click the **Test** button, the machine moves to the feature and begins the scan.

Offline

When you run PC-DMIS in Offline mode, you can simulate a Directly Measured Laser Auto Feature, check the scan settings, and adjust them as needed without running the machine.

To simulate a Directly Measured Laser Auto Feature:

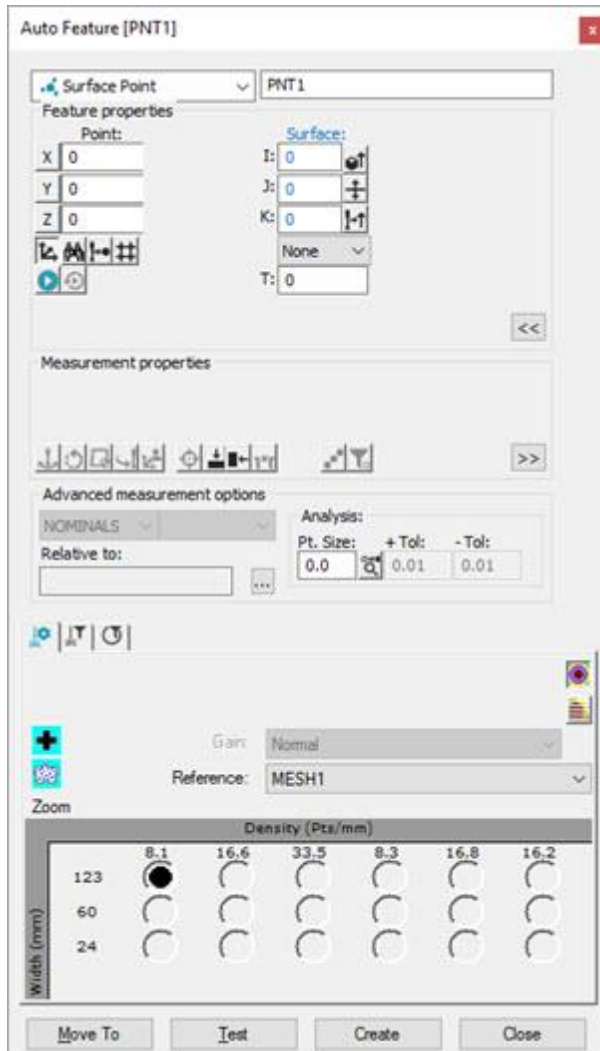
1. Start PC-DMIS in Offline mode.
2. Select the **DCC Mode** option from the **Probe Mode** toolbar (**View | Toolbars | Probe Mode**).
3. Open the **Auto Feature** dialog box (**Insert | Feature | Auto**) and select the feature you want to create.
4. Select the **Disabled** option from the **Reference** list.
5. Click the **Show/Hide Stripes** button  to view the simulated stripes.
6. Click the **Test** button to preview the internal scan stripes by projecting them as simulated scan stripes on the CAD model.



Example of a Directly Measured Laser Auto Feature with simulated scan lines displayed offline


Laser Surface Point

Three methods exist to calculate the laser surface point: Planar, Spherical, and Extended Surface Point. For more information on these methods, see "Calculation Methods".

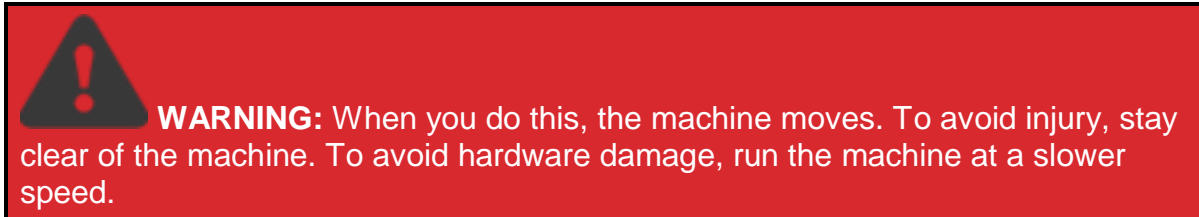


Auto Feature dialog box - Surface Point

To measure a laser surface point with a laser sensor:

1. Access the **Auto Feature** dialog box (**Insert | Feature | Auto | Point**), and click **Surface Point**.
2. Do one of the following:
 - Click the CAD in the Graphic Display window to give the point a location and vector. Manually enter any remaining information.
 - From the Graphic Display window, use the **Laser** tab to move the machine to the point location. Next, from the **Feature Properties** area, click the **Read Point from Position** button (). Manually enter any remaining information.
 - Manually enter the theoretical information for X, Y, Z, I, J, K, and so on.

3. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan Properties**, **Laser Filtering Properties**, and **Laser Clipping Region Properties** tabs to enter the information.
4. If desired, click the **Test** button to test the feature.



5. Click **Create** and then **Close**.

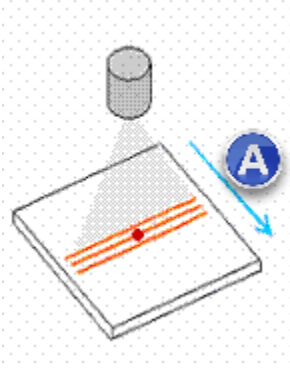
Surface Point Command Mode Text

The Surface Point command inside the Edit window's Command Mode looks like this:

```
PNT1 =FEAT/LASER/SURFACE POINT,CARTESIAN
  THEO/<1.895,1.91,1>,<0,0,1>
  ACTL/<1.895,1.91,1>,<0,0,1>
  TARG/<1.895,1.91,1>,<0,0,1>
  SHOW FEATURE PARAMETERS=YES
    SURFACE=THEO_THICKNESS,1
    MEASURE MODE=NOMINALS
    RMEAS=NONE,NONE,NONE
    AUTO WRIST=NO
    GRAPHICAL ANALYSIS=NO
    FEATURE LOCATOR=NO,NO,""
  SHOW_LASER_PARAMETERS=YES
    POINT CLOUD ID=DISABLED
    SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
    FILTER=NONE
```

Auto Surface Point Path

The direction of the path is determined based on the stripe.



Path direction of scan for surface point

(A) - Scan motion

Calculation Methods

There are three methods for calculating laser surface points:

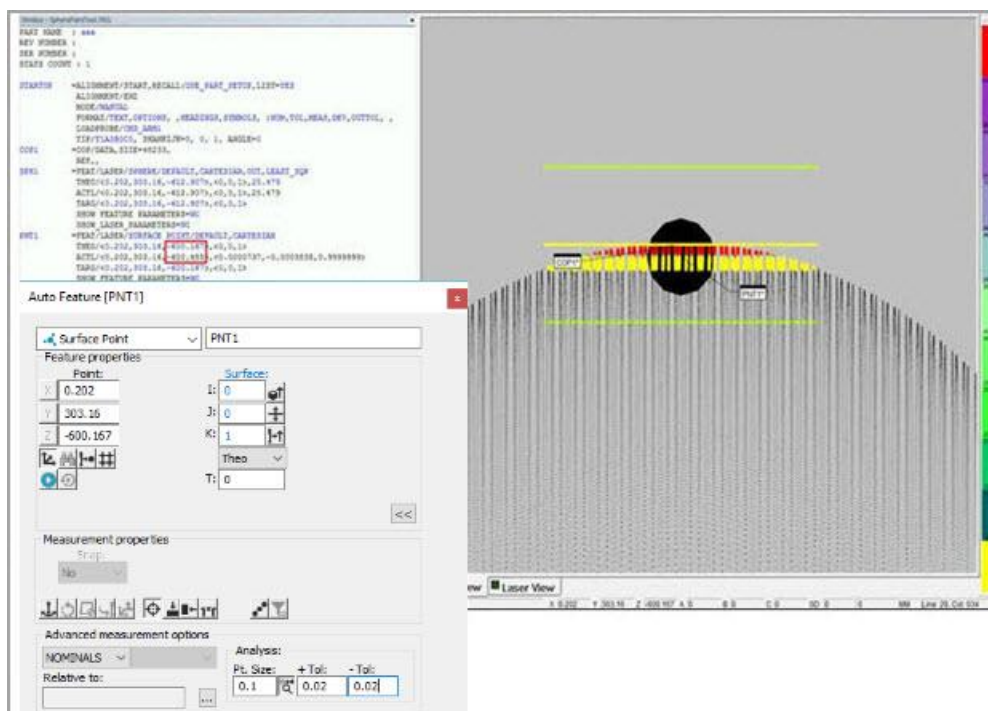
- Planar
- Spherical
- Extended Surface Point

Changing the Calculation Method

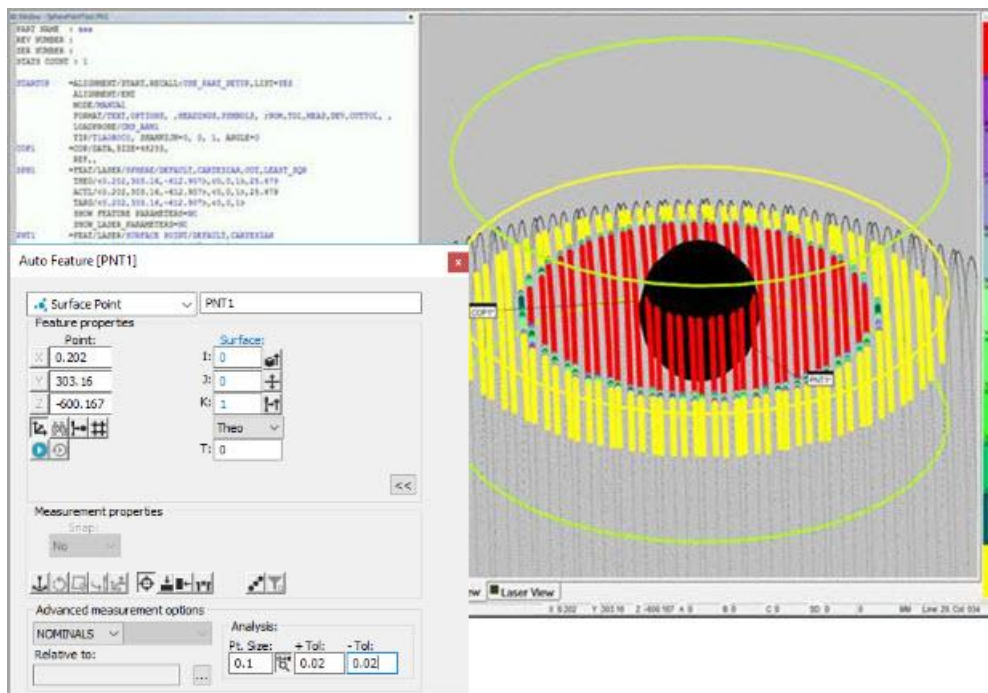
To change the calculation method, modify the `SurfacePointType` registry entry located in the **AutoFeatures** section of the PC-DMIS Settings Editor. For information on this entry, launch your PC-DMIS Settings Editor and press F1 to access its help file. For more information, see the PC-DMIS Settings Editor documentation.

Planar Surface Point Calculation Method

This method calculates the laser surface point by fitting a local plane on the scan points within the circular area defined by the horizontal and vertical clipping parameters; this is the default method. The following is an example and its details:



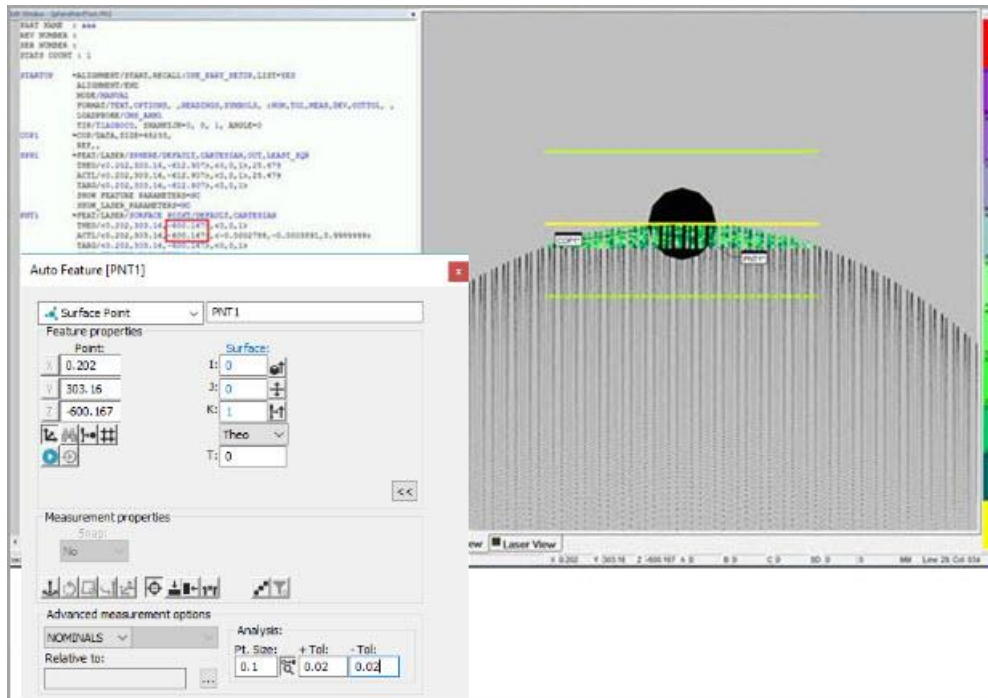
Planar Surface Point example



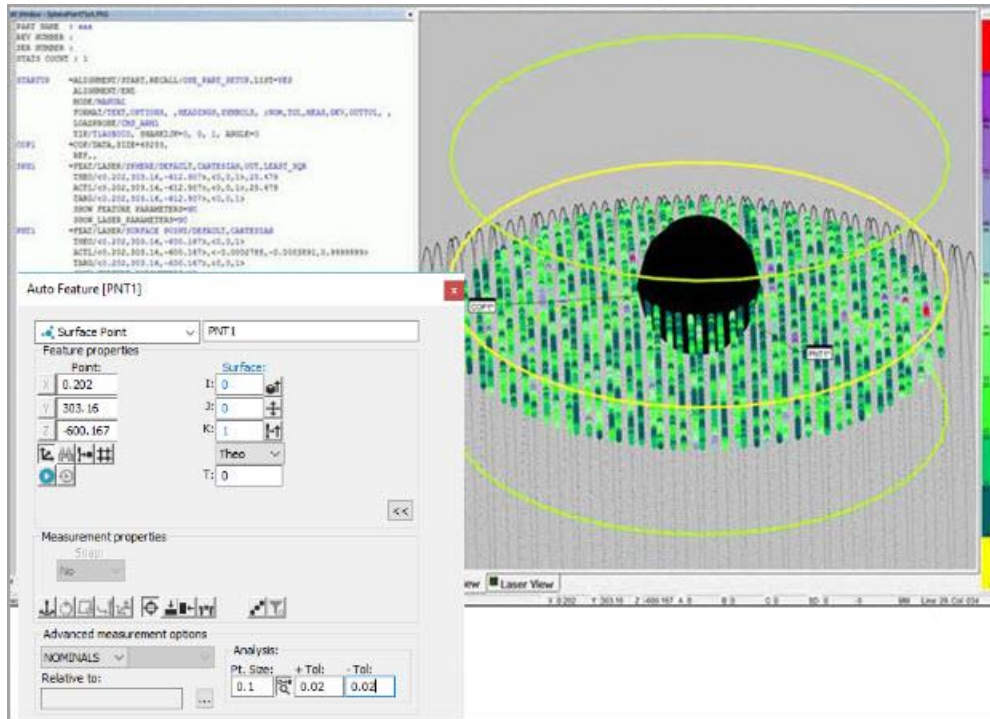
Planar surface point example - details

Spherical Surface Point Calculation Method

This method calculates the laser surface point by fitting a local sphere on the scan points within the circular area defined by the horizontal and vertical clipping parameters. Following is an example and its details:



Spherical surface point example



Spherical surface point example - details

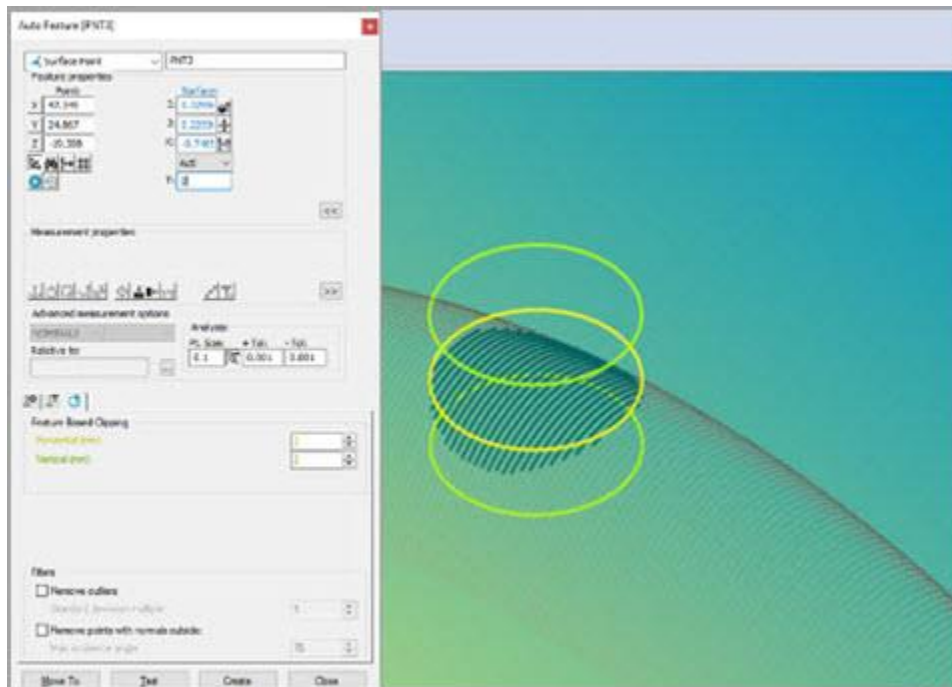
Extended Surface Point Calculation Method

This algorithm is able to calculate the Surface Point by fitting a local 2-curvatures manifold on the scan points within the circular area defined by the horizontal and vertical clipping parameters.

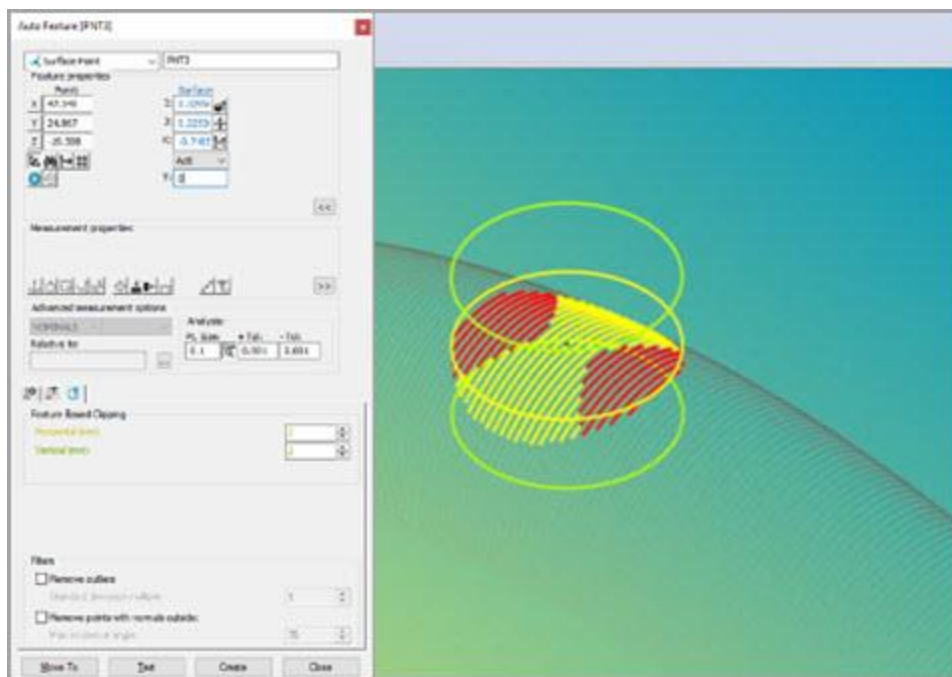
This method is especially useful when you need to calculate Surface Points on fillet surfaces.

The pictures below show the comparative results of algorithms applied to a point on a 2-curvatures filleted surface for the following:

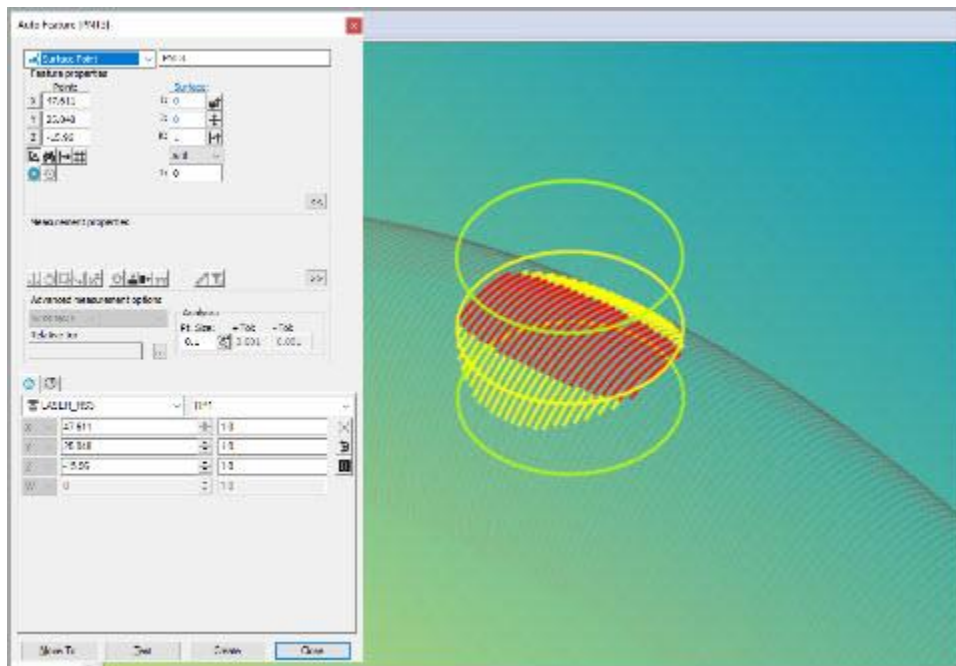
Extended Surface Point, Extended Spherical Surface Point, and Extended Planar Surface



Extended Surface Point details



Extended Spherical Surface Point details



Extended Planar Surface Point details

If a log file is enabled, additional results from the calculation of Extended Surface Points are available in the file "WaiFE_Debug.txt" found in the C:\ProgramData\Hexagon\PC-DMIS\ (PC-DMIS version) \NCSensorsLogs\FeatureExtractor folder:

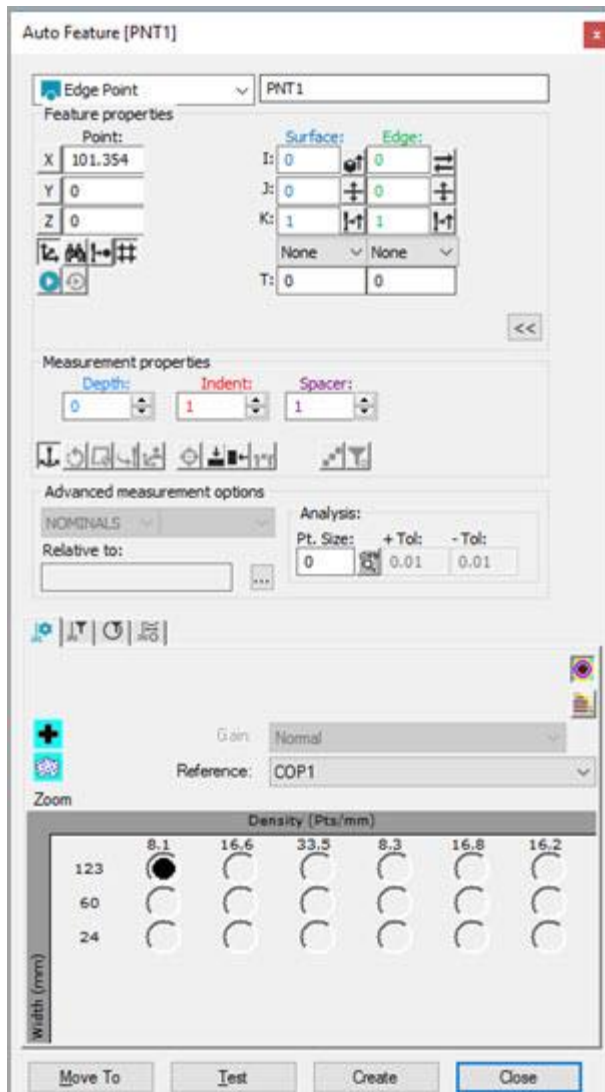
```

----- SURFACE POINT - begin: -----
TYPE: EXTENDED
ACTUAL LOCAL CURVATURES: -0.028572 : -0.200001
ACTUAL SURFACE POINT: i= 47.141291, j= 24.067065, k= -10.597570
ACTUAL SURFACE VECTOR: i= 0.553249557, j= 0.232507664, k= -0.799909441
ACTUAL PRINCIPAL CURVATURE VECTOR: i= -0.832996099, j= 0.147852741, k= -0.533157637
ACTUAL SECONDARY CURVATURE VECTOR: i= -0.005694434, j= 0.961290671, k= 0.275477440
STANDARD DEVIATION: 0.000001
CONDITION INDICATOR: 0.810149
----- SURFACE POINT - end -----

```


The Condition Indicator value is a number from 0 (zero) to 1 inclusive, indicating the quality of the points distribution. 0 (zero) indicates a poor distribution, and 1 indicates a good distribution. Generally, a value greater than 0.4 is considered acceptable.

Laser Edge Point

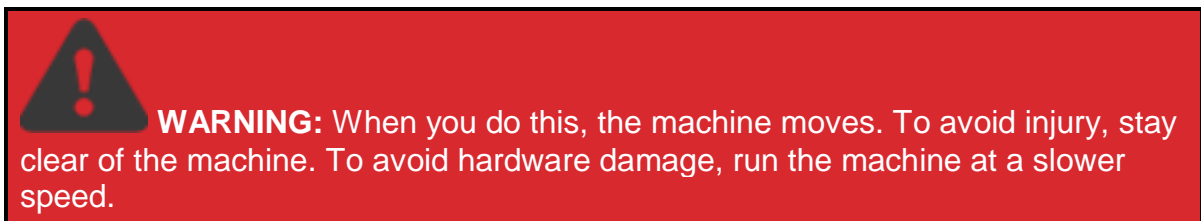


Auto Feature dialog box - Edge Point

To measure an edge point with a laser sensor:

1. Access the **Auto Features** dialog box, and select **Edge Point**.
2. Do one of the following:
 - Perform clicks on the CAD to give the point a location and vector. Manually enter any remaining information.
 - From the Graphic Display window, use the **Laser** tab to move the machine to the point location. Next, from the **Feature Properties** area, click the **Read Point from Position** button (). Manually enter any remaining information.

- Manually enter all of the theoretical information for X, Y, Z, I, J, K, and other parameters.
3. From the **Contact Path Properties** tab of the **Probe Toolbox**, specify values for **Depth**, **Indent**, and **Spacer**. PC-DMIS displays a corresponding graphical visualization of the change in the Graphic Display window.
 4. Enter the necessary information in the other **Probe Toolbox** tabs. Cycle through the **Laser Scan Properties**, **Laser Filtering Properties**, **Laser Clipping Region Properties**, **Feature Extraction**, and **Laser AF Multiple Creation** tabs to enter the information.
 5. If desired, click the **Test** button to test the feature.



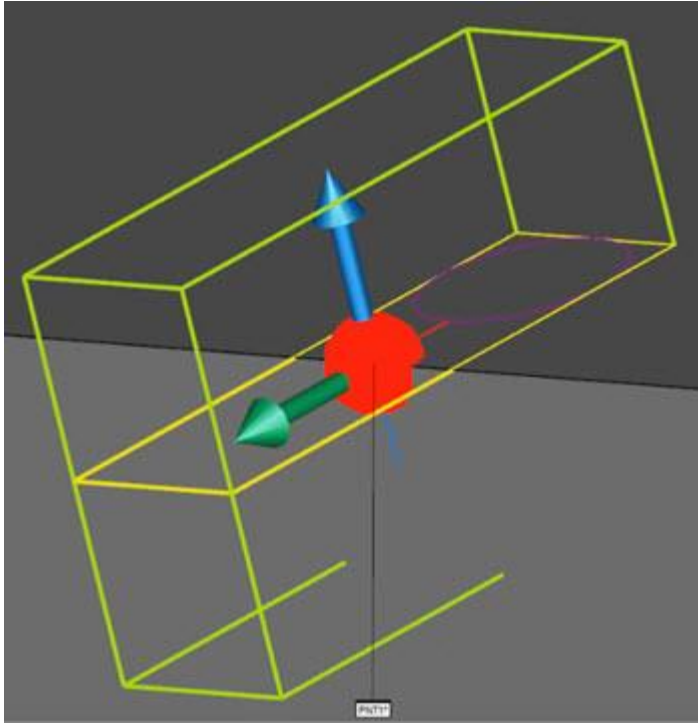
6. Click **Create** and then **Close**.

Edge Point Specific Parameters

Depth: This defines the depth to use when calculating the edge point. This corresponds to the blue graphical visualization in the Graphic Display window. A depth of 0 will cause this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. A depth of any other value will cause it to be calculated at that depth.

Spacer: This controls the size of the area PC-DMIS uses to calculate the feature normal. This corresponds to the purple graphical visualization in the Graphic Display window.

Indent: This lets you define the location of the area PC-DMIS uses to calculate the feature normal. This corresponds to the red graphical visualization in the Graphic Display window.



Sample Edge Point with Depth, Spacer, and Indent graphical visualizations used in the Graphic Display window

Notes on Graphical Analysis and Feature Extraction of Edge Points

If you don't see some graphical analysis points computed to the edge plane, consider the following:

- **Edge Line Points** - All the edge line points on the reference plane returned by the feature extractor are displayed. For analysis, the edge line points are computed using the distance (**Indent** value) from the reference plane center (center of the circular surface area defined by the **Spacer** value) to the edge line.
- **Reference Plane Points** - If the Spacer value is 0.0 then the reference plane points are not displayed. If the Space value is not 0.0 then the reference plane points are extracted from the point cloud, applying the following rules using the plane statistical data returned by the feature extractor:

- Rule 1: All points that are outside of an *imaginary cylinder* are discarded.

This cylinder is identified by using the following values:

Center = Indent Center Point

Vector = Surface Vector

Radius = Spacer

- Rule 2: All points with a distance from an *imaginary plane* greater than the maximum plane error value are discarded.

This plane is identified by using the following values:

Center = Measured Edge Point

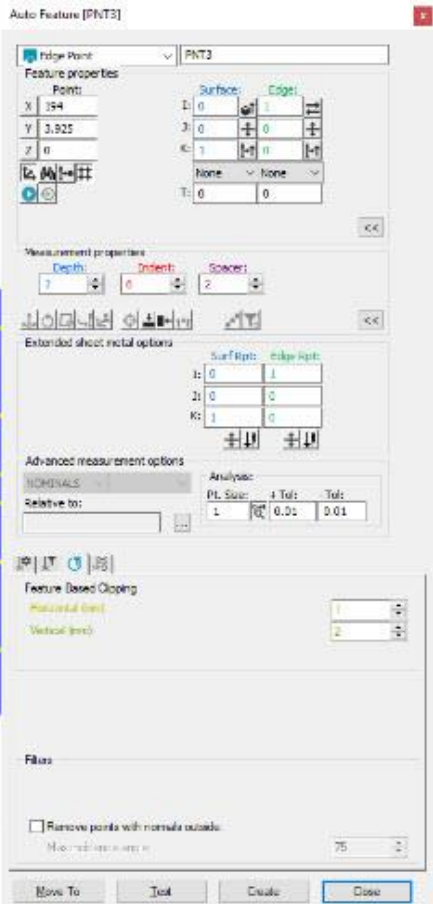
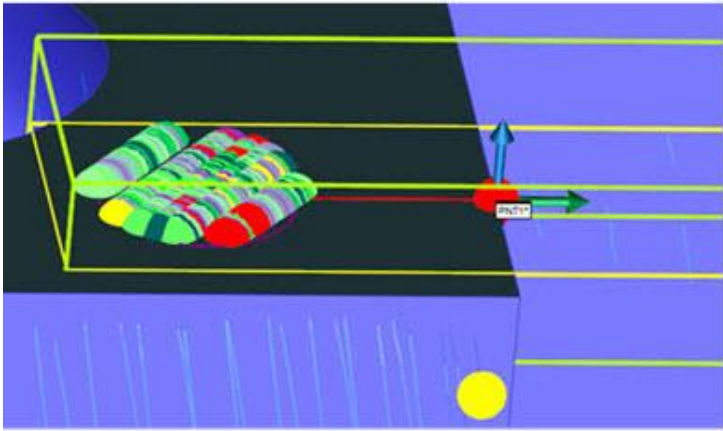
Vector = Measured Surface Vector

- Rule 3: If any remaining points are higher than the allowed number (19900), the points are uniformly reduced to the allowed value.

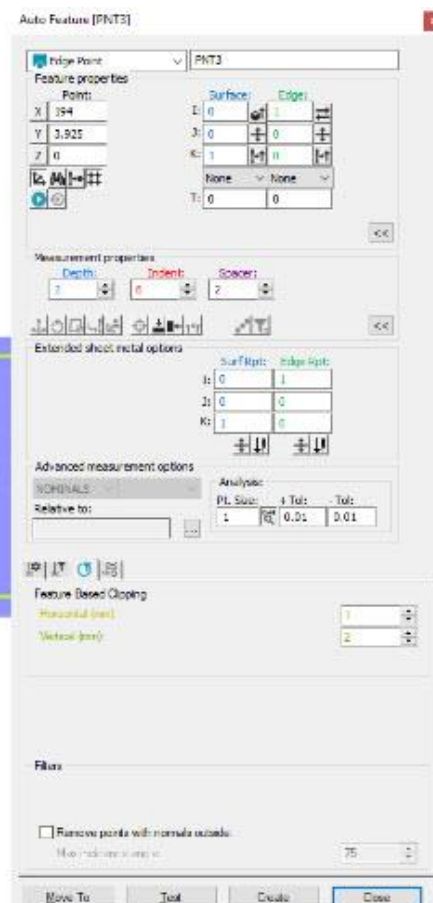
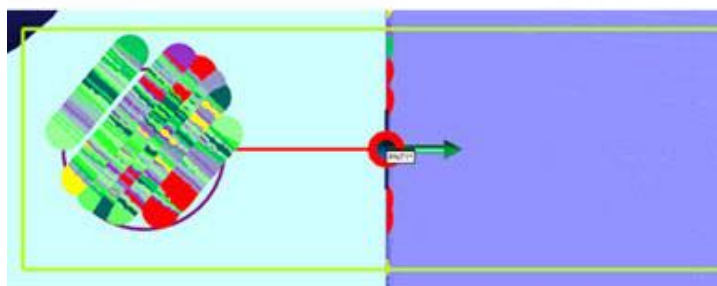
For analysis, each reference plane point is computed using the distance from the reference plane and the measured surface plane.

The following two images show the Edge Point laser graphical analysis:

- *Graphical Analysis Example - Side View*



- *Graphical Analysis Example - Top View*



Edge Point Command Mode Text

The Edge Point command inside the Edit window's Command Mode looks like this:

```
PNT2 =FEAT/LASER/EDGE POINT,CARTESIAN
      THEO/<1.895,1.91,1>,<0,1,0>,<0,0,1>
      ACTL/<1.895,1.91,1>,<0,1,0>,<0,0,1>
      TARG/<1.895,1.91,1>,<0,1,0>,<0,0,1>
      SHOW FEATURE PARAMETERS=YES
          SURFACE1=THEO_THICKNESS,1
          SURFACE2=THEO_THICKNESS,0
          MEASURE MODE=NOMINALS
          RMEAS=NONE,NONE,NONE
          AUTO WRIST=NO
          GRAPHICAL ANALYSIS=NO
          FEATURE LOCATOR=NO,NO,""
```

SHOW_LASER_PARAMETERS=YES

POINT_CLOUD_ID=DISABLED

SENSOR_FREQUENCY=25,OVERSCAN=2,EXPOSURE=18

FILTER=NONE

Laser Plane

Auto Feature [PLN1]

Plane PLN1

Feature properties

Center:

X: 101.354

Y: 0

Z: 0

Surface: Angle:

I: 0 1

J: 0 0

K: 1 0

None

T: 0

Measurement properties

Display: Width: Length:

None 0.5 1.5

Advanced measurement options

NOMINALS LEAST_SQR

Analysis:

Pt. Size: + Tol: - Tol:

0 0.01 0.01

Feature Based Clipping

Horizontal (mm): 1

Vertical (mm): 2

CAD offset: 2

Filters

Remove outliers

Standard deviation multiple: 1

Remove points with normals outside:


Max incidence angle: 75

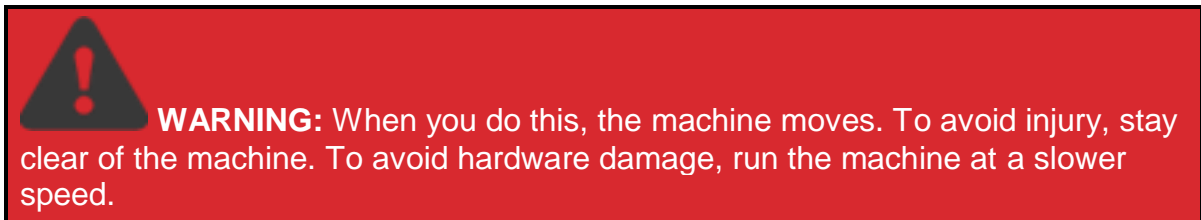
Move To Test Create Close

Auto Feature dialog box - Plane

To create an auto plane using a laser sensor:

1. Access the **Auto Features** dialog box (**Insert | Feature | Auto**), and select **Plane**.
2. Do one of the following:

- Click on the CAD to give the plane a location and vector. Manually enter any remaining information.
 - From the Graphic Display window, use the **Laser** tab to move the machine to the center of the plane location. Click the **Read Point from Position** button (). Manually enter any remaining information such as the display, width, length, and other parameters.
 - Manually enter the theoretical information for X, Y, Z, I, J, K, display, width, length, and other parameters.
3. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping** properties tabs to enter the information as needed.
 4. If desired, click the **Test** button to test the feature.



5. Click **Create** and then **Close**.

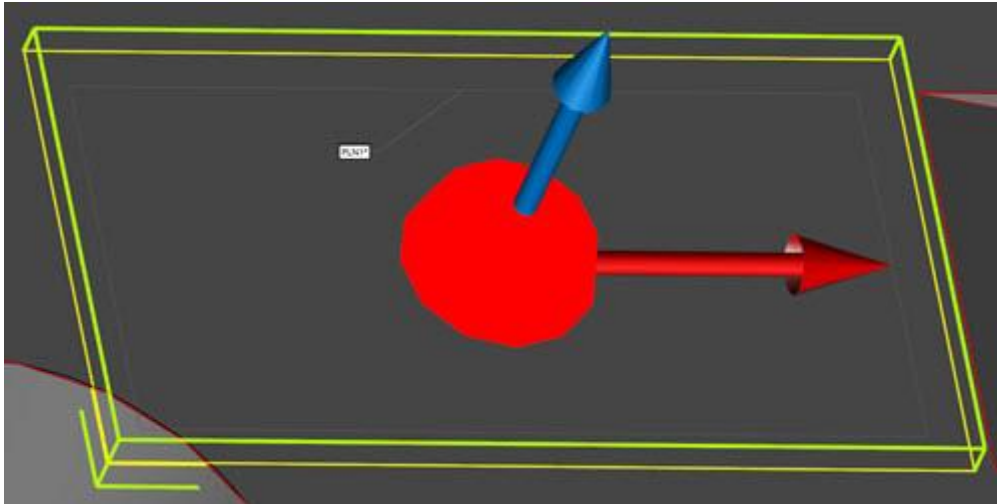
Plane Specific Parameters

Width: The value in this box determines the width of the measurement area of the plane.

Length: The value in this box determines the length of the measurement area of the plane.

Display: This list lets you choose how to present the plane inside the Graphic Display window. You can choose **NONE** or **TRIANGLE** or **OUTLINE**:

- If you choose **NONE**, the plane is not displayed.
- If you choose **TRIANGLE**, PC-DMIS displays the plane with a triangle symbol at the very center of the plane.
- If you choose **OUTLINE**, PC-DMIS displays an outline of the edges of the plane.



Sample Plane in the Graphic Display window with:

Outline display (Gray dotted line)

Overscan display (Yellow rectangle)

Vertical Clipping (Green rectangular box)

Plane Command Mode Text

The Plane command inside the Edit window's Command Mode looks like this:

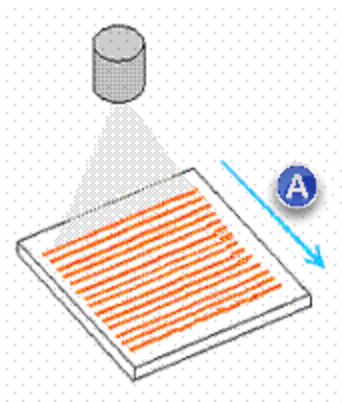
```
PNT1 =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN,TRIANGLE
THEO/<-19.594,3.822,0>,<-1,0,0>,<0,0,1>
ACTL/<-19.594,3.822,0>,<-1,0,0>,<0,0,1>
TARG/<-19.594,3.822,0>,<-1,0,0>,<0,0,1>
DEPTH=4
INDENT=7
SPACER=1
SHOW FEATURE PARAMETERS=YES
    SURFACE1=THEO_THICKNESS,0
    SURFACE2=THEO_THICKNESS,0
    RMEAS=NONE,NONE,NONE
    AUTO WRIST=NO
    GRAPHICAL ANALYSIS=NO
SHOW_LASER_PARAMETERS=YES
    POINT CLOUD ID=COP2
```

HORIZONTAL CLIPPING=9, VERTICAL CLIPPING=9

Auto Plane Paths

PC-DMIS provides two different paths for a plane. It automatically chooses the appropriate path based on the diameter and the size of the usable portion of the laser stripe. For auto planes, PC-DMIS always scans perpendicular to the direction of the stripe.

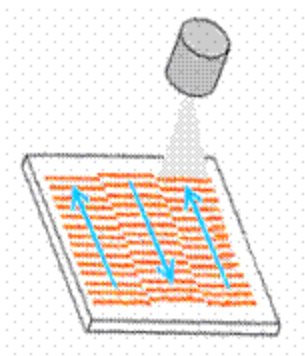
Path 1: Smaller Width



Planes with a width smaller than the usable portion of the stripe

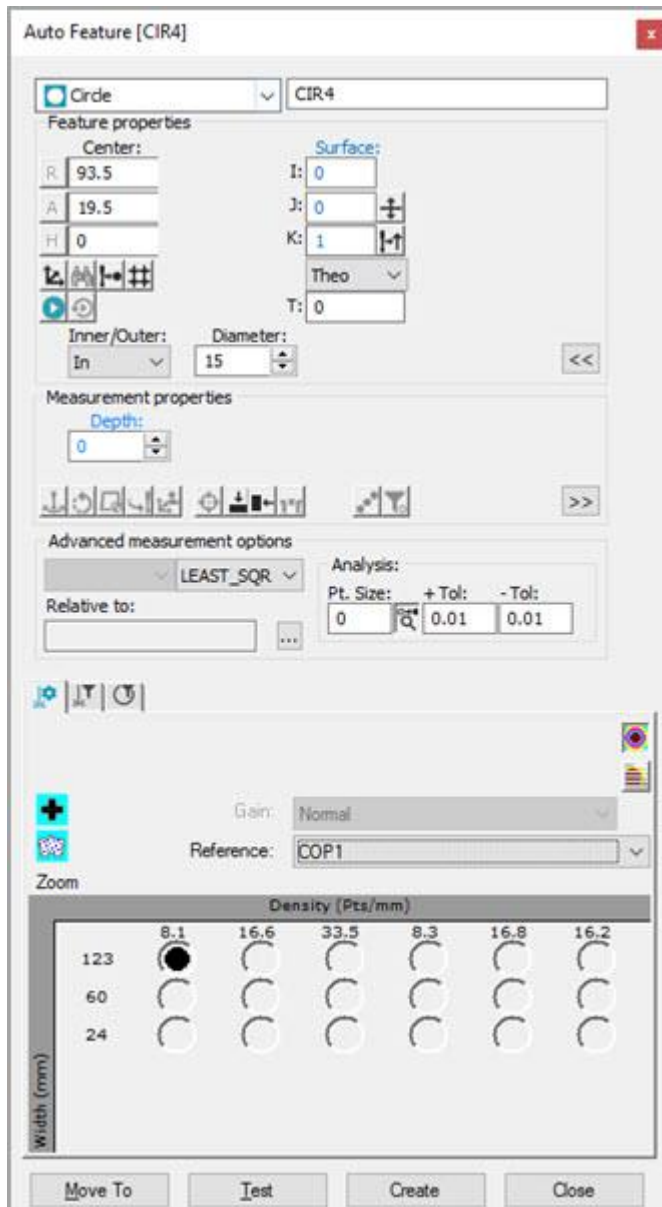
(A) - Scan motion

Path 2: Larger Width



Planes with a width larger than the usable portion of the stripe


Laser Circle



Auto Feature dialog box - Circle

To create a laser auto circle:

1. Access the **Auto Features** dialog box, and select **Circle**.
2. Do one of the following:
 - Perform clicks on the CAD to give the circle a location and vector. Manually enter any remaining information.

- From the Graphic Display window, use the **Laser** tab to move the machine to the circle location. Next, from the **Feature Properties** area, click **Read Point from Machine** . Manually enter any remaining information, such as the diameter, depth, and other parameters.
 - Manually enter all of the theoretical information for X, Y, Z, I, J, K, diameter, depth, and other parameters.
3. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan Properties**, **Laser Filtering Properties**, and **Laser Clipping Properties** tabs to enter the information.
 4. If desired, click the **Test** button to test the feature.



WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

5. Click the **Create** button and then click **Close**.



Currently, you can only measure inner circles (holes) with laser sensors.

Circle Specific Parameters

Diameter - This box specifies the circle's diameter. When you select a circle with the mouse in the Graphic Display window, PC-DMIS automatically places the circle's diameter from the CAD model in this box.

Depth - This parameter controls what data PC-DMIS uses to calculate the feature characteristics. You can use the depth value to eliminate data on a chamfer or some other transitional portion of the feature that you don't want in the feature calculation. Specifying a positive value tells PC-DMIS where along the feature PC-DMIS looks to calculate the feature characteristics. A depth of 0 will cause this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. A depth of any other value causes it to be calculated at that depth. Due to hardware limitations, for this feature type if you do use a depth value higher than 0, you must use a minimum of 0.3 mm (.01181 inches).

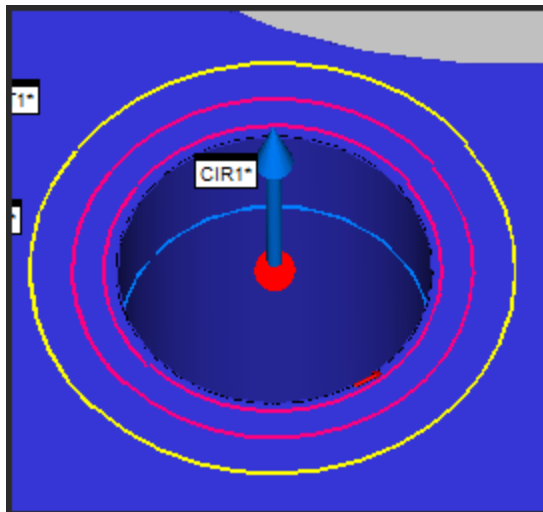


The depth defaults to zero. This is the default value for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth, resulting in a feature calculation error from the feature extraction module.

For example, a depth of 3 indicates that you want to use all data 3 mm (or inches, depending on the units of the measurement routine) and above for the calculation. If you specify 0, this indicates that you want to use all available data for the calculation. For thin walled features, a 0 value may be valid; but for parts with any depth to them, you will probably need to specify a depth to get accurate results.



Even if you specify a depth greater than zero, the measured results are always projected into the plane where the feature resides.



Sample Circle in the Graphic Display window showing:
Depth (blue circle)
Ring Band (pink circles)
Overscan (yellow circle)

Auto Circle Command Mode Text

The auto circle command inside the Edit window's Command Mode looks like this:

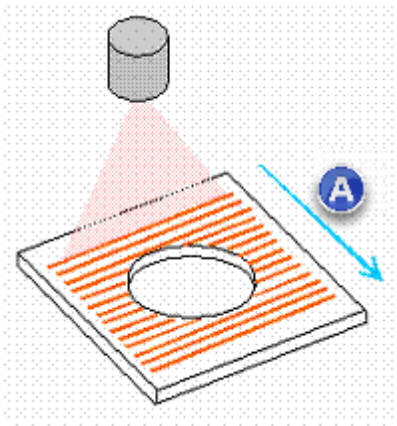
```
CIR2 =FEAT/LASER/CIRCLE,CARTESIAN
      THEO/<1.895,1.91,1>,<0,0,1>,1.895
```

```
ACTL/<1.895,1.91,1>,<0,0,1>,1.895  
TARG/<1.895,1.91,1>,<0,0,1>  
ANGLE VEC=<0,0,1>  
DEPTH=3  
SHOW FEATURE PARAMETERS=YES  
    MEASURE MODE=NOMINALS  
    RMEAS=NONE,NONE,NONE  
    AUTO WRIST=NO  
    GRAPHICAL ANALYSIS=NO  
    FEATURE LOCATOR=NO,NO,""  
SHOW_LASER_PARAMETERS=YES  
    POINT CLOUD ID=DISABLED  
    SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18  
    FILTER=NONE
```

Auto Circle Paths

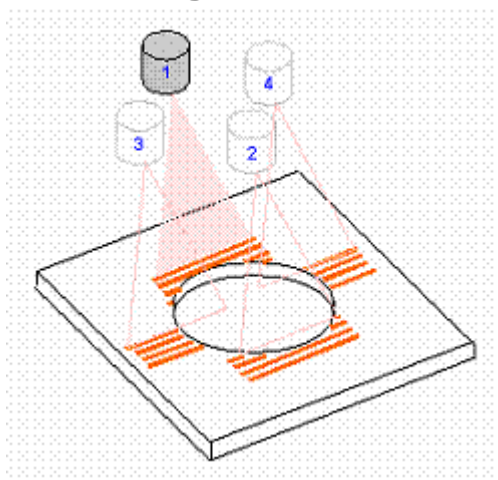
PC-DMIS provides two different paths for a circle. It automatically chooses the appropriate path based on the diameter and the size of the usable portion of the laser stripe. For auto circles, PC-DMIS always scans perpendicular to the direction of the stripe.

Path 1: Smaller Diameter



Circles with a diameter smaller than the usable portion of the stripe

(A) - Scan motion

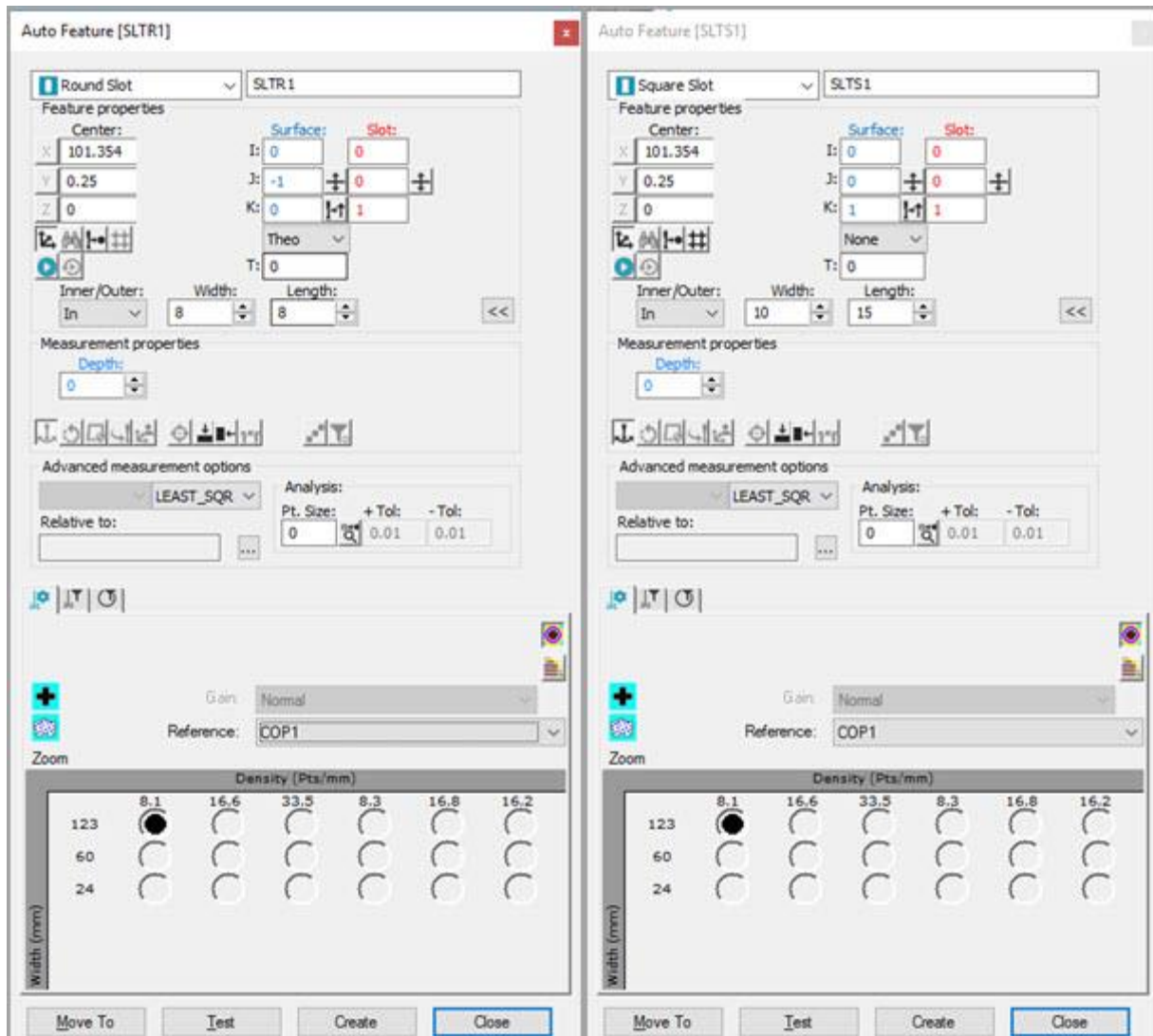
Path 2: Larger Diameter

Circles with a diameter larger than the usable portion of the stripe



The method for measuring circles with a larger diameter has been improved to measure the four passes at 1:30, 4:30, 7:30, and 10:30 rather than 12:00, 3:00, 6:00, and 9:00 as depicted in the image.

Laser Slot



Auto Feature dialog box - Round Slot (left) and Square Slot (right)


To measure a slot with a laser sensor:

1. Access the **Auto Features** dialog box (**Insert | Feature | Auto**), and select **Round Slot** or **Square Slot**.
2. Do one of the following:
 - a. Click on the CAD to collect the X, Y, Z, I, J, K information:

For Round Slots:

1. Click one of the rounded edges of the slot in the Graphic Display window. PC-DMIS prompts you to click two more times on the same rounded edge.
2. Click twice on this edge. PC-DMIS then prompts you to click on the other rounded edge.
3. Click on the other rounded edge. PC-DMIS prompts you to click two more times on that same rounded edge.
4. Click twice on the second rounded edge. PC-DMIS establishes the orientation of the round slot.

For Square Slots:

1. Click on one of the long edges of the slot in the Graphic Display window. PC-DMIS prompts you to click another location on the same edge to determine the direction.
 2. Click on a second edge, 90 degrees from the first.
 3. Click on the third edge, 90 degrees from the second. This sets the width.
 4. Click on the fourth edge and final edge. This sets the length.
- b. From the Graphic Display window, use the **Laser** tab, and move the machine to the slot location. Next, from the **Feature Properties** area, click the **Read Point from Position** button (.
3. Manually enter the theoretical X, Y, Z, I, J, K, width, length, depth, height, and other parameters.
 4. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping** properties tabs to enter the information.
 5. If desired, click the **Test** button to test the feature.



WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

6. Click **Create** and then **Close**.

Slot Specific Parameters

Inner/Outer - This list lets you choose whether or not the slot is an Inner slot (a hole) or an Outer slot (a stud).

Width - The value in this box determines the slot's width.

Length - The value in this box determines the slot's length.

Depth - This parameter controls what data PC-DMIS uses to calculate the feature characteristics. You can use the depth value to eliminate data on a chamfer or some other transitional portion of the feature that you don't want in the feature calculation. A depth of 0 causes this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. A depth of any other value causes it to be calculated at that depth. Specifying a positive value tells PC-DMIS where along the feature PC-DMIS looks to calculate the feature characteristics. Due to hardware limitations, for this feature type if you do use a depth value higher than 0, you must use a minimum of 0.3 mm (.01181 inches).

For example, a depth of 3 indicates that you want to use all data 3 mm (or inches, depending on the units of the measurement routine) and above for the calculation. If you specify 0, this indicates that you want to use all available data for the calculation. For thin walled features, a 0 value may be valid; but for parts with any depth to them, you should specify a depth to get accurate results.

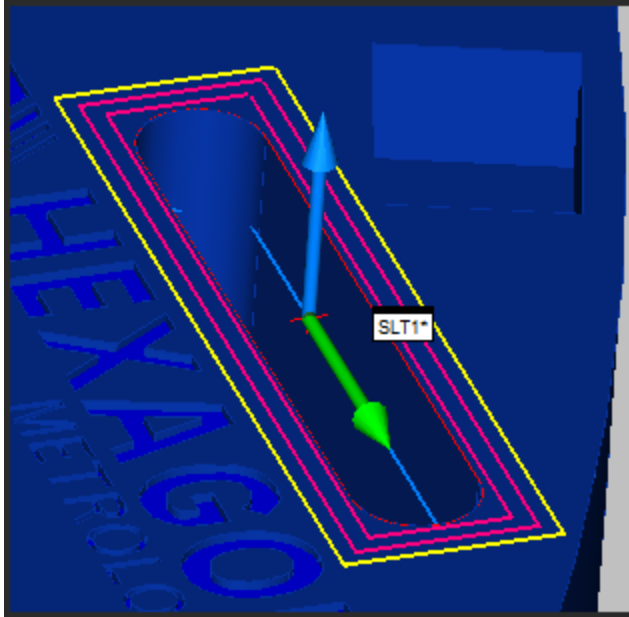


Even if you specify a depth greater than zero, PC-DMIS always projects the measured results into the plane where the feature resides.



The depth defaults to zero. This is the default value for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth, resulting in a feature calculation error from the feature extraction module.

Slot (Vector) - These boxes define the slot's orientation.



*Sample Round Slot in the Graphic Display window showing:
 Depth (blue slot line)
 Ring Band (pink rectangles)
 Overscan (yellow rectangle)*

Slot Command Mode Text

The Slot command inside the Edit window's Command Mode looks like this:

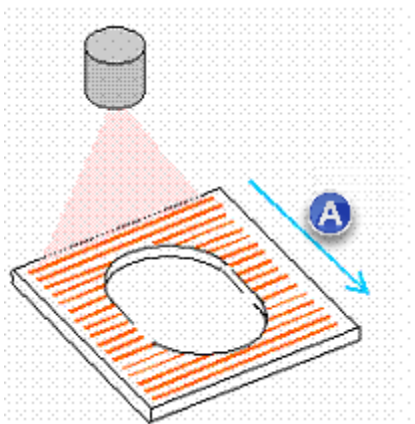
```
SLT1 =FEAT/LASER/SQUARE SLOT,CARTESIAN
      THEO/<1.895,1.91,1>,<0,0,1>,<0,1,0>,3,7
      ACTL/<1.895,1.91,1>,<0,0,1>,<0,1,0>,3,7
      TARG/<1.895,1.91,1>,<0,0,1>
      DEPTH=3
      SHOW FEATURE PARAMETERS=YES
          SURFACE=THEO_THICKNESS,1
          MEASURE MODE=NOMINALS
          RMEAS=NONE,NONE,NONE
          AUTO WRIST=NO
          GRAPHICAL ANALYSIS=NO
          FEATURE LOCATOR=NO,NO,""
      SHOW_LASER_PARAMETERS=YES
          POINT CLOUD ID=DISABLED
          SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
```


`FILTER=NONE`

Auto Round Slot Paths

Depending on the width of the round slot, PC-DMIS takes one of these paths when it performs the measurement:

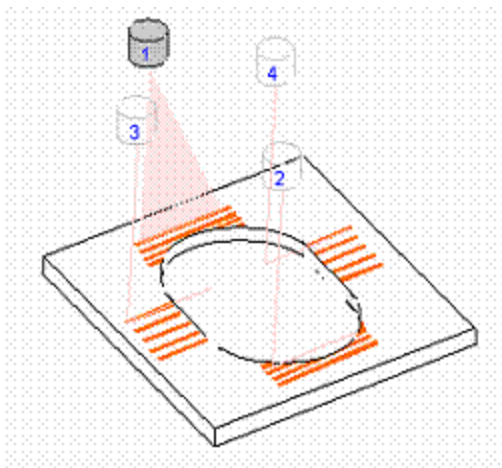
Path 1: Narrow Width



Round slots with a width less than the usable portion of the stripe

(A) Scan motion

Path 2: Larger Width

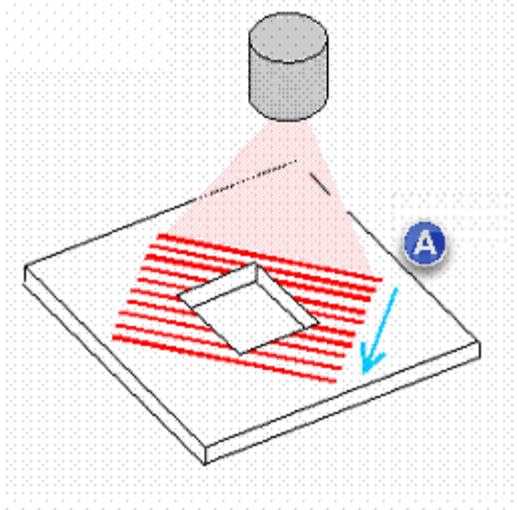


Round slots with a width larger than the usable portion of the stripe

Auto Square Slot Paths

PC-DMIS must measure Auto Square slots at a 45 degree angle to the slot (see the illustrations below). Depending on the size of the slot, PC-DMIS takes one of these two paths.

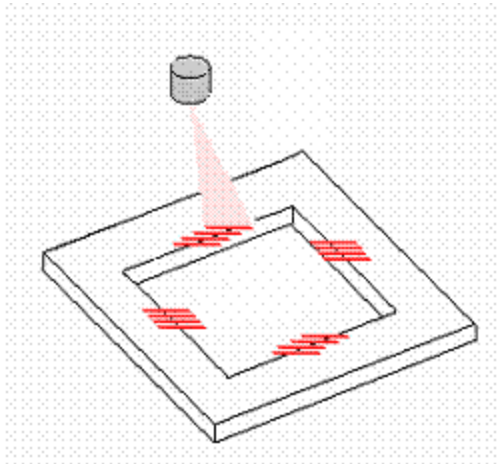
Path 1: Small Slot - Measured with a single pass of the laser sensor



Small square slots require a single pass of the laser sensor's stripe

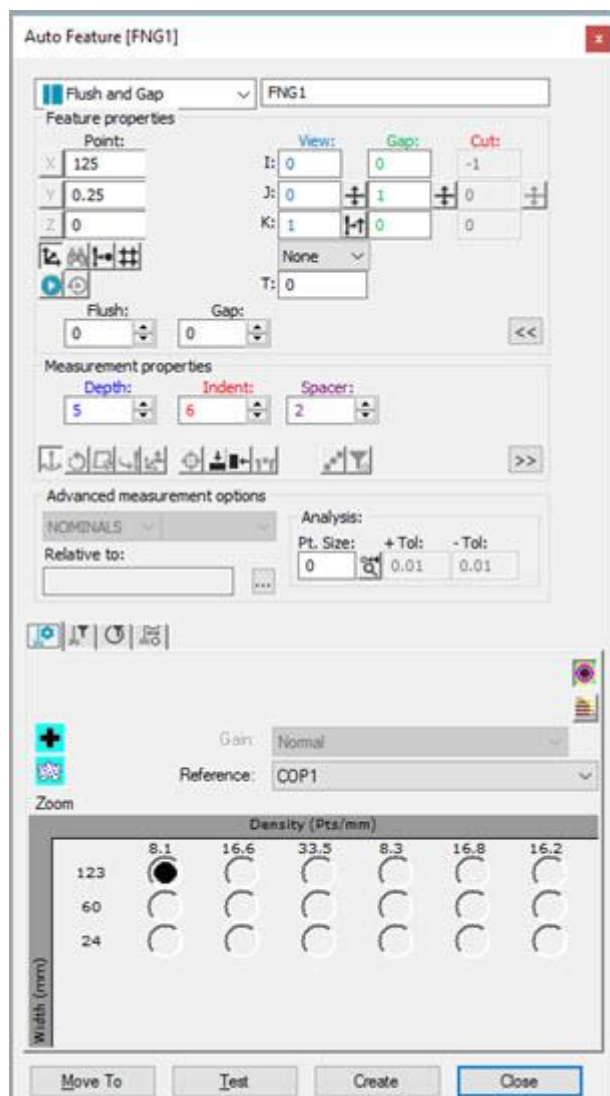
(A) - Diagonal scan motion

Path 2: Large Slot - Measured with multiple passes of the laser sensor



Large square slots require multiple passes of the laser sensor's stripe

Laser Flush and Gap



Auto Feature dialog box - Flush and Gap

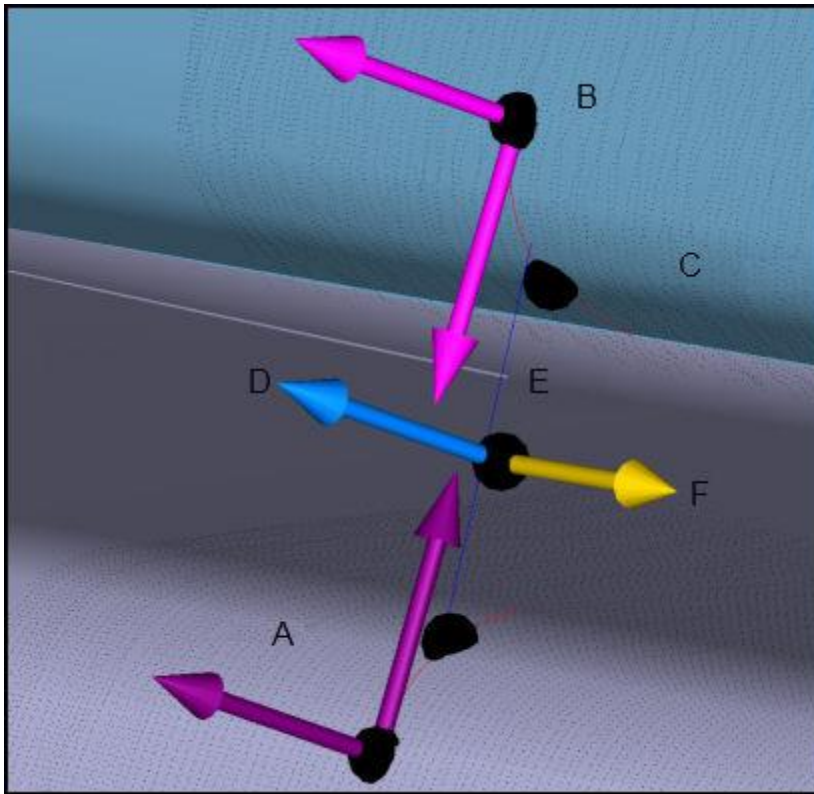
Flush and Gap measures the height difference between two mating sheet metal parts (the Flush) and the distance between two mating parts (the Gap).

To measure a Flush and Gap using a laser sensor, access the **Auto Features** dialog box and select **Flush and Gap**. The dialog box automatically expands the **Extended sheet metal options** area. This area provides **XYZ** position boxes and **IJK** vector boxes for the Master and Gauge points. Follow one of the procedures below.

With CAD Data

1. Load a CAD model.
2. Click on the Master side.

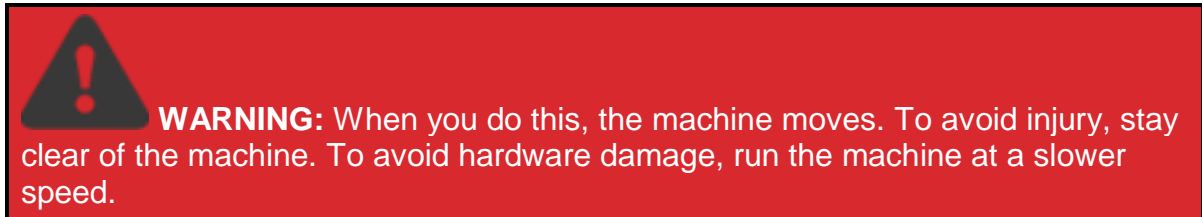
- Click on the Gauge side.



- A - Master
- B - Gauge
- C - CAD learned curves
- D - View Vector
- E - Depth line
- F - Cut Vector

- These points must be on the reference "flat" surfaces, where PC-DMIS sets the planes used to calculate flush, and not on the curves.
- PC-DMIS learns the theoretical Flush.
- PC-DMIS learns the curves from the CAD model.
- PC-DMIS learns the point coordinate and vectors for both the Master and Gauge sides of the gap.
- PC-DMIS applies the defined Depth value, and after piercing the curves it calculates the theoretical gap at the specified depth.
- PC-DMIS also calculate the cut vector (along the rail) and the Gap direction (crossing the rail).
- Set the **Indent** and the **Spacer** values so that they only include points on the flat surfaces and not points on the curved portion.

11. Set other parameters as needed. See "Flush and Gap Specific Parameters".
12. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping** properties tabs to enter the information.
13. If desired, click the **Test** button to test the feature.



14. Click the **Create** button and then **Close**.

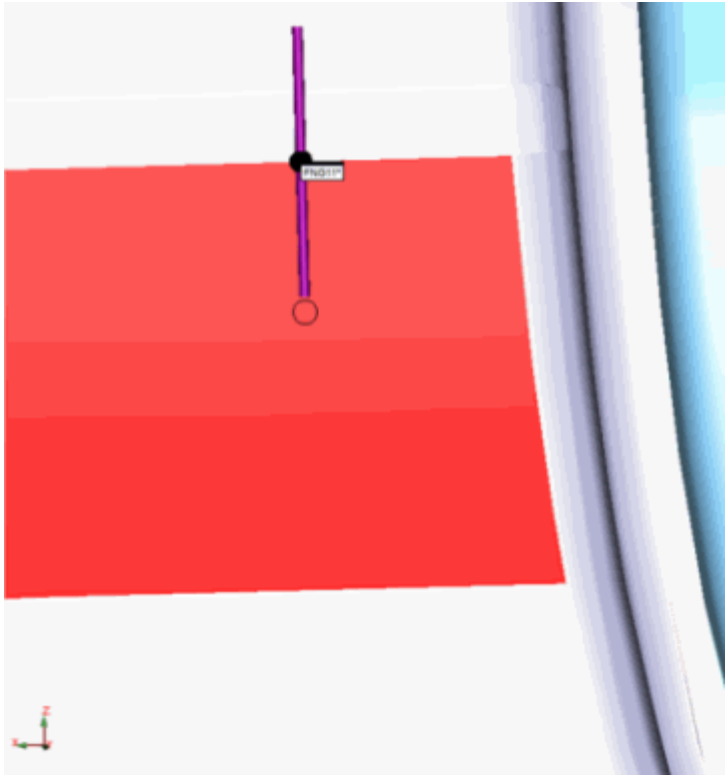
Flush and Gap CAD Selection Capability

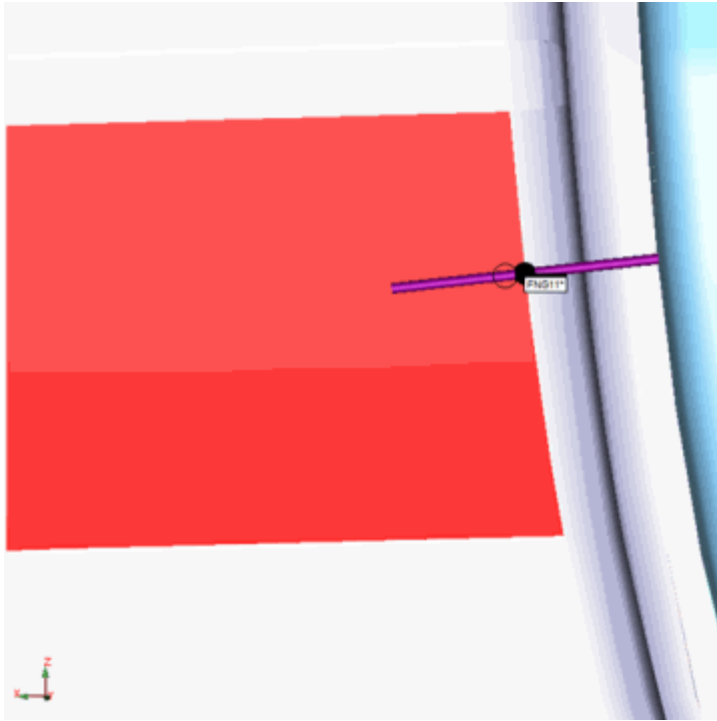
The ability to re-click the first CAD point on a selected surface is often a requirement when defining or re-defining a measurement routine.

The first point clicked on in the Graphic Display window, other than the Master Side Point and the Edge Vector, now displays as a black circle centered in the picked point, and the selected surface is highlighted.

Sometimes it happens that the Master Side Point found is in a wrong surface boundary location, and the point is required to be clicked on again. The following describes two ways this can be done:

1. If the desired Master Side Point is on the edge of the highlighted surface, then it is sufficient just to re-click a point on the surface very close to the edge.
2. If the desired Master Side Point doesn't lie on the highlighted surface, clicking the drawn circle area causes the interface to be reset. PC-DMIS is then ready for you to re-take the first point. To help re-define the new surface selection, the previous surface is left highlighted. See pictures below.





Example of Flush and Gap CAD Selection Capability

Without CAD Data

1. Move the machine to the gap location using the **Laser** tab of the Graphic Display window.
2. Click the **Read Point from Position** button.
3. Manually type all of the theoretical XYZ and IJK values. These include the Flush and Gap **Point**, the **View Vector**, **Gap Dir** (gap direction), **Master Pnt** (master point), **Gauge Pnt** (gauge point), **Master Vec** (master vector), and **Gauge Vec** (gauge vector).
4. Note that when you change some parameters, and you don't have any CAD data, PC-DMIS adjusts some parameter values automatically. For information, see "Automatically Adjusted Flush and Gap Values".
5. Set the **Indent** and the **Spacer** values so that they only include points on the flat surfaces and not points on the curved portion.
6. Set other parameters as needed. For information, see "Flush and Gap Specific Parameters".
7. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping** properties tabs to enter the information.
8. If desired, click the **Test** button to test the feature.



WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

9. Click the **Create** button and then **Close**.

Flush and Gap Specific Parameters

For a visual example of these parameters, consult the diagrams below.

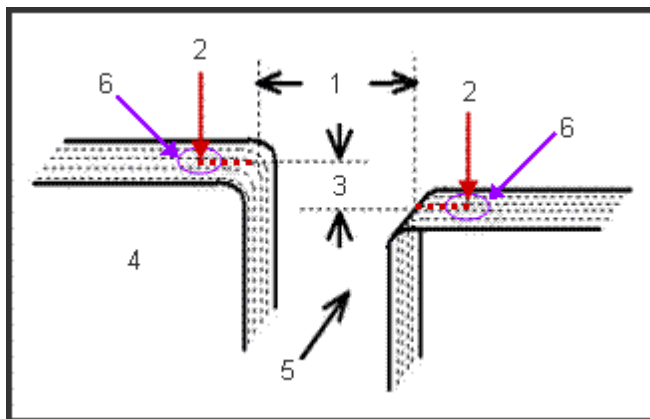
Flush - This box determines the height difference between two mating sheet metal parts. Whether or not the flush value is positive or negative depends on whether it's higher or lower than the "Master" side.

Gap - This box determines the distance (in the same plane) between two mating sheet metal parts.

Indent - The indent specifies the distance from the gap's edge at which PC-DMIS measures the flush.

Spacer - This is a circle at the indent point used to calculate the surface normals used in the calculation.

Gap Dir (Vector) - These boxes in the **Feature Properties** area define the direction of the gap.



Flush and Gap Diagram

Key:

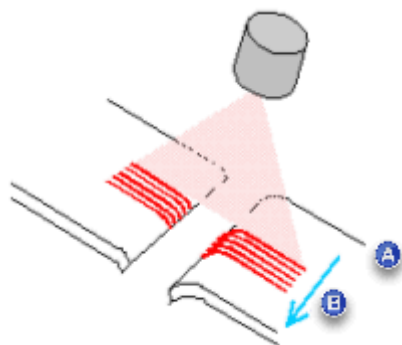
1 - Gap

- 2 - Indent
- 3 - Flush (negative flush is shown at left)
- 4 - Master side
- 5 - Cut Vector
- 6 - Spacer



The "Master" side is always to the left of the scan/gap direction.

The direction of the scan is controlled by the specified Cut Vector and not the direction of the laser stripe.

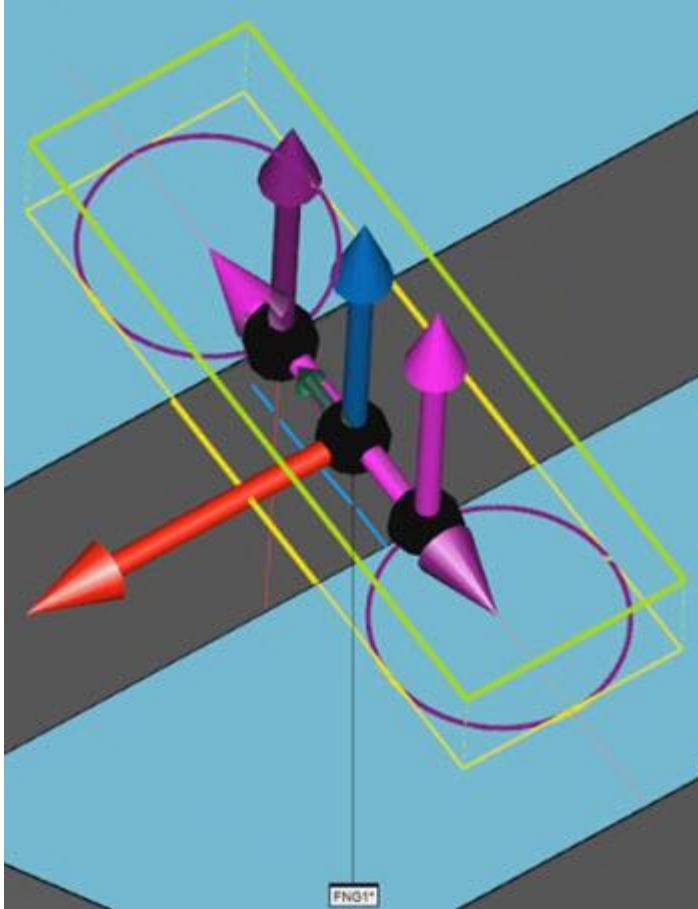


Direction of scan

(A) - Masterside (B) - Scan motion



The "Master" side is always to the left of the Cut Vector.



Sample Flush and Gap in the Graphic Display window showing the Indent (red lines), Spacer (purple circles), Depth (blue line), Horizontal clipping region (yellow lines), Vertical clipping region (green), the View vector (blue arrow) and the Cut vector (red arrow).

Flush and Gap Command Mode Text

The Flush and Gap command inside the Edit window's Command Mode looks like this:

```
FNG2 =FEAT/LASER/FLUSH AND GAP/DEFAULT,CARTESIAN
  THEO/<124.012,13.241,0>,<0,0,1>,<1,0,0>,0,7.985
  ACTL/<124.012,13.241,0>,<0,0,1>,<1,0,0>,0,7.985
  TARG/<124.012,13.241,0>,<0,0,1>
  MASTER SIDE POINT
  THEO/<128,13.241,0>,<0,0,1>
  ACTL/<0,0,0>,<0,0,0>
  GAUGE SIDE POINT
  THEO/<120,13.241,0>,<0,0,1>
```

```
ACTL/<0,0,0>,<0,0,0>
CUT PLANE VECTOR<0,1,0>,<0,1,0>
Depth=1
INDENT=3
SPACER=1.5
SHOW FEATURE PARAMETERS=NO
SHOW_LASER_PARAMETERS=YES
    POINT CLOUD ID=DISABLED
    ZOOM=2A,GAIN=NORMAL,OVERLAP=1
    OVERSCAN=5
    REDUCTION FILTER=OFF
    FILTER LINES=Disabled
    CLIPPING TOP=100,BOTTOM=0,LEFT=0,RIGHT=100
    SOUND=ON
    HORIZONTAL CLIPPING=2,VERTICAL CLIPPING=5
```

Flush and Gap Graphical Analysis

The Flush and Gap analysis is comprised of these three regions. Consult the diagram at the bottom of this topic:

1. **Gap region** - In the Gap region, the points that are analyzed are in a box centered on the gap point and oriented along the Gap vector. The box's height is 60% of Gap length value. The width is 130% of Gap length value.
2. **Master Flush region** - In the Master Flush region, the points are analyzed in an area that begins at the Master Side point in a direction opposite from the Master Edge Vector. It has a length that is 60% of the Gap length value.
3. **Gauge Flush region** - In the Gauge Flush region, the points are analyzed in an area that begins at the Gauge Side point in the direction opposite from the Gauge Edge Vector. It has a length that is 60% of the Gap length value.

The Flush and Gap analysis is performed using these measured items.

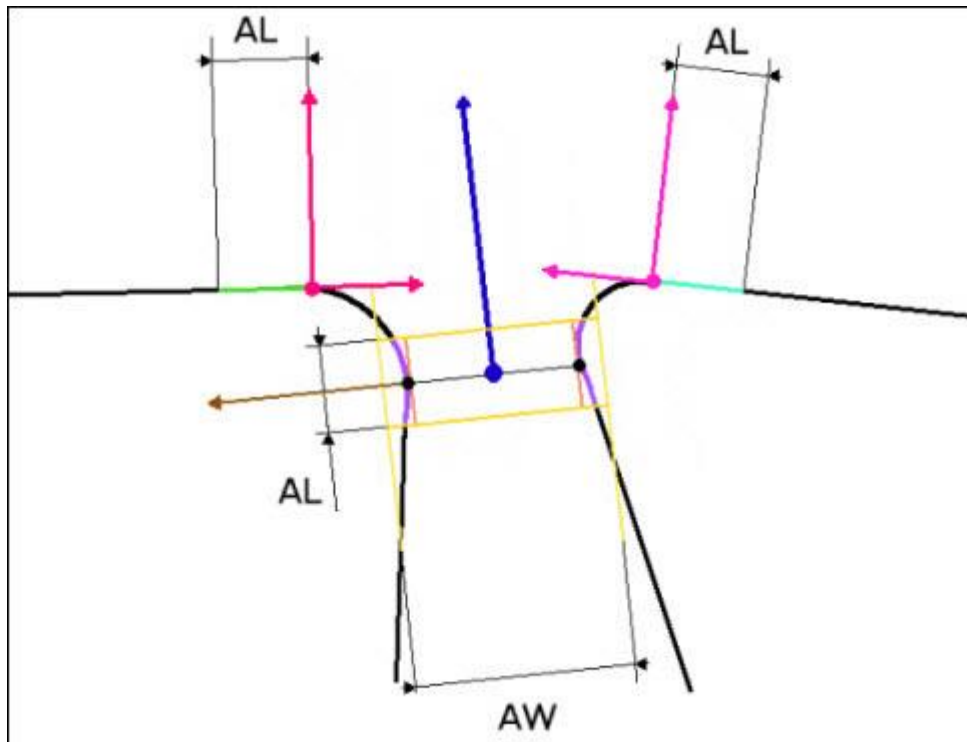
- Gap Point and Vector
- Master Side Point
- Master Side Surface and Edge Vectors
- Gauge Side Point
- Gauge Side Surface and Edge Vectors

PC-DMIS computes the Flush and Gap measured point's distance from these four measured reference planes:

- The first two planes are the Gap analysis reference planes defined from the two measured Minimum Distance Points (where the Gap distance is calculated) and the measured Gap Vector.
- The third plane is the measured Master Side analysis reference plane. It is defined by the measured Master Side Point and the measured Master Side Surface Vector.
- The fourth plane is the measured Gauge Side analysis reference plane. It is defined by the measured Gauge Side Point and the measured Gauge Side Surface Vector.

To reduce the analysis time, PC-DMIS only uses the points closest to the cut plane (less than 0.5 mm or 0.19685 inches).

Graphical Analysis Diagram:











Key:

AL - Analysis Length. It is 60% of the Gap length value.

AW - Analysis Width. It is 130% of the Gap length value.

● - Minimum Distance Points

-  - Gap Vector
-  - Gap Point and View Vector
-  - Gauge Side Point and Vectors
-  - Master Side Point and Vectors
-  - Master Side Flush analysis region. Reference Plane.
-  - Gauge Side Flush analysis region. Reference Plane.
-  - Gap analysis region
-  - Gap analysis reference plane

Automatically Adjusted Flush and Gap Values

Note that when you change some Flush and Gap parameters, and you don't have any CAD data, PC-DMIS will adjust some parameters' values automatically. This topic details what gets changed and how the software computes those automatic values.



Key: Use these abbreviations when viewing the equations below:

CPV = Cut Plane Vector
 VV = View Vector
 x = Cross Product
 GV = Gap Vector
 GD = Gap Distance
 GP = Gap Point
 GPV = Gap Point Vector

When Typing in a Gap Point Value or Modifying it by Read Position...

- The current probe vector is used as the View Vector.
- The current stripe vector is used as the Gap Vector.
- The new cut plane is located in the Gap point, and the new Cut Plane Vector is computed: $CPV = VV \cdot x(GV)$

- Master Side Point and Gauge Side Point are ESTIMATED at $\frac{GD}{2}$ from the new Gap Point along the Gap Vector.

If the Flush distance is positive, the Master Side Point is translated along the View Vector of the flush value.

If the Flush distance is negative, the Gauge Side Point is translated along the View Vector of the flush value.

- The Master Side Surface Vector and the Gauge Side Surface Vector are set with the View Vector.

When Typing in a View Vector Value...

- The new cut plane is located in the Gap point, and the new Cut Plane Vector is computed: $CPV = VV.x(GV)$
- The Gap Vector is computed to be orthogonal to the new View Vector: $GV = CPV.x(VV)$
- The Master Side Surface Vector and the Gauge Side Surface Vector are projected onto the new cut plane.
- The Master Side Point and Gauge Side Point are projected onto the new Cut Plane.

When Typing in a Gap Vector Value...

- The new cut plane is located in the Gap point and the new Cut Plane Vector is computed: $CPV = VV.x(GV)$
- The View Vector is computed to be orthogonal to the new Gap Vector: $VV = GV.x(CPV)$
- The Master Side Surface Vector and the Gauge Side Surface Vector are projected onto the new cut plane.
- Master Side Point and Gauge Side Point are projected onto the new Cut Plane.

When Typing in a Master Side Point Value or Modifying it by Read Position...

- The new cut plane is computed orthogonal to the View Vector and the Master Side Point minus the Gap Point: $CPV = VV.x(MSP - GP)$
- The Gap Vector is computed orthogonal to the new View Vector: $GV = CPV.x(VV)$
- The Master Side Surface Vector, the Gauge Side Surface Vector and Gauge Side Point are translated onto the new cut plane.

When Typing in a Gauge Side Point Value or Modifying it by Read Position...

- The new cut plane is computed centered on the new Master Side Point and orthogonal to the View Vector and the Master Side Point minus the Gauge Side Point: $CPV = VV.x(MSP - GSP)$
- The Gap Vector is computed orthogonal to the new View Vector: $GV = CPV.x(VV)$
- The Master Side Surface Vector, the Gauge Side Surface Vector, and Gap Point are translated onto the new cut plane.

When Typing in a Flush Distance Value...

- The Master Side Point and/or the Gauge Side Point are translated according the new flush value along the Master or Gauge Side Surface Vector.

When Typing in the Distance Value...

- The Master Side Point and/or the Gauge Side Point are translated according the new gap value along the Gap Vector.

Flush and Gap Features Around a Defined Contour

The ability to extract a series of Flush and Gap features around a defined contour is available. See the following examples.

First Curve Selection

Auto Feature [FNG1]

Flush and Gap

FNG1

Feature properties

Point:

X

2616.647

Y

841.174

Z

506.797

Flush:

0.488

Gap:

2.701

Measurement properties

Depth:

2

Indent:

6

Spacer:

3

Advanced measurement options

NOMINALS

Relative to:

Analysis:

Pt. Size:

2

+ Tol:

0.01

- Tol:

0.01

Create Multiple Features

CAD Selection Method:

CURVE

Step:

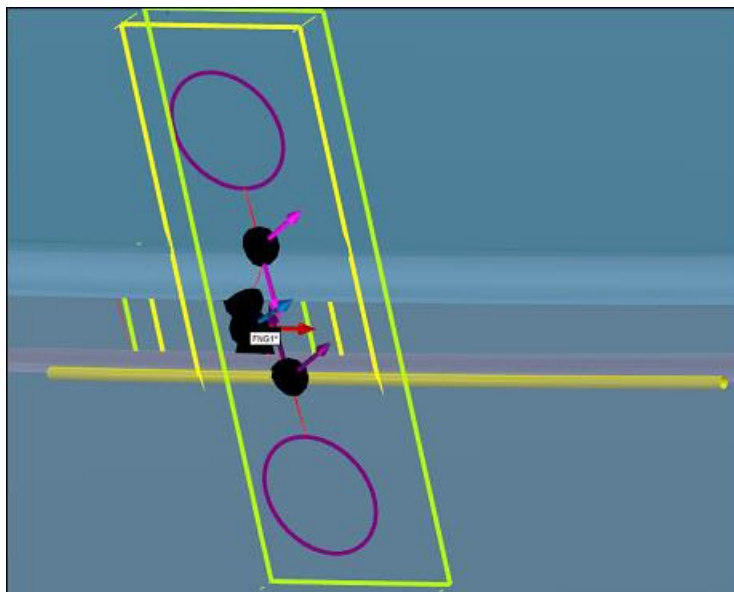
10.0

Move To

Test

Create

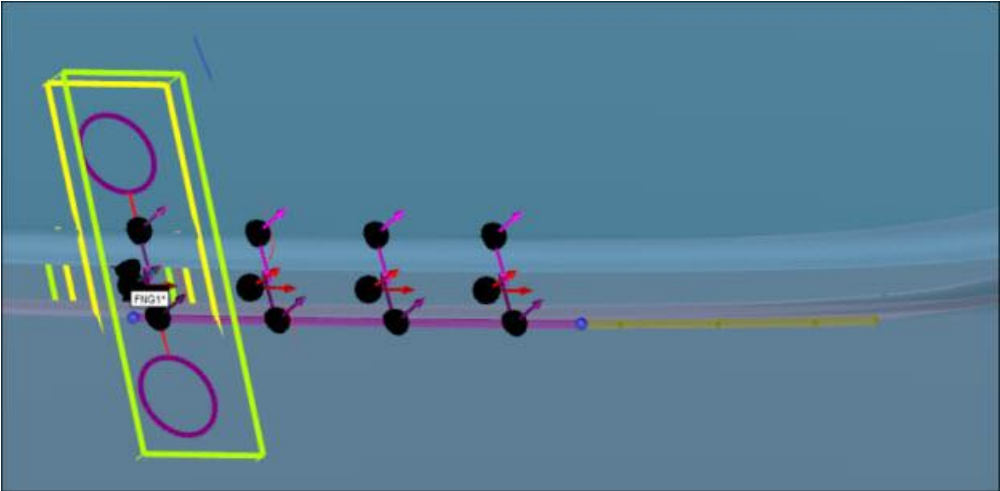
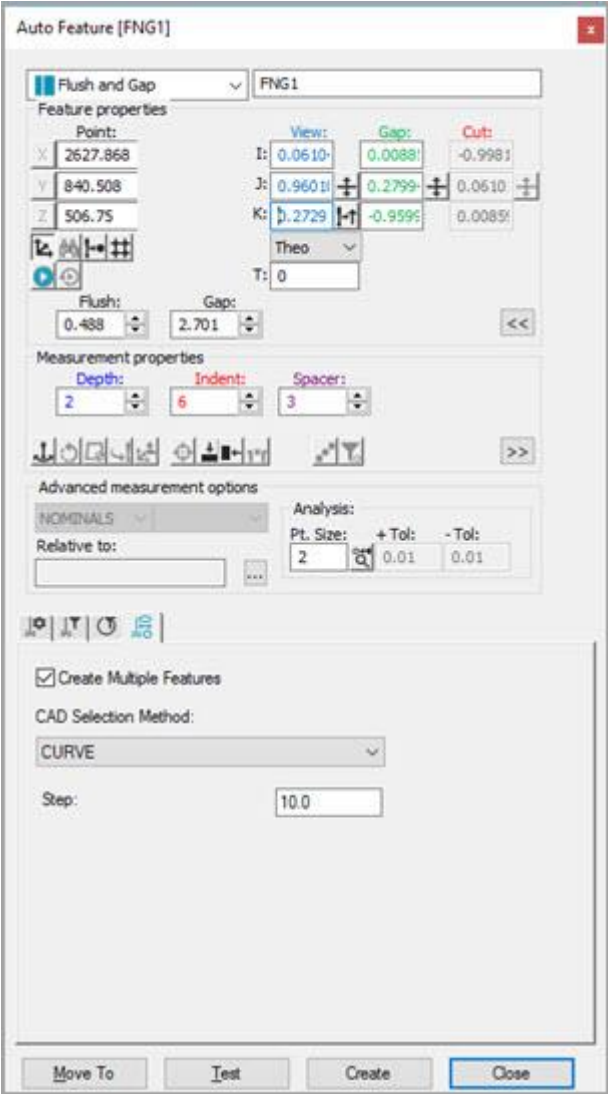
Close



Flush and Gap Auto Feature - First Curve Selection

Additional Curve Selection with Ctrl Key

To select additional curves, press and hold the Ctrl key.

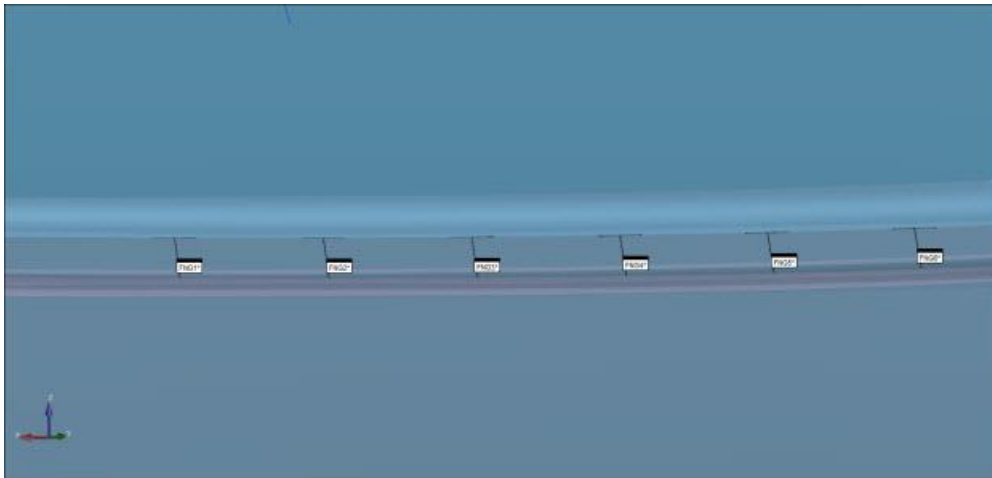


Flush and Gap Auto Feature - Additional Curve Selection

To select additional curves, continue to press and hold the Ctrl key to create the Flush and Gap features.

Result

```
TIP1 = Set Active Tip
COP1 = Pointcloud
COP1 FLUSHGAP GRP1 = Group
  Id : COP1_FLUSHGAP_GRP1
  FNG1 = FLUSHANDGAP (LASER)
  FNG2 = FLUSHANDGAP (LASER)
  FNG3 = FLUSHANDGAP (LASER)
  FNG4 = FLUSHANDGAP (LASER)
  FNG5 = FLUSHANDGAP (LASER)
  FNG6 = FLUSHANDGAP (LASER)
```



Flush and Gap Auto Feature - Result

Laser Polygon

Auto Feature [POL1]

☒ Polygon POL1

Feature properties

Center:

X: 26.364

Y: 49.5

Z: 15

Surface: I: 0 J: 0 K: 1 T: 0

Angle: 3.8660; 0.5 0

Inner/Outer: In Diameter: 15 Num Sides: 0

Measurement properties

Depth: 0

Advanced measurement options

Analysis: LEAST_SQR

Relative to:

Pt. Size: 0 + Tol: 0.01 - Tol: 0.01

Feature Based Clipping

Horizontal (mm): 2

Vertical (mm): 1

☒ Ring Band

Inner offset (mm): 0.5

Outer offset (mm): 1

Filters

☐ Remove points with normals outside:

Max incidence angle: 75

Move To Test Create Close


Auto Feature dialog box - Polygon

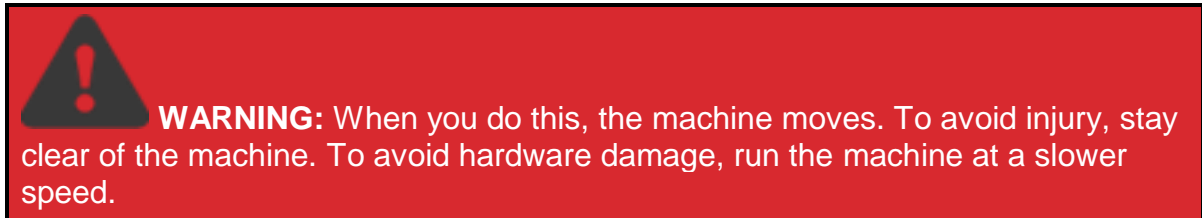


You can only use this dialog box to measure a hexagon feature (a polygon with 6 sides).

To measure a hexagon feature with a laser sensor:

1. Access the **Auto Features** dialog box, and select **Polygon**.
2. Do one of the following:

- Perform clicks on the CAD to give the polygon a location and vector. Manually enter any remaining information.
 - Move the machine to the sphere location using the **Laser** tab of the **Graphic Display window**. Click the **Read Point from Position** button (). Manually enter any remaining information, such as the diameter, as needed.
 - Manually enter all of the theoretical information for X, Y, Z, I, J, K, Diameter, and other parameters.
3. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping** properties tabs to enter the information.
 4. If desired, click the **Test** button to test the feature.



5. Click **Create** and then **Close**.

Polygon Specific Parameters

Num Sides - This parameter defines the number of sides used on the polygon. For Laser devices the number of sides for the Auto Feature Polygon is fixed at 6.

Diameter - The value in this box defines the polygon's diameter.

Depth - This parameter controls what data PC-DMIS uses to calculate the feature characteristics. You can use the depth value to eliminate data on a chamfer or some other transitional portion of the feature that you don't want in the feature calculation. Specifying a positive value tells PC-DMIS where along the feature PC-DMIS looks to calculate the feature characteristics. A depth of 0 causes this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. A depth of any other value causes it to be calculated at that depth. Due to hardware limitations, for this feature type if you do use a depth value higher than 0, you must use a minimum of 0.3 mm (.01181 inches).

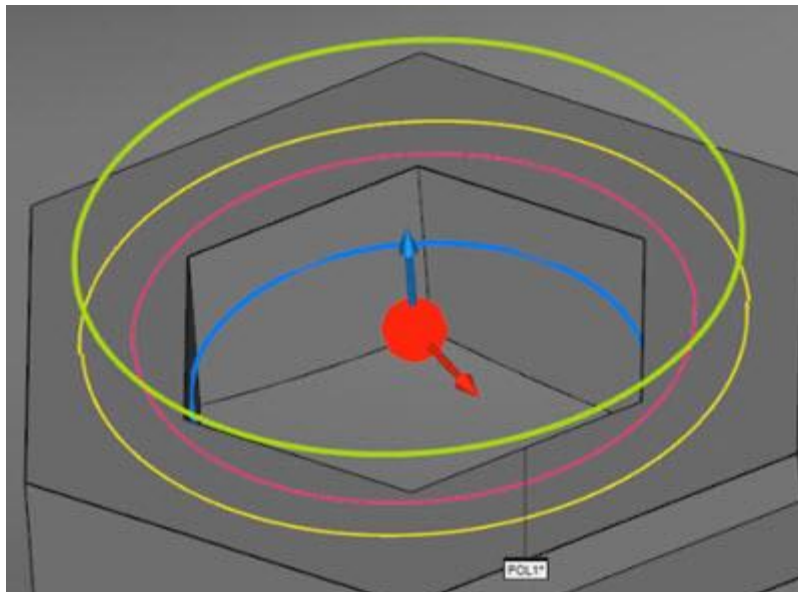


The depth defaults to zero. This is the default value for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth, resulting in a feature calculation error from the feature extraction module.

For example, a depth of 3 indicates that you want to use all data 3 mm (or inches, depending on the units of the measurement routine) and above for the calculation. If you specify 0, this indicates that you want to use all available data for the calculation. For thin walled features, a 0 value may be valid; but for parts with any depth to them, you probably need to specify a depth to get accurate results.



Even if you specify a depth greater than zero, the measured results are always projected into the plane where the feature resides.



Sample Polygon in the Graphic Display window showing:

- *the Ring Band (pink circles)*
- *the Horizontal overscan (yellow circle)*
- *the Vertical overscan (green circles)*
- *the Depth (blue)*

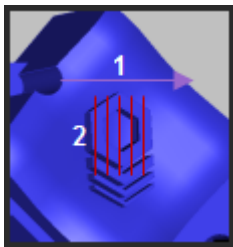
Polygon Command Mode Text

The Polygon command inside the Edit window's Command Mode looks like this:

```
POL1 =FEAT/LASER/POLYGON,CARTESIAN
      THEO/<1.0379,1.9488,0.5906>,<0,0,1>,<0.8660254,-
      0.5,0>,0.5118
      ACTL/<1.0379,1.9488,0.5906>,<0,0,1>,<0.8660254,-
      0.5,0>,0.5118
      TARG/<1.0379,1.9488,0.5906>,<0,0,1><0.8660254,-0.5,0>
      NUMSIDES=6
      DEPTH=0
      SHOW_FEATURE_PARAMETERS=NO
      SHOW_LASER_PARAMETERS=YES
      POINT_CLOUD_ID=DISABLED
      SENSOR_FREQUENCY=30,OVERLAP=0.0394
      OVERSCAN=0.0787,EXPOSURE=35
      FILTER=NONE
      PIXEL_LOCATOR=GRAY SUM,Min=30,Max=300
      CLIPPING TOP=100,BOTTOM=0,LEFT=0,RIGHT=100
      RINGBAND=OFF
```

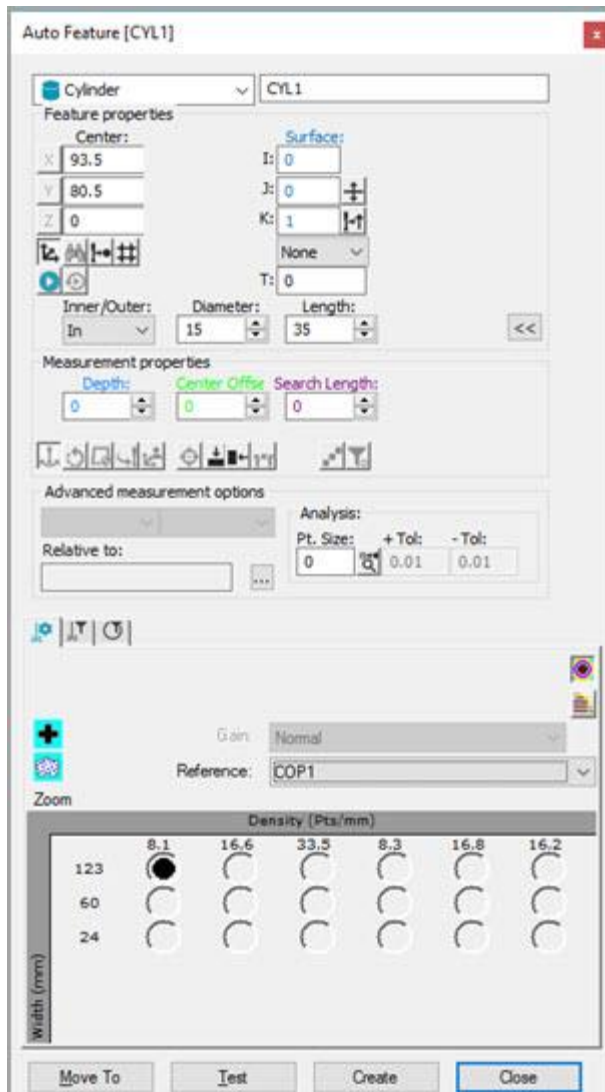
Auto Polygon Paths

PC-DMIS uses the **Angle** IJK vector to determine the direction of the scan.



The feature's scan lines or laser stripes (shown in 2) are perpendicular to the feature's angle vector (shown in 1)


Laser Cylinder



Auto Feature dialog box - Cylinder

To measure a cylinder with a laser sensor:

1. Access the **Auto Features** dialog box, and select **Cylinder**.
2. From the **Inner/Outer** box, choose **In** or **Out**.
3. Do one of the following:
 - Perform clicks on the CAD to give the cylinder location and vector. Manually enter any remaining information.
 - From the Graphic Display window, use the **Laser** tab to move the machine to the cylinder location. Next, from the **Feature Properties** area, click **Read**

Point from Machine . Manually enter the any remaining information such as the Inner/Outer value, diameter, length, and other parameters.

- Manually enter all of the theoretical information for X, Y, Z, I, J, K, Inner/Outer value, diameter, length, depth, and other parameters.
4. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping** properties tabs to enter the information.
 5. If desired, click the **Test** button to test the feature.



WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

6. Click the **Create** button and then **Close**.



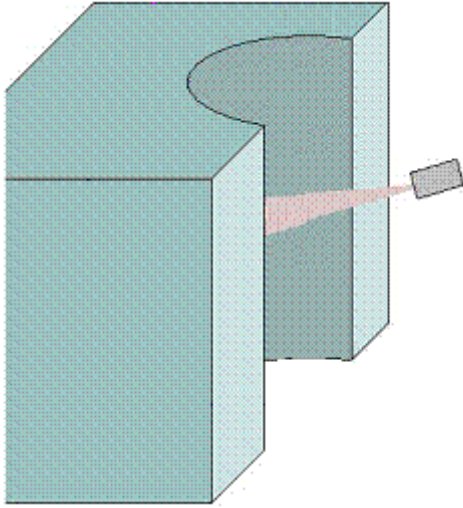
The location and direction vector of the feature defines the center axis of the cylinder.

Cylinder Specific Parameters

Diameter - The value in this box defines the cylinder's diameter.

Length - The value in this box provides the length (height) of the cylinder's axis. The length parameter is only valid as a nominal. The software does not actually measure the length.

Inner/Outer - This parameter defines whether or not the cylinder is an inner cylinder (hole) or an outer cylinder (including a stud).



Unlike other Laser Auto Features, for the **Overscan** value on the **Laser Scan Properties** tab of the **Probe Toolbox**, you should use negative values. This limits measurement in the cylindrical region along the cylinder axis.

Depth - This parameter controls the location of the laser focal point in relation to the cylinder's outside diameter (outer cylinders) or the cylinder's center axis (inner cylinders). This allows you to control how the laser stripes fall on the cylinder's surface because you can specify how far or close the laser is to the cylinder's surface. A depth of 0 for an internal feature means that the laser sensor center is on the cylinder center axis. For an external feature, it is on the surface of the outer cylinder.

- A negative depth value moves the laser sensor's center away from the cylinder's surface.
- A positive depth value moves the laser sensor's center closer to the cylinder's surface.

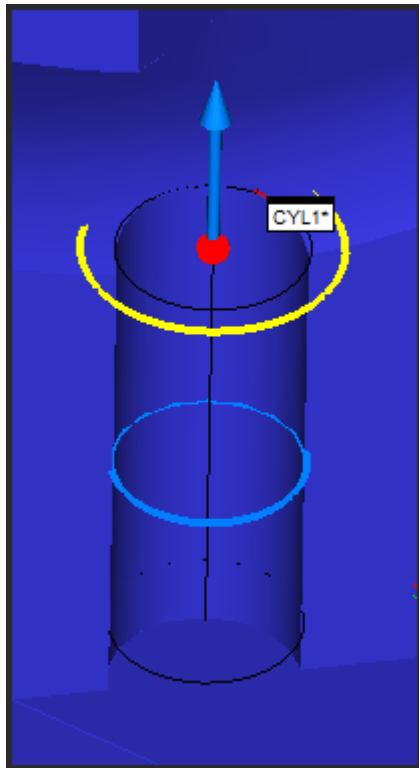
Center Offset - This value identifies the center of the cylinder portion of the stud.

Search Length - This value identifies the length of the cylinder portion.



The depth defaults to zero for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth. This results in a feature calculation error from the feature extraction module.

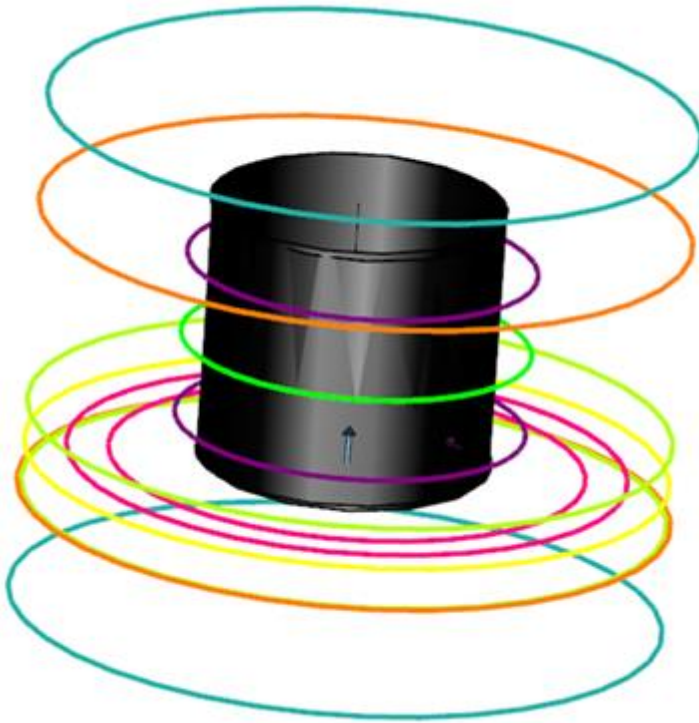
Sample Inner Cylinder



Sample inner Cylinder that shows:

- The **Depth** (blue circle)
- The **Length** (bottom black circle)
- The **Center Point** (yellow circle)

Sample Outer Cylinder



Sample stud Cylinder that shows:

- The **Search Length** (purple circles)
- The **Center Offset** (lime green circle)
- The **Point Segregation** (orange circles)
- The **Center Point** (yellow circle)
- The **Clipping Plane** (light green circles)
- The **Overscan** (sea green circles)
- The **Ring Band** (pink circles)

Cylinder Command Mode Text

Sample Cylinder

```

CYL1 =FEAT/LASER/CYLINDER/DEFAULT,CARTESIAN,OUT
      THEO/<3.1425,2.7539,0>,<0,0,1>,0.25,0.25
      ACTL/<3.1425,2.7539,0>,<0,0,1>,0.25,0.25
      TARG/<3.1425,2.7539,0>,<0,0,1>
      DEPTH=0
      CENTER OFFSET=0
  
```

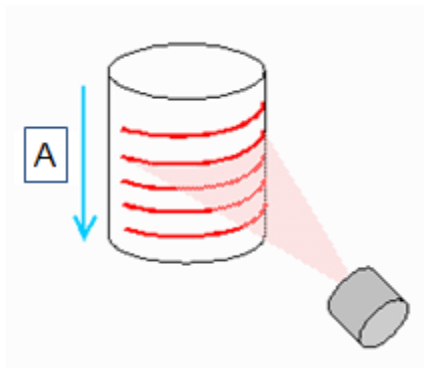
```
SEARCH_LENGTH=0
SHOW_FEATURE_PARAMETERS=NO
SHOW_LASER_PARAMETERS=YES
POINT_CLOUD_ID=COP1
HORIZONTAL_CLIPPING=0.0787,VERTICAL_CLIPPING=0.0787
RINGBAND=ON,INNER_OFFSET=0.5,OUTER_OFFSET=2
```

Auto Cylinder Paths

Cylinder Measurements

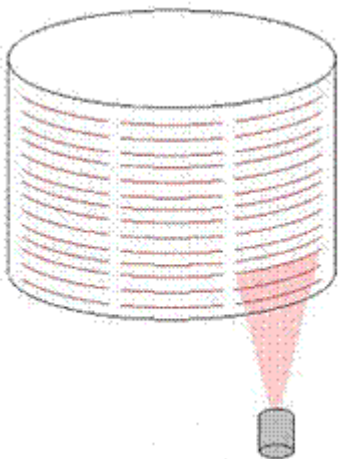
Adjust the processing window in the Laser View to include as much of the cylindrical surface as possible. Laser plane must be roughly normal to the cylinder axis (< 30 degree deviation). Depending on the diameter of the cylinder, PC-DMIS takes one of these paths when performing the measurement:

Path 1: Single Scan



Cylinders with a diameter less than the usable portion of the stripe. A is the scan motion.

Path 2: Multiple Scans

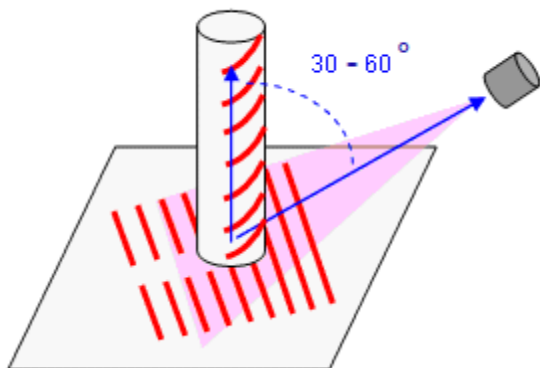


Cylinders with a diameter larger than the usable portion of the stripe

Stud Measurements

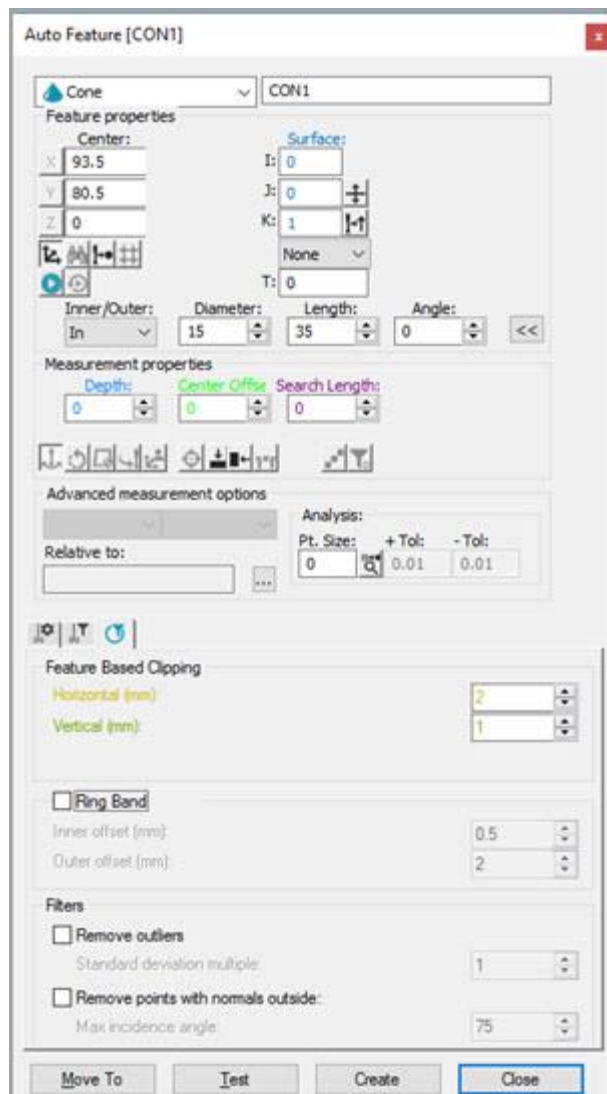
Single Scan

Adjust the processing window in the Laser View to include as much of the cylindrical surface as possible. The laser plane must be roughly 30~60 degrees to the cylinder axis. The scan must capture the region on the base plane of the stud where the cylinder is mounted.



Single pass laser scan on stud cylinder


Laser Cone



Auto Feature dialog box - Cone

To measure a cone with a laser sensor:

1. Access the **Auto Feature** dialog box, and select **Cone**.
2. From the **Inner/Outer** box, select **In** or **Out**.
3. Do one of the following:
 - Click on the CAD to give the cone location and vector, and then manually enter any remaining information.
 - From the Graphic Display window, use the **Laser** tab to move the machine to the cone location. Next, from the **Feature Properties** area, click the **Read**

Point from Position button (). Manually enter any remaining information such as the inner/outer value, diameter, length, and other parameters.

- Manually enter the theoretical information for X, Y, Z, I, J, K, inner/outer value, diameter, length, depth, and other parameters.
4. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping** properties on the tabs to enter the information.
 5. If desired, click the **Test** button to test the feature.



WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

6. Click **Create** and then **Close**.



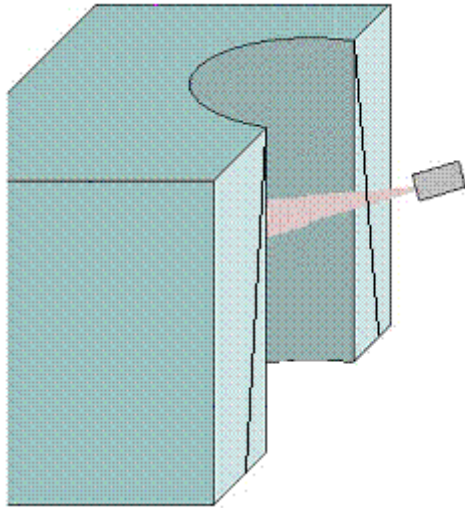
The location and direction vector of the feature define the center axis of the cone.

Cone Specific Parameters

Diameter: The value in this box defines the cone's diameter.

Length: The value in this box provides the length (height) of the cone's axis. The length parameter is only valid as nominal. No actual length measurement will be performed.

Inner/Outer: This parameter defines whether or not the cone is an inner cone (hole) or an outer cone (stud).



The **OverScan** value on the **Laser Scan Properties** tab of the **Probe Toolbox** should use negative values unlike other Laser Auto Features. This limits measurement in the conical region along the cone axis.

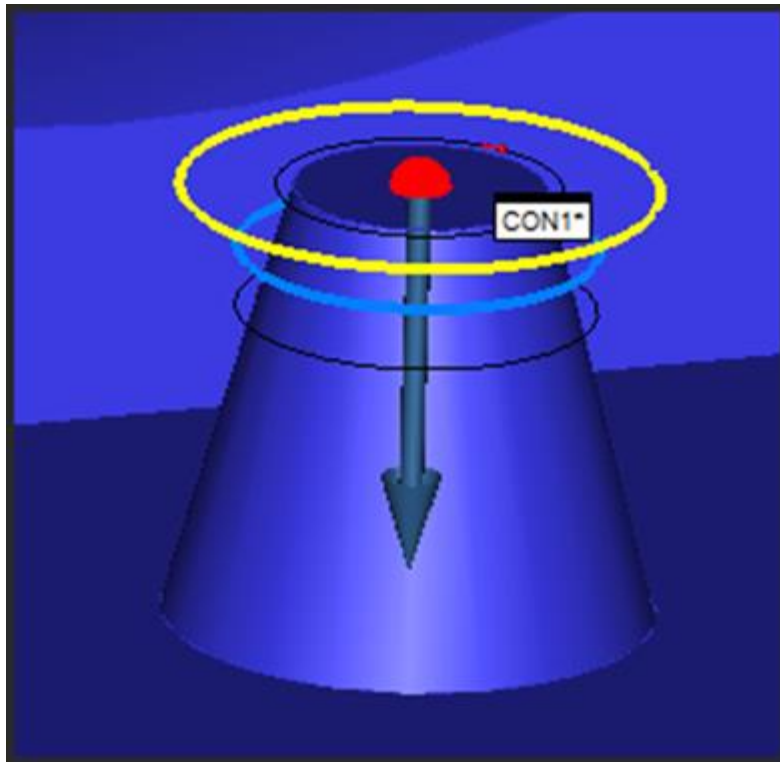
Depth - This parameter controls the location of the laser focal point in relation to the cone's outside diameter (outer cones) or the cone's center axis (inner cones). This allows you to control how the laser stripes fall on the cone's surface by specifying how far or close the laser is to the cone's surface. A depth of 0 (zero) causes this feature to be calculated at the surface plane height using data found at the lowest possible depth from the surface plane. A depth of any other value causes it to be calculated at that depth.

Center Offset - This value identifies the center of the cone portion of the stud.

Search Length - This value identifies the length of the cone portion.

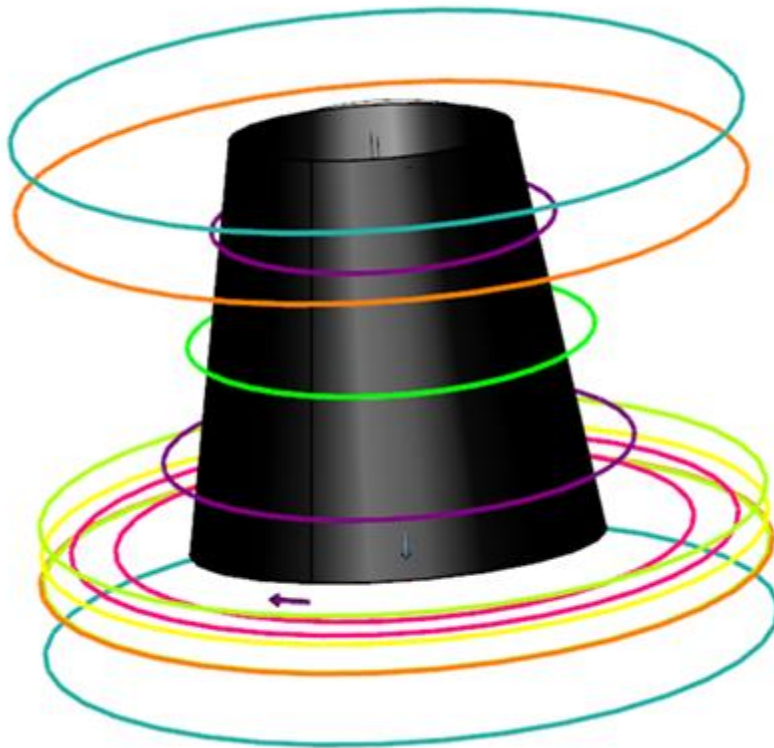


The depth defaults to 0 (zero). This is the default value for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth, resulting in a feature calculation error from the feature extraction module.



Sample external Cone in the Graphic Display window showing:

- The **Diameter** (top black circle)
- The **Length** (bottom black circle)
- The **Depth** (blue circle)
- The **Center Point** (yellow circle)



Sample external Stud Cone in the Graphic Display window showing:

- The **Search Length** (purple circles)
- The **Center Offset** (lime green circle)
- The **Point Segregation** (orange circles)
- The **Center Point** (yellow circle)
- The **Clipping Plane** (light green circle)
- The **Overscan** (sea green circles)
- The **Ring Band** (pink circles)

Cone Command Mode Text

```
CON1 =FEAT/LASER/CONE/DEFAULT,CARTESIAN,OUT
      THEO/<3.1425,2.7539,0>,<0,0,1>,0.5,20,12.7
      ACTL/<3.1425,2.7539,0>,<0,0,1>,0.5,20,12.7
      TARG/<3.1425,2.7539,0>,<0,0,1>
      DEPTH=0
      CENTER OFFSET=3
      SEARCH LENGTH=2
      SHOW FEATURE PARAMETERS=YES
```

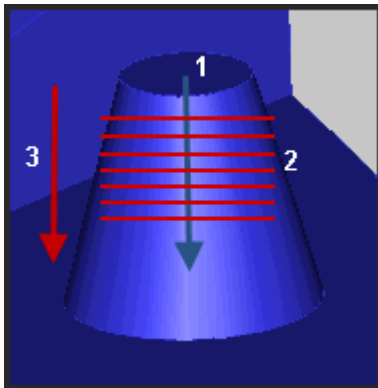
```

SURFACE=THEO_THICKNESS,0
RMEAS=NONE,NONE,NONE
AUTO WRIST=YES
GRAPHICAL ANALYSIS=NO
SHOW_LASER_PARAMETERS=YES
POINT CLOUD ID=COP1
SOUND=OFF
HORIZONTAL CLIPPING=0.0787,VERTICAL CLIPPING=0.0787
RINGBAND=ON,INNER OFFSET=0.5,OUTER OFFSET=2
OUTLIER_REMOVAL=ON,1

```

Auto Cone Paths

The laser sensor scans along the length of the cone. It moves in the direction of the cone's vector. The laser must be nearly perpendicular to that vector.

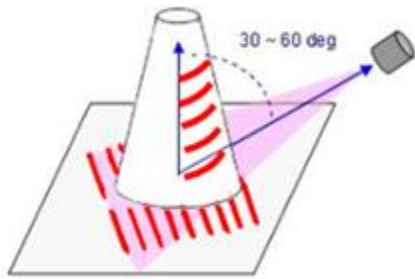


1 - The feature's vector. 2 - The feature's scan lines or laser stripes are perpendicular to the feature's vector. 3 - Scan direction follows the feature's vector

Stud Measurements

Single Scan

Adjust the processing window in the Laser View to include as much of the cone surface as possible. The laser plane must be roughly 30-60 degrees to the cone axis. The scan must capture the region on the base plane of the stud where the cone is mounted.



Single pass laser scan on stud cone

Laser Sphere

Auto Feature [SPH1]

Sphere SPH1

Feature properties

Center:

X: 93.5
Y: 80.5
Z: 0

Surface:

I: 0
J: 0
K: 1
None
T: 0

Inner/Outer: In Diameter: 15

Measurement properties

Advanced measurement options

Relative to:

Analysis:

Pt. Size: 0 + Tol: 0.01 - Tol: 0.01

Gain: Normal Reference: COP1

Zoom


	8.1	16.6	33.5	8.3	16.8	16.2
123						
60						
24						

Width (mm)

Move To Test Create Close

Auto Feature dialog box - Sphere

To measure a sphere with a laser sensor:

1. Access the **Auto Features** dialog box, and select **Sphere**.
2. From the **Inner/Outer** box, choose **In** or **Out**.
3. Do one of the following:
 - Clicks on the CAD to give the sphere a location and vector. Manually enter any remaining information.
 - From the Graphic Display window, use the **Laser** tab to move the machine to the sphere. Next, from the **Feature Properties** area, click the **Read Point from Position** button (). Manually enter any remaining information, such as the Inner/Outer value, diameter, and other parameters.
 - Manually enter the theoretical information for X, Y, Z, I, J, K, Inner/Outer value, diameter, and other parameters.
4. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping** properties tabs to enter the information.
5. If desired, click the **Test** button to test the feature.



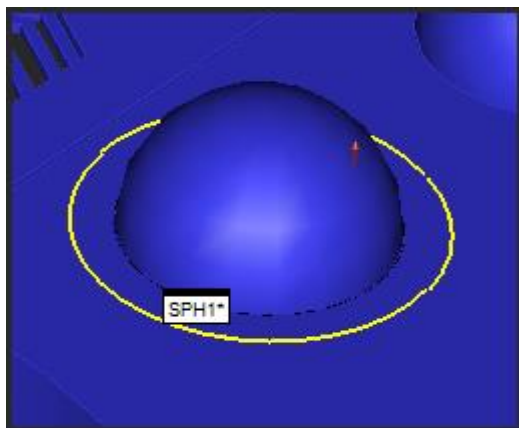
WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

6. Click **Create** and then **Close**.

Sphere Specific Parameters

Inner/Outer: This parameter defines whether or not the sphere is an inner sphere (concave) or an outer sphere (convex).

Diameter: The value in this box defines the sphere's diameter.



Sample outer Sphere in the Graphic Display window showing the Overscan (yellow circle)

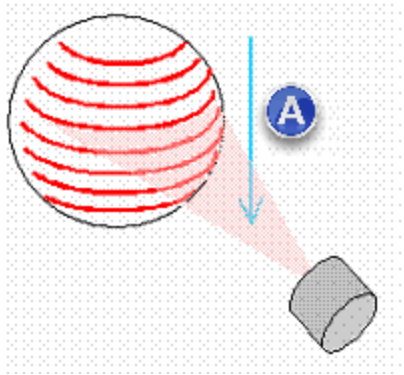
Sphere Command Mode Text

The Sphere command inside the Edit window's Command Mode looks like this:

```
SPH1 =FEAT/LASER/SPHERE,CARTESIAN,IN,LEAST_SQR
      THEO/<1.895,1.91,1>,<0,0,1>,1.895
      ACTL/<1.895,1.91,1>,<0,0,1>,1.895
      TARG/<1.895,1.91,1>,<0,0,1>
      START ANGLE 1=0,END ANG 1=0
      START ANGLE 2=0,END ANG 2=0
      SHOW FEATURE PARAMETERS=YES
          SURFACE=THEO_THICKNESS,0
          MEASURE MODE=NOMINALS
          RMEAS=NONE,NONE,NONE
          AUTO WRIST=NO
          GRAPHICAL ANALYSIS=NO
          FEATURE LOCATOR=NO,NO,""
      SHOW_LASER_PARAMETERS=YES
          POINT CLOUD ID=DISABLED
          SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
          FILTER=NONE
```

Auto Sphere Path

The direction of the path is determined based on the stripe.



Path direction of scan

(A) Scan motion

Clearing Auto Feature Scan Data

PC-DMIS's Laser Auto Features sometimes store scanned data as internal clouds of points following their creation. This occurs if the Point Cloud parameter on the Laser Scan Properties tab is set to **Disabled**.

Two menu items exist to clear out this internal data based on your needs. Located under the **Operations | Laser Autofeatures** sub menu, these menu items remove the internal data, thereby helping to reduce measurement routine size:

- **Clear All Scan Data Now** - This menu item, once selected, immediately deletes all the internal clouds of points from all the laser Auto Features in the measurement routine.
- **Clear All Scan Data after Execution** - This menu item can take a check mark. By default, this menu item is unmarked but becomes marked when you first select it. If marked then any laser auto feature that executes will delete its internal clouds of points data following execution.



This only operates on internal clouds of points from Auto Features. It does not affect COP commands in the measurement routine.

Scanning Your Part Using a Laser Sensor

When you scan the surface of the part with a laser sensor, you can define an area of measurement. The software collects a group of point data that it passes to the reference Point Cloud object in the measurement routine. When you work with point clouds and scans, note that the scans themselves DON'T contain any data. They only define the machine movement. The Point Cloud object always stores the point data.

The main topics in this section discuss the scanning options available from the **Insert | Scan** submenu when you use a laser sensor:

- Introduction to Performing Advanced Scans
- Common Functions of the Scan Dialog Box
- Performing a Linear Open Advanced Scan
- Performing a Patch Advanced Scan
- Performing a Perimeter Advanced Scan
- Performing a Freeform Advanced Scan
- Performing a Grid Advanced Scan
- Performing a Surface Advanced Scan
- Performing a Manual Laser Scan on DCC Machines
- Setting Machine Speed for Scanning
- CWS Parameter Tab

Introduction to Performing Advanced Scans

Advanced scans are DCC continuous-motion scans that follow a predefined path. PC-DMIS follows the predefined path regardless of the shape of the actual part. The path can be defined in several ways that are explained later.

These advanced scans use a laser scanning probe. This enables you to automatically digitize surfaces.

To perform an advanced scan:

1. Specify the necessary parameters for the DCC scan that you selected.
2. Click the **Generate** button. PC-DMIS generates the scan.
3. Once it finishes, click the **Create** button. The PC-DMIS scanning algorithm then takes control of the measurement process.

The types of advanced scans that PC-DMIS supports include:

- Linear Open Scan
- Patch Scan
- Perimeter Scan
- Freeform Scan
- Grid Scan
- Manual Laser Scan on DCC Machines

This document first covers the common functions that are available in the **Scan** dialog box (the dialog box that you use to perform these scans). It then describes how to perform the available advanced scans.

For information on setting your machine's scan speed, see "Setting the Machine Speed for Scanning".

Common Functions of the Scan Dialog Box

Many of the functions described below are common to both DCC and Manual scans. Functionality that relates specifically to one scan mode is appropriately indicated.

Scan Type



Scan type list

Use the **Scan type** list in the **Scan** dialog box to change scan types without closing the dialog box and selecting a different scan type.

ID

The **ID** box in the **Scan** dialog box displays the ID of the scan to be created.

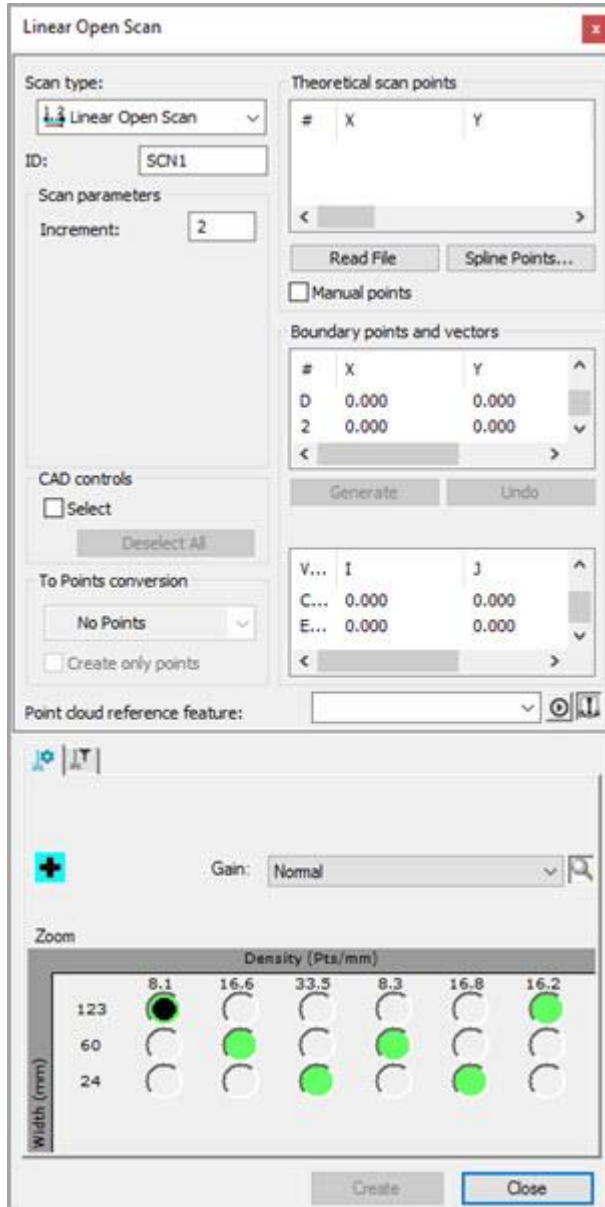
Scan Parameters Area

The **Scan parameters** area in the **Scan** dialog box provides different controls depending on the type of scan that is being performed. See the specific topics located under each scan type:

- Linear Open Scan Parameters
- Patch Scan Parameters
- Perimeter Scan Parameters
- Grid Scan Parameters

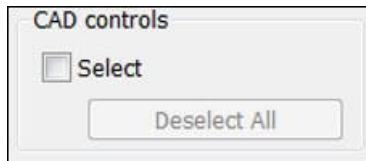
CAD Controls Area

Click the **Advanced >>** button in the **Scan** dialog box to display the full dialog box if necessary.



Scan dialog box for Linear Open Scan


Click the **Graphics** tab to display the **CAD controls** area. You can use this area to specify the CAD surface elements that define the "Theoretical Points".



CAD controls area

In some cases, a scan might start over a certain surface and travel over many other surfaces before completion. In such cases, PC-DMIS does not know which CAD elements to use to generate the scan. It must therefore search through every surface in the CAD model. If the CAD model has many surfaces, it might take a long time before the scan generation is successful.



To use this functionality to select CAD surfaces, you must have the ability to import and use CAD surface data. Ensure that you select the **Draw Surfaces** button (). If you don't, when you click on the CAD model, the nearest wire gets selected instead of the selected surface.

To avoid this delay:

1. Select the **Select** check box.
2. Click on the appropriate surfaces. Once a CAD surface is selected, it is highlighted in the Graphic Display window. The status bar displays the number of surfaces that you selected.

If you mistakenly select a surface, press Ctrl, and click on that surface a second time. This deselects the surface. Clicking the **Deselect All** button deselects all highlighted surfaces at once.

Once you are done selecting surfaces, clear the **Select** check box. The selected surfaces are kept.

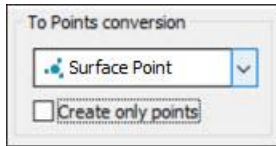
If you clear the **Select** check box, PC-DMIS assumes any clicks on the surface to be those that create the scan path.

The following options are available:

Select check box - Enables you to select the CAD surface and wireframe elements used to find the nominal.

Deselect All button - Deselects all of the highlighted surfaces at once that were created with the **Select** check box.

To Points Conversion Area



To Points conversion area

The **To Points conversion** area in the **Scan** dialog box enables you to create Point laser commands. The commands start from the points that make up the scan.

Hit Type List

The default setting is **No Points**.

For a Perimeter scan, you can select either Surface Point or Edge Point in the list. For all other types of scans, you can select only Surface Point.

The points are collected in a collapsed **GROUP** command. The name of the command includes the name of the related scan, the pointcloud associated with it, and the point ID preceded by "Edge" (if you selected Edge Point).

Surface Point Group Command Mode Text

Following is an example of a collapsed **GROUP** command collecting Surface Points:

```

COP                = COP/DATA,TOTAL SIZE=468492,REDUCED SIZE=468492,
                   FINDNOMS=NO,REF,SCN1,,
SCN1               = FEAT/SCAN,PERIMETER,NUMBER OF HITS=4,
                   SHOW HITS=NO,SHOWALLPARAMS=NO,POINTCLOUDID=COP
                   MEAS/SCAN
                   BASICSCAN/PERIMETER,NUMBER OF HITS=4,
                   SHOW HITS=NO,SHOWALLPARAMS=NO
                   ENDSCAN
                   ENDMEAS/
SCN1_COP_PNT_GRP1=GROUP/SHOWALLPARAMS=NO
                   EXECUTION CONTROL=AS MARKED
                   ENDGROUP/ID=SCN1_GRP1
  
```

Following is an example of a **GROUP** command collecting Edge Points:

```

SCN2          =FEAT/SCAN,PERIMETER,NUMBER OF HITS=3,SHOW
HITS=NO,SHOWALLPARAMS=NO,POINTCLOUDID=COP
              MEAS/SCAN
              BASICSCAN/PERIMETER,NUMBER OF HITS=3,SHOW
HITS=NO,SHOWALLPARAMS=NO
              ENDSCAN
              ENDMEAS/
SCN2_COP_EDGE PNT_GRP2=GROUP/SHOWALLPARAMS=YES
              EXECUTION CONTROL=AS MARKED
PNT5          =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN
              THEO/<133.992,0,0>,<0,-1,0>,<0,0,1>
              ACTL/<133.992,0,0>,<0,-1,0>,<0,0,1>
              TARG/<133.992,0,0>,<0,-1,0>,<0,0,1>
              DEPTH=0
              INDENT=1.5
              SPACER=0.5
              SHOW FEATURE PARAMETERS=NO
              SHOW_LASER_PARAMETERS=YES
              POINT CLOUD ID=COP
              SOUND=OFF
              HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=3
              REMOVE POINTS WITH NORMALS OUTSIDE=ON,10
PNT6          =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN
              THEO/<138.992,0,0>,<0,-1,0>,<0,0,1>
              ACTL/<138.992,0,0>,<0,-1,0>,<0,0,1>
              TARG/<138.992,0,0>,<0,-1,0>,<0,0,1>
              DEPTH=0
              INDENT=1.5
              SPACER=0.5
              SHOW FEATURE PARAMETERS=NO
              SHOW_LASER_PARAMETERS=YES
              POINT CLOUD ID=COP
              SOUND=OFF
              HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=3
              REMOVE POINTS WITH NORMALS OUTSIDE=ON,10

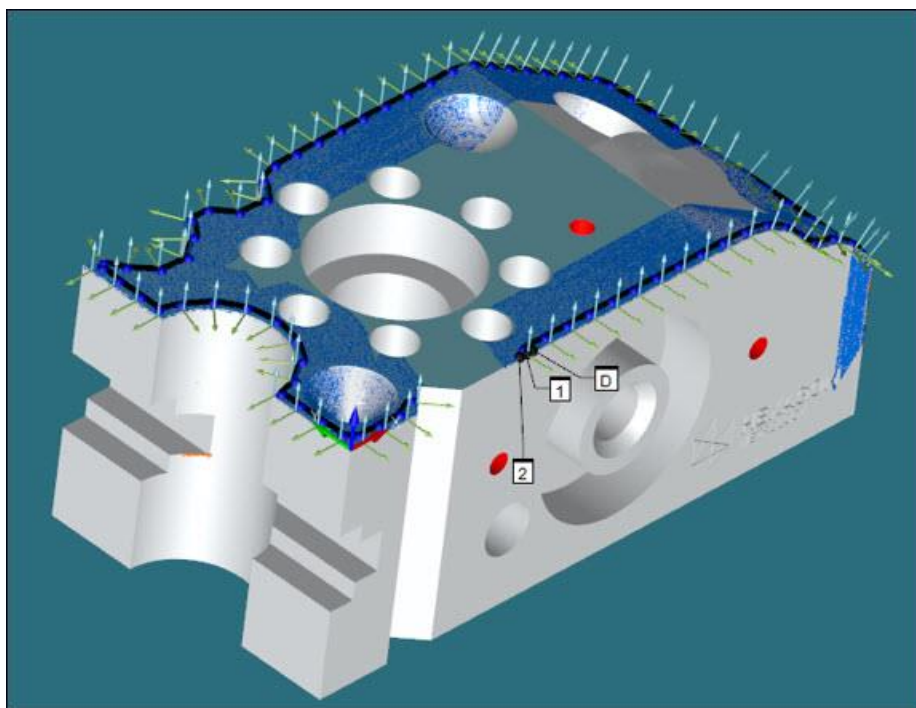
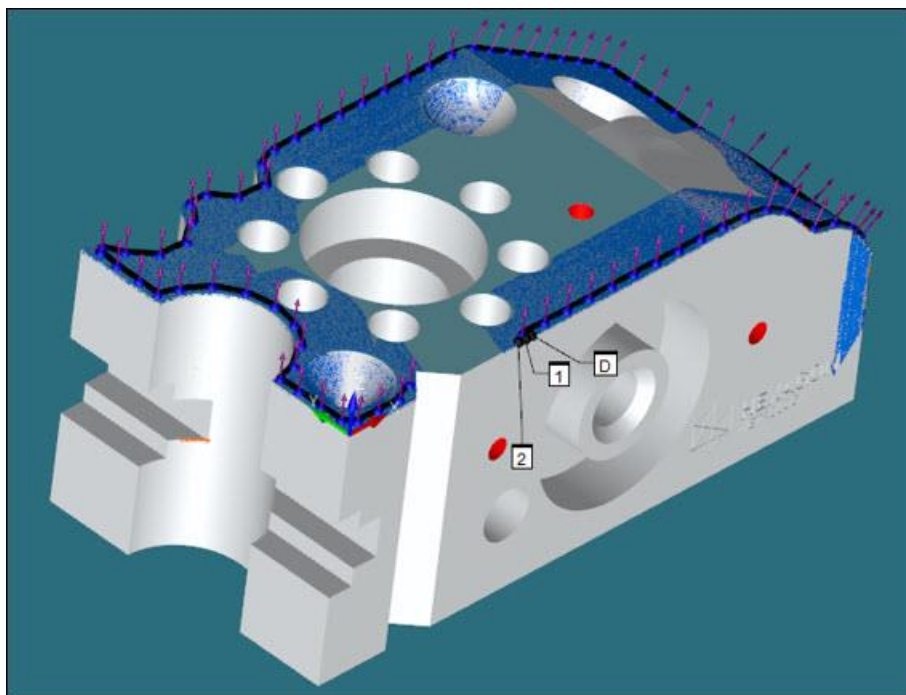
```

```
PNT7      =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN
          THEO/<143.992,0,0>,<0,-1,0>,<0,0,1>
          ACTL/<143.992,0,0>,<0,-1,0>,<0,0,1>
          TARG/<143.992,0,0>,<0,-1,0>,<0,0,1>
          DEPTH=0
          INDENT=1.5
          SPACER=0.5
          SHOW FEATURE PARAMETERS=NO
          SHOW_LASER_PARAMETERS=YES
          POINT CLOUD ID=COP
          SOUND=OFF
          HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=3
          REMOVE POINTS WITH NORMALS OUTSIDE=ON,10
          ENDGROUP/ID=SCN2_COP_EDGEPOINT_GRP2
```



Surface Points and Edge Points are extracted from the COP that you specified in the scan.

Consider the following figures that show Surface Points and Edge Points extracted from a COP using the **Scan** dialog box for a Perimeter scan:



Create only points

If you select the **Create only points** check box, PC-DMIS does not create the scan command. In this case, the [GROUP](#) command does not contain the name of the scan.



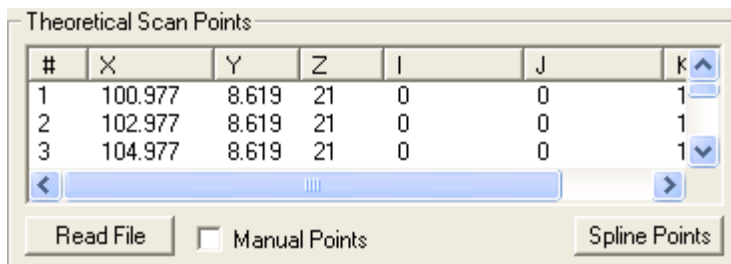
The **SCAN** command precedes the **GROUP** command in the Edit window if you create both commands.

Theoretical Scan Points Area

You can define the Theoretical Points of a scan through any of these actions:

- Reading them from a file
- Reading machine positions
- Generating them from the defined boundary points
- Using CAD data

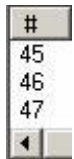
These topics are discussed in greater detail later in this section.



Theoretical Scan Points area

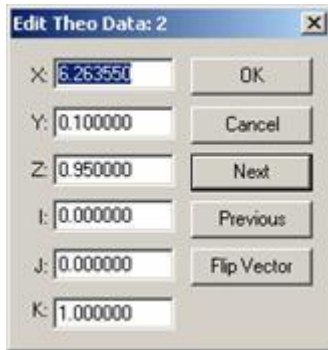
Editing Theoretical Points

To edit theoretical points, double-click the number of the desired point in the **#** column.



column

This displays the **Edit Theo Data** dialog box. Use this dialog box to edit the X, Y, Z, I, J, K values. The dialog box's title bar displays the ID of the point you are editing.



Edit Theo Data dialog boxes depicting Next, Previous, and Flip Vector buttons

Click the **Next** or **Previous** buttons to cycle between the theoretical points.

Click the **Flip Vector** button to flip the vector for the selected point.

Deleting Theoretical Points

You can easily clear the **Theoretical Points** list of any of the scan types. Right-click inside the **Theoretical Points** list. A **Reset Theoretical Points** prompt appears. Click the prompt to clear any points from the list.

Read File

The **Read File** button tells PC-DMIS to read the Theoretical points in from a text file. The points must be in X,Y,Z,I,J,K comma delimited format. A blank space between points denotes the start of a new scan line.

Manual Points

By selecting the **Manual Points** check box, you can manually add points into the **Theoretical Points** list. To take these points, move the probe to the desired location and click the **Probe Enable** button on your jog box, or click points on the CAD file.

New Line

The **New Line** check box functions only for Patch Scans. By selecting the **New Line** check box, you tell PC-DMIS that manual points you take should begin a new line.

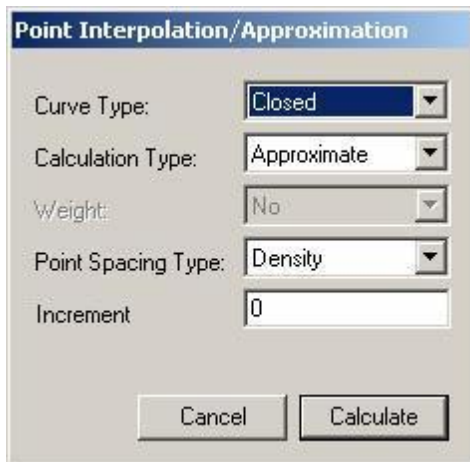
Spline Points

When you take manual points the spacing and path are usually inconsistent. With the **Spline Points** button, however, you can construct a spline curve along a path through a list of manual points and create a smooth, evenly spaced path. For a Linear Open scan PC-DMIS places all the points on the Cut Plane. For a Patch Scan it places the points for each scan line on the Cut Plane for that scan line.



The **Spline Points** button is not available for a Perimeter scan.

Clicking the **Spline Points** button displays the **Point Interpolation / Approximation** dialog box.



Point Interpolation/Approximation

Curve Type

There are three types of curves that can be constructed with the spline routines:

Open - This option creates an open ended curve. This means the curve starts in one location and ends in another.

Closed - This option creates a close ended curve. This means the curve starts and ends in the same location.

Line - This option differs from the **Open** or **Closed** options. It doesn't use theoretical points but instead uses boundary points and creates straight lines within the boundary points, following the boundary points' direction rules.

Calculation Type

There are two calculation types you can use in spline routines.

Approximate: This option allows the path to deviate from the actual input point by a small amount in order to produce a smooth curve from which the new points are taken.

Interpolate: This option forces the curve to go through each of the input points exactly.

Weight

This list becomes available when you select the **Approximate** calculation type. When constructing the curve, it allows more weight to be given to points that are further apart. The two choices for the option are **YES** and **NO**.

Point Spacing Type

This option allows you to control the output points of the spline routine.

Density: This option lets you specify the incremental distance between each output point. PC-DMIS determines the number of output points by the length of the curve and the user supplied increment.

Number of Hits: This option lets you specify how many points they want in the output. No matter the length of the curve, PC-DMIS evenly spaces the user supplied points over the length of the curve.

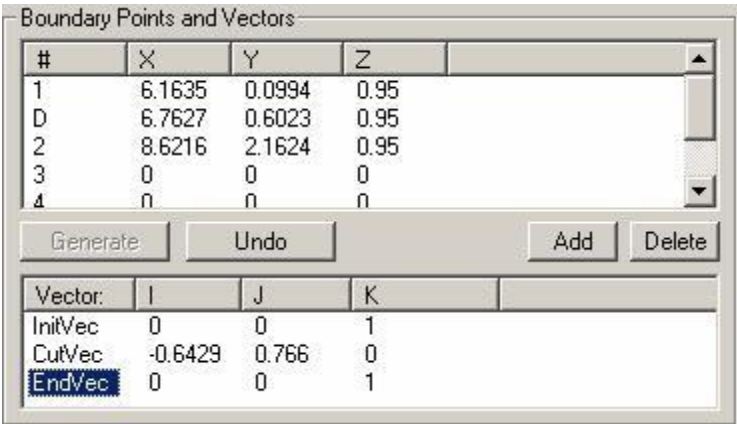
Increment

This box holds the increment value for the Point Spacing Type; either **Density** or **Number of Hits**.

Boundary Points Area

PC-DMIS lets you define the boundary of a scan. You can do this in these ways:

- Type the XYZ values for the individual boundary points directly
- Measure the points using the laser sensor
- Use the CAD data



Boundary Points and Vectors area



Boundary Points are not available or necessary for Free-form scans

You can change the column widths of the **Boundary Point** list if you click and drag the right or left edge of a column header to the desired size. The software saves this information to your PC-DMIS Settings Editor each time it changes.

Setting Boundary Points by Typing

To set the boundary of a scan by typing:

1. Double-click the desired boundary point in the '#' column to display the **Edit Scan Item** dialog box.



Edit Scan Item dialog box

2. Manually edit the X, Y, or Z value.
3. Click the **OK** button to apply the changes.

Click **Next** to accept the changes and display the next boundary point for editing.

Setting Boundary Points Using the Measured Point Method

To set the boundary of the scan using measured points:

1. Place the laser sensor at the desired location.
2. On the jog box, press the **Probe Enable** button (only available on DEA and Brown and Sharpe machines).
 - This automatically updates the value of the selected boundary point in the **Boundary Points and Vectors** list. The software then selects the next boundary point (if any) in the list.
 - In the case of a PATCH scan, PC-DMIS adds an extra boundary point automatically if the selected point is the last point in the list. The PATCH scan displays the last point (this is the same as the previous point). PC-DMIS deletes this last point when you click the **OK** button.




The **Probe Enable** light on the jog box alternates between off and on every time you press the **Probe Enable** button. This is not important and has no influence on the probe itself.

Setting Boundary Points Using the CAD Data Method

PC-DMIS lets you select the boundary points by using surface CAD data.

When using CAD surface data:

1. Make sure that you have imported solid CAD data.
2. Select that the **Draw Surfaces** icon .
3. Select a boundary point by clicking on the desired location in the Graphic Display window. PC-DMIS highlights the selected surface and automatically updates the value of the currently selected Boundary point. PC-DMIS then moves the focus to the next boundary point (if any are available). For PATCH scans, PC-DMIS automatically adds an extra boundary point if the current point is the last point in the list.

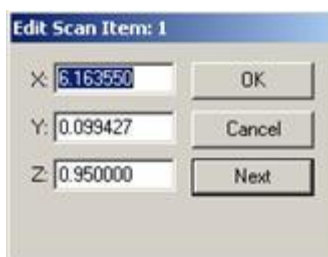
Editing Boundary Points

Boundary points can be edited by double clicking the number of the desired point in the '#' column.



column

This displays the **Edit Scan Item** dialog box allowing you to edit the X, Y, Z values.



Edit Scan Item dialog box

Clearing Boundary Points

You can easily clear the **Boundary Points** list of any of the scan types.

1. Right-click while the cursor is inside the **Boundary Points** list.
2. Click the **Reset Boundary Points** button that appears to reset all the boundary points to zero. The number of boundary points is set to the minimum for each scan type.

Generate

The **Generate** button is available only for DCC scans using CAD data.

After the boundary points for a scan has been defined, click the **Generate** button. PC-DMIS slices the CAD with the plane defined by the start point and cut vector and then generate the theoretical points from the curve defined by this slice. If the **Create** button is then clicked, PC-DMIS inserts a scan with nominal hit data is inserted into the measurement routine.

Undo

The **Undo** allows you to remove the hits that have been generated using the **Generate** button as outlined in the Generate topic.

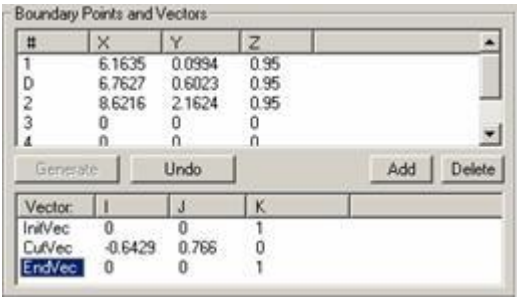
Adding and Deleting Boundary Points



Add / Delete buttons

The **Add** and **Delete** buttons allow you to add or delete boundary points to the list of boundary points. There are some restrictions regarding each type of scan. For example, a LINEAROPEN scan only takes a Start Point, a Direction point and an Ending Point. It will not allow you to add more points or delete these points. Refer to each scan for specific restrictions.

Vectors Area



Boundary Points and Vectors area

The bottom portion of the **Boundary Points and Vectors** area displays a list of vectors that PC-DMIS will use to start and stop a scan. Some of the vectors listed below may not be found in the list for a specific scan, indicating that they are not used for that scan. Please refer to each scan for more details. You can edit each of these vectors by double-clicking on the vector to edit in the vector column.



Vector column

This displays the **Edit Scan Item** dialog box:



Edit Scan Item dialog box

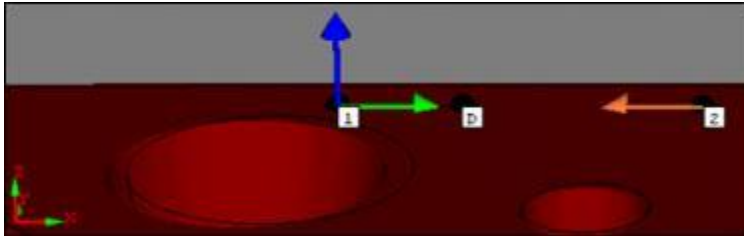
With the I, J, and K boxes, you can edit the I, J, and K values.

- **Next** - This button cycles through the available vectors in the **Initial Vectors** list. Some of the initial vectors can be flipped. If so, then the **Flip** button becomes available on the **Edit Scan Item** dialog box.
- **Flip** - This button reverses the direction of the selected vector.

Graphical Representation of Vectors

When setting up the start, direction, and end points of the scan, PC-DMIS allows you to see a graphical representation of the initial touch vector, the direction vector, and the vector that is normal to the boundary plane where the scan will stop.

These vectors are shown as blue, green and orange colored arrows in the Graphical Display area of your part.



Colored arrows showing vectors

Vector	Graphical Representation
Initial Touch	Blue arrow
Direction	Green arrow
Boundary Plane	Orange arrow

Initial Touch Vector (InitVec)

The values that are displayed in the **Initial Touch Vector** row indicate the vector PC-DMIS will use to take the first touch in the scanning process.

To edit the I, J, K Initial Touch vector:

1. Double-click on **InitVect** in the vector column to open the **Edit Scan Item** dialog box.
2. Change the values.
3. Click the **OK** button to accept the changes and close the dialog box.

Cut Plane Vector (CutVec)

A cut plane is used internally for DCC scanning calculations. This cut plane is derived from the Initial Touch vector, and the vector between the first and last points for the Linear Open DCC scan. Please refer to the individual scans for details on how the Cut Plane vector is derived.

End Touch Vector (EndVec)

The End Touch vector is the approach vector of the scan at the end of the row. This is used only to stop the scan or to move to the next row (in the case of a Patch scan).

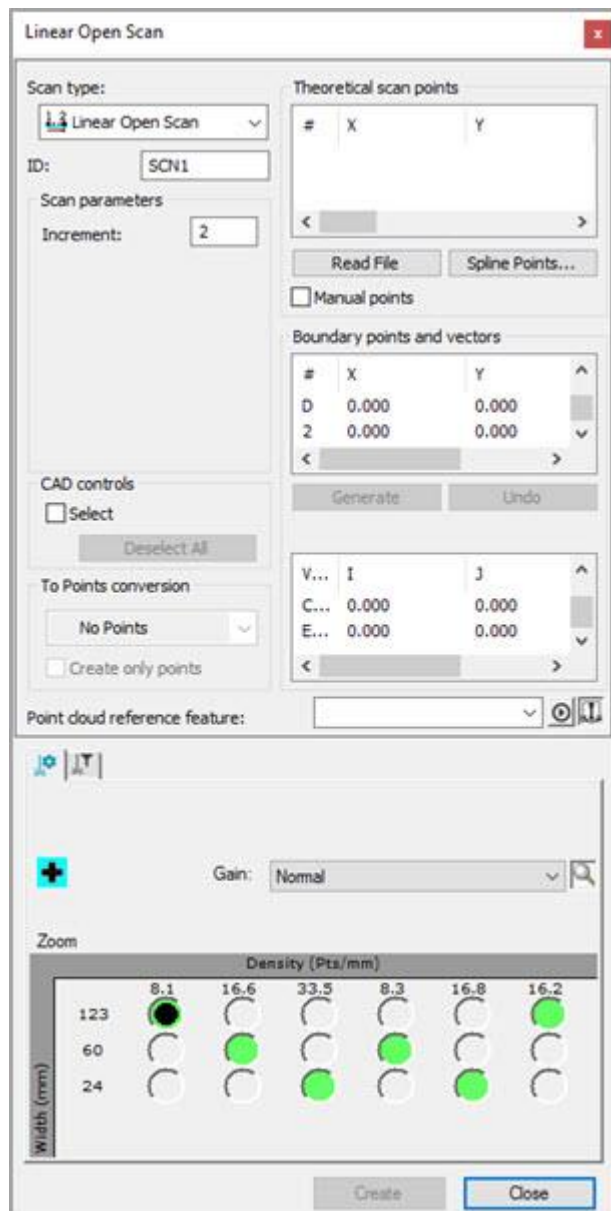
Point Cloud Reference Feature

The **Point Cloud Reference Feature** defines the Point Cloud object into which PC-DMIS places the surface data. Select the needed Pointcloud from the combo box to which data will be added. This field must be supplied, or PC-DMIS cannot create the scan.

Measure

If you select the **Measure** check and click the **Create** button, PC-DMIS will start measuring the scan immediately. If you don't select the **Measure** check box when you click **Create**, PC-DMIS inserts a scan object into the Edit window that can be measured later. This allows you to set up a series of scans that can be inserted into the Edit window and measured at a later time.

Performing a Linear Open Advanced Scan



Linear Open Scan dialog box

The **Linear Open Scan** method scans the surface along a line. This procedure uses the starting and ending points for the line, and also includes a direction point for the calculation of the cut plane. The probe always remains within the cut plane while doing the scan.

To Create a Linear Open Scan

1. Ensure that you have a Laser probe enabled.
2. Place PC-DMIS into DCC mode.
3. Select the **Insert | Scan | Linear Open** menu item. The **Scan** dialog box appears with the **Linear Open Scan** option already selected in the **Scan type** list.
4. If your scan traverses multiple surfaces, consider selecting surfaces as discussed in the "CAD Controls" topic. To access these controls, click the **Advanced >>** button in the upper-right corner of the dialog box if necessary, and then click the **Graphics** tab at the bottom.
5. If you are going to use the boundary points to help define the scan path, add the 1 point (starting point), the D point (direction to scan), and the 2 point (ending point) to the scan by following the appropriate procedure discussed in the "Boundary Points" topic.
6. Make any needed changes to the vectors in the **Vectors** list by double-clicking on the vector. Make any changes in the **Edit Scan Item** dialog box, and then click **OK** to return to the **Scan** dialog box.
7. Type the name of the scan in the **ID** box.
8. Select the **Measure** check box if needed.
9. Set the distance between generated theoretical points in the **Increment** box.
10. Select the method for defining the scan path from the **Read File**, **Manual Hits**, **Generate**, and **Spline Points** options.
11. If needed, you can delete individual points. To delete them, select them one at a time in the **Theoretical Path** area and press the Delete key on your keyboard.
12. If needed, make additional modifications to your scan.
13. In the **Point cloud reference feature** box, type the ID of the cloud of points object that will receive the surface data.
14. In the **Hit Type** list, you can select **Surface Point** in case you want to convert scan data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.



WARNING: Once you mark the **Measure** check box, and click **Create**, you must keep clear of the machine. The software starts the measurement routine and the machine will move. Injury may result if you are not clear of the machine.

15. Click the **Create** button. PC-DMIS inserts the scan into the Edit window if the **Create only points** check box is cleared.

Scan Parameters

The **Increment** box in the **Scan Parameters** area lets you set the increment distance between the theoretical points when you click the **Generate** button.

Vectors

Vectors used:

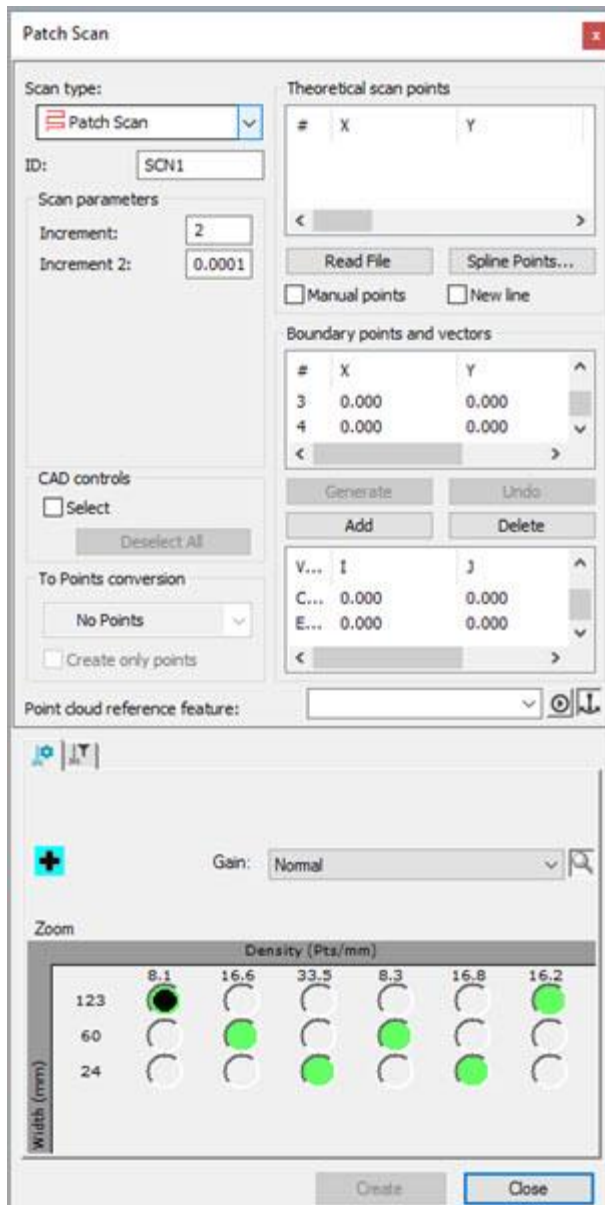
- Cut Plane (CutVec)
- Initial Touch (InitVec)
- End Touch (EndVec)

For additional information, see "Vectors" under "Common Functions of Scan Dialog Boxes".



The Cut Plane vector (CutVec) is the cross product of the Initial Touch vector(InitVec) and the line between the start and end point.

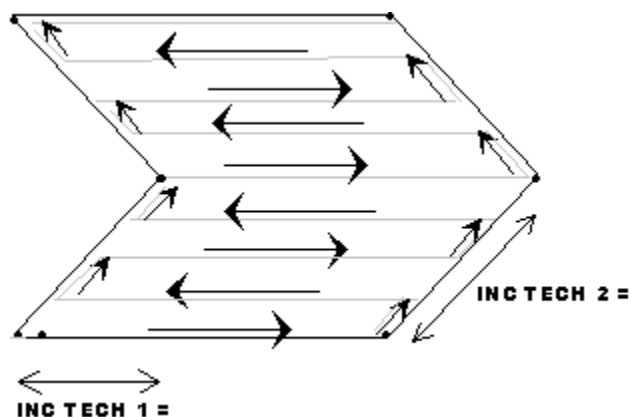
Performing a Patch Advanced Scan



Patch Scan dialog box

The Patch scan is like a series of Linear Open scans done parallel to each other.

The **Patch Scan** method scans the surface of the part based on the Scan Parameters. The probe always remains within the cut plane while doing each scan line. It uses the **Increment** value to determine the distance between points on each line. When the scan reaches the boundary at the end of a line, the scan moves to the next line by the **Increment 2** value and starts a new scan line that moves in the opposite direction. The following figure describes this process.



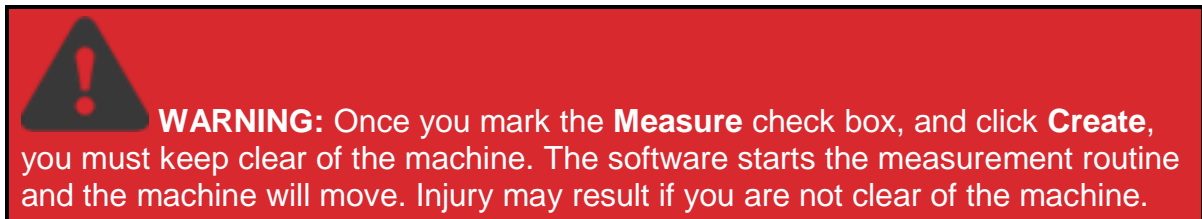
Patch scan increment example

To Create a Patch Scan

1. Ensure that you have a Laser probe enabled.
2. Place PC-DMIS into DCC mode.
3. Select the **Insert | Scan | Patch** menu item. The **Scan** dialog box appears with the **Patch Scan** option already selected in the **Scan type** list.
4. Set the values for **Increment** and **Increment 2**. These determine the spacing of the points if you select the **Generate** or **Spline** buttons or the **New Line** check box to define the scan. **Increment** defines the spacing between each point on a scan line and **Increment 2** defines the spacing between scan lines.
5. If your scan traverses multiple surfaces, consider selecting surfaces as discussed in the "CAD Controls" topic.
6. If you are going to use the boundary points to help define the scan path, add the 1 point (starting point), the D point (the direction to begin scanning), the 2 point (the end point of the first line), the 3 point (to generate a minimum area), and, if desired, the 4 point (to form a square or rectangular area). This will select an area that you wish to scan. Pick these points by following the appropriate procedure discussed in the "Boundary Points" topic.
7. Make any needed changes to the vectors in the **Vectors** list. To do this, double-click on the vector, make any changes to the **Edit Scan Item** dialog box, and then click **OK** to return to the **Scan** dialog box.
8. Type the name of the scan in the **ID** box.
9. Mark the **Measure** check box if you want to execute the scan and measure it at creation time.
10. Select the **Generate** button to generate a preview of the scan on the CAD model in the Graphic Display window. When you generate the scan, PC-DMIS starts the scan at the start point and follows the chosen direction until it reaches the

boundary point. The scan then moves back and forth scanning in rows along the chosen area and at the specified increment value until it finishes the process.

11. If needed, you can delete individual points. To delete them, select them one at a time in the **Theoretical Path** area and press the Delete key on your keyboard.
12. If needed, make additional modifications to your scan.
13. In the **Point cloud reference feature** box, type the ID of the cloud of points object that will receive the surface data.
14. In the **Hit Type** list, you can select **Surface Point** in case you want to convert scan data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.



15. Click the **Create** button. PC-DMIS inserts the scan into the Edit window if the **Create only points** check box is cleared.

Patch Scan Parameters

The **Increment** and **Increment 2** boxes described below are available when creating and measuring a **Patch** scan.

Increment

The **Increment** allows you to set the increment distance between each point when Generate or Spline/Line is used to define the scan path.

Increment 2

The **Increment 2** allows you to set the increment distance between scan lines when Generate or Spline/Line is used to define the scan path.

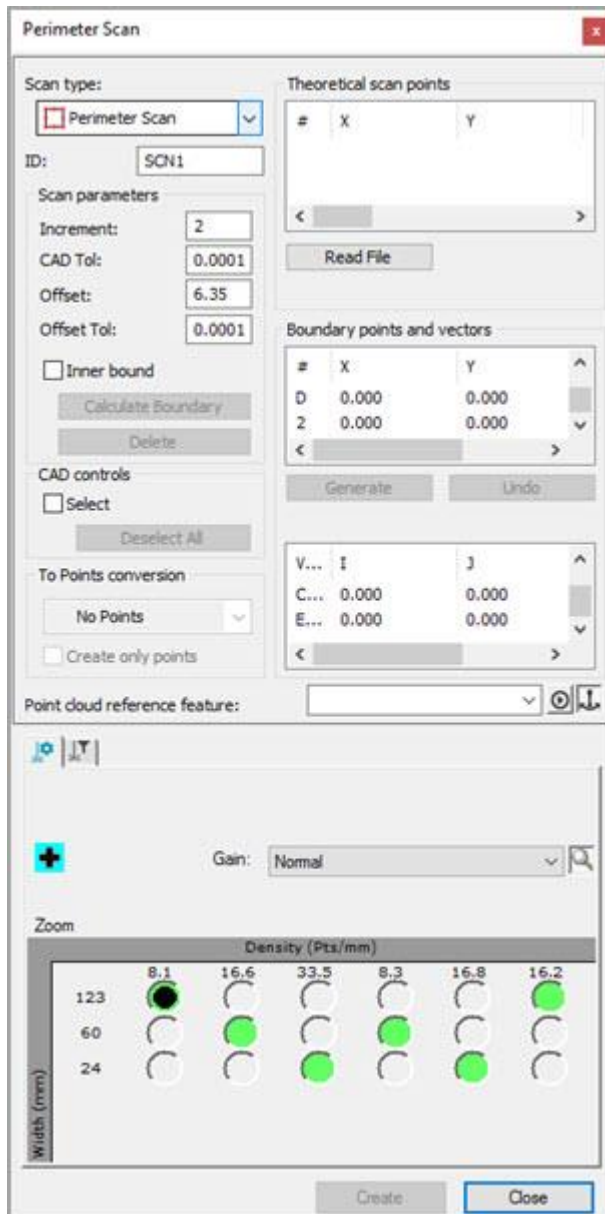
Initial Vectors

Vectors used:

- Cut Plane (CutVec)
- Initial Touch (InitVec)
- End Touch (EndVec)

The cut plane vector is derived by crossing the Initial Touch vector (InitVect) and the line between the first and second point. The cut plane vector is then set to the correct direction by using the line between the second and third points. The End Touch vector (EndVec) is the vector used to take the second boundary points and is used to jump to the second row after completing the first row.

Performing a Perimeter Advanced Scan



Perimeter Scan dialog box

The **Perimeter Scan** method scans the surface of the part based on the selected surfaces. This procedure traverses the selected surfaces within the created boundaries.

To Create a Perimeter Scan

To create a Perimeter scan:

1. Ensure that you have a Laser probe enabled.
2. Place PC-DMIS into DCC mode.
3. Select the **Insert | Scan | Perimeter** menu item. The **Scan** dialog box appears with **Perimeter Scan** already selected in the **Scan type** list.
4. Select the surface or surfaces to use to create the boundary. If you select multiple surfaces, you should select the surfaces in the same order that they are to be traversed by the scan. To select the necessary surface or surfaces:
5. Verify that the **Select** check box is selected. Each surface will be highlighted as it is selected.
6. After you select the desired surfaces, clear the **Select** check box.
7. Click on the surface near the boundary where the scan is to begin. This is the Start Point.
8. Click on the same surface a second time in the direction that the scan is to be executed. This is the Direction Point.
9. Click the point where the scan is to end. This point is *optional*. If you do not provide an End Point, the scan ends at its Start point.
10. Type the appropriate values into the **Scan parameters** area. These include the following boxes:
 - **Increment** box
 - **CAD Tol** box
 - **Offset** box
 - **Offset Tol (+/-)** box
11. Select the **Calculate Boundary** button to calculate the boundary from which the scan will be created. The red dots on the boundary indicate where the hits are taken on the perimeter scan.



The boundary calculation should be a relatively quick process.

If the boundary does not look correct, click the **Delete** button. This will delete the boundary and allow another to be created.

If the boundary appears incorrect, it usually means that the CAD tolerance needs to be increased.

After changing the CAD tolerance, click the **Calculate Boundary** button to recalculate the boundary.

Verify that the boundary is correct before calculating a perimeter scan because it takes much longer to calculate the scan path than it does to recalculate the boundary.

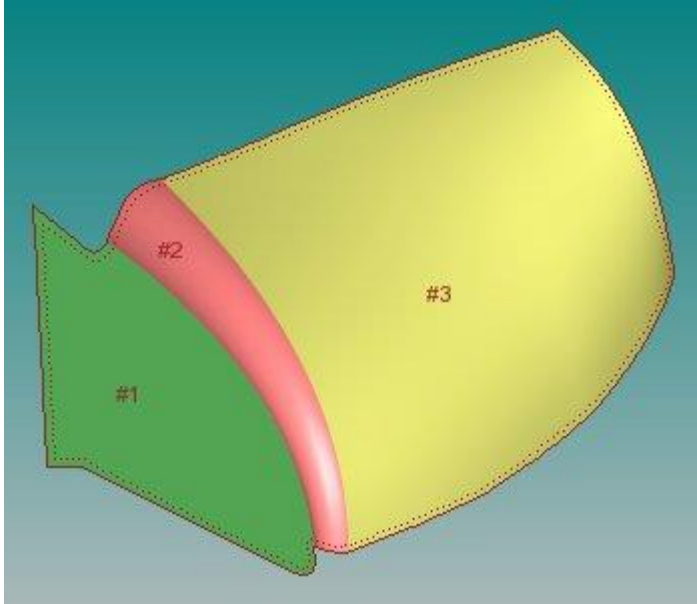
12. Verify that the **Offset** value is correct.
13. Click the **Generate** button. PC-DMIS calculates the theoretical values used to execute the scan. This process involves a very time intensive algorithm. Depending on the complexity of the selected surfaces and the amount of points that are being calculated, it may take a while to compute the scan path. (A five-minute wait is common.) If the scan does not appear to be correct, you can use the **Undo** button to delete the proposed scan path. If needed, you can alter the offset tolerance and recalculate the scan.
14. If needed, you can delete individual points by selecting them one at a time in the **Theoretical Path** area and pressing the Delete key on your keyboard.
15. In the **Point cloud reference feature** box, type the ID of the cloud of points object that will receive the surface data.
16. In the **Hit Type** list, you can select **Surface Point** or **Edge Point** in case you want to convert scan data into Surface Point or Edge Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.



WARNING: Be aware that if the **Measure** check box is marked, the machine begins to move as soon as you click **Create**. Make sure you are well clear of the machine to avoid injury.

17. Click the **Create** button to store the perimeter scan in the Edit window if the **Create only points** check box is not selected. It is executed like any other scan. If you have the PC-DMIS AutoWrist method enabled but do not have any calibrated tips, PC-DMIS displays a message that informs you when it adds new probe tips that need calibration. In all other cases, PC-DMIS prompts you whether it should use the closest calibrated tip to the needed tip angle or add in a new non-calibrated tip at the needed angle.

Three surfaces have been selected. Each surface borders another, but the outside of each surface makes up the composite boundary (indicated by the solid line) The offset distance is the amount that the scan will be offset from the composite boundary (indicated by the dotted line)



Perimeter Scan Example

Perimeter Scan Parameters

Scan parameters	
Increment:	<input type="text" value="2"/>
CAD Tol:	<input type="text" value="0.01"/>
Offset:	<input type="text" value="6.35"/>
Offset Tol:	<input type="text" value="0.01"/>
<input type="button" value="Calculate Boundary"/>	
<input type="button" value="Delete"/>	

Scan Parameters area

The **Scan Parameters** area of the dialog box allows various options for constructing a Perimeter Scan. These include:

Increment

The **Increment** box indicates the distance between each of the hit points on the scan.

CAD Tol

The **CAD Tol** box is useful in detecting neighboring surfaces. The larger the tolerance, the farther apart the CAD surfaces can be and still be recognized as a neighboring surface.

Offset

The **Offset** box indicates the distance in from the perimeter where the scan will be created and executed.

Offset + / -

The **Offset Tol (+/-)** box indicates the amount of allowable deviation from the offset value. It is a user supplied value.

Calculate Boundary

The **Calculate Boundary** button determines the composite boundary of the input surfaces. The Calculated boundary appears as red dots in the Graphic Display window.

Delete

The **Delete** button deletes the previously created boundary.

Performing a Freeform Advanced Scan



Freeform Scan dialog box

The **Freeform Scan** method defines a scan path that is not restricted to following any particular rule set. You can define the scan path to move in any direction, including crossing back over itself.

Creating a Freeform Scan

1. Place PC-DMIS into DCC mode.

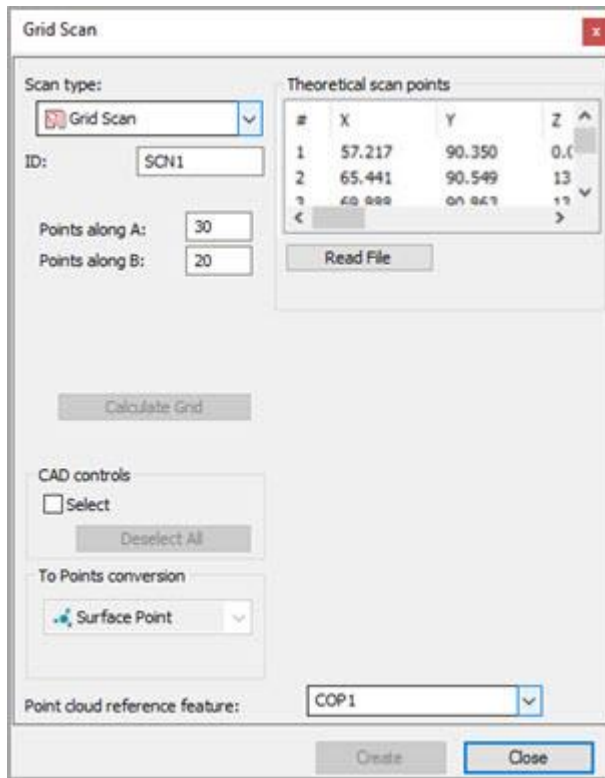
2. Select the **Insert | Scan | Freeform** menu item. The **Scan** dialog box appears with the **Freeform Scan** already selected in the **Scan type** list.
3. You need to define the scan path. You can do this by using the **Read File** option or by the **Manual Points** method.
4. If needed, you can delete individual points. To delete them, select them one at a time in the **Theoretical Path** area and press the Delete key on your keyboard.
5. Once five or more **Theoretical Points**, use the **Spline Points** option to better define the path.
6. If needed, make additional modifications to your scan.
7. In the **Point cloud reference feature** box, type the ID of the cloud of points object that will receive the surface data.
8. In the **Hit Type** list, you can select **Surface Point** in case you want to convert scan data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.



WARNING: Once you mark the **Measure** check box, and click **Create**, you must keep clear of the machine. The software starts the measurement routine and the machine will move. Injury may result if you are not clear of the machine.

9. Click the **Create** button. PC-DMIS inserts the scan into the Edit window if the **Create only points** check box is not selected. If you have the PC-DMIS AutoWrist method enabled but don't have any calibrated tips, PC-DMIS displays a message that informs you when it adds new probe tips that need calibration. In all other cases, PC-DMIS asks you whether it should use the closest calibrated tip to the needed tip angle or add in a new non-calibrated tip at the needed angle.

Performing a Grid Advanced Scan

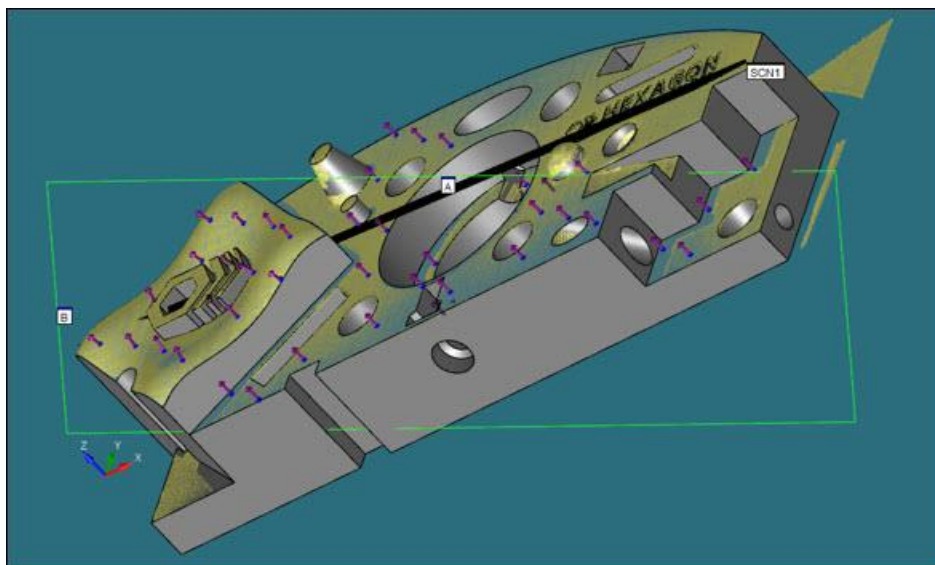
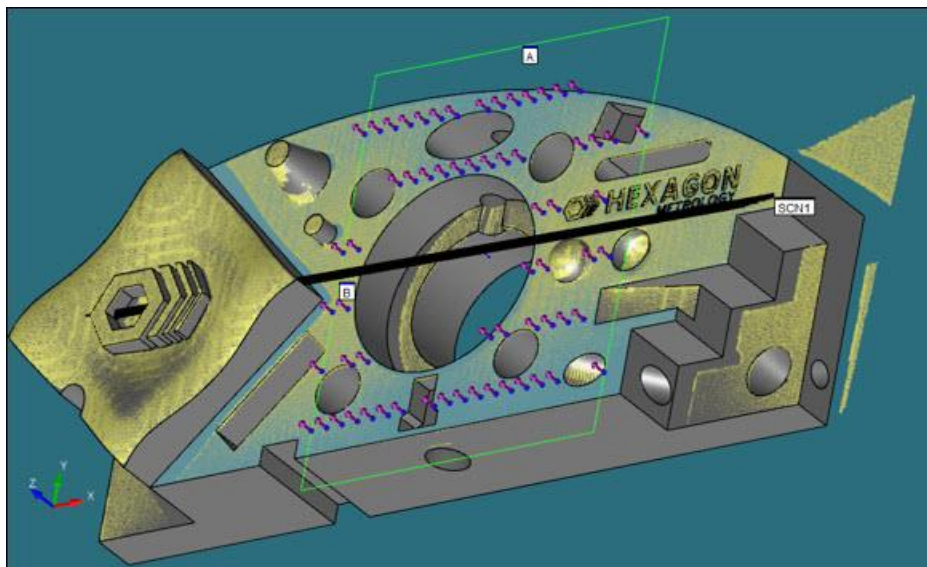


Grid Scan dialog box

The **Grid Scan** method creates a grid of points within a visible rectangle and then projects those points down on top of any selected surfaces. The rectangle and, consequently, the grid of points, depend on the orientation of the CAD model in the **CAD** tab.

Use the **Hits along A** and **Hits along B** boxes to define how many hits inside the boundary will get spaced and dropped onto the selected surface or surfaces.

Consider the following figures that show grid Surface Points extracted from a COP:

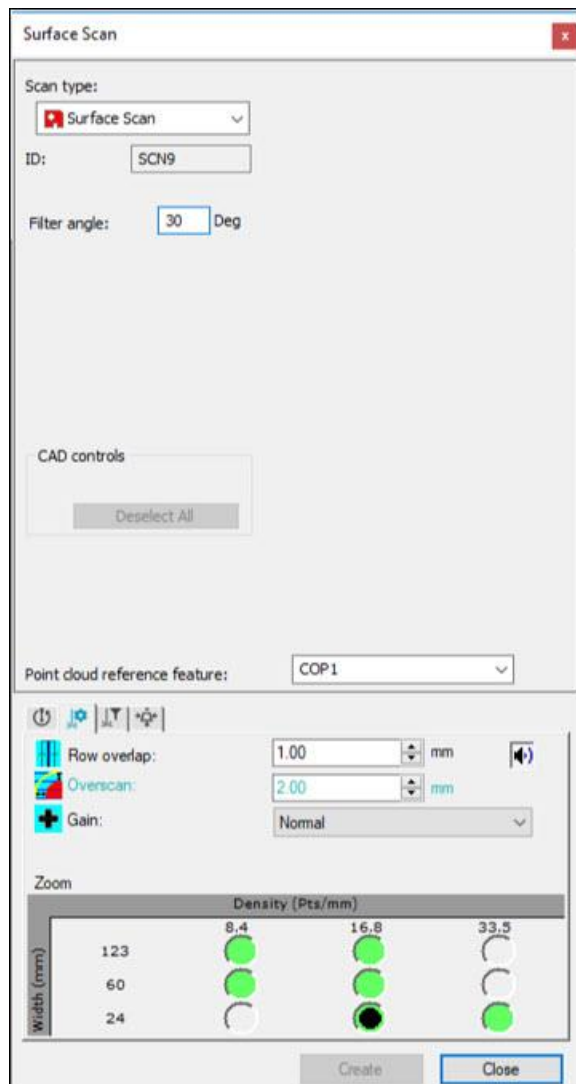


Creating a Grid Scan

1. Ensure that you have a Laser probe enabled.
2. Place your CAD model into Solid mode.
3. Place PC-DMIS into DCC mode.
4. Select the **Insert | Scan | Grid** menu item. The **Scan** dialog box appears with the **Grid Scan** already selected in the **Scan type** list.
5. If you want to use a custom name for the grid, type the name of the grid in the **ID** box.

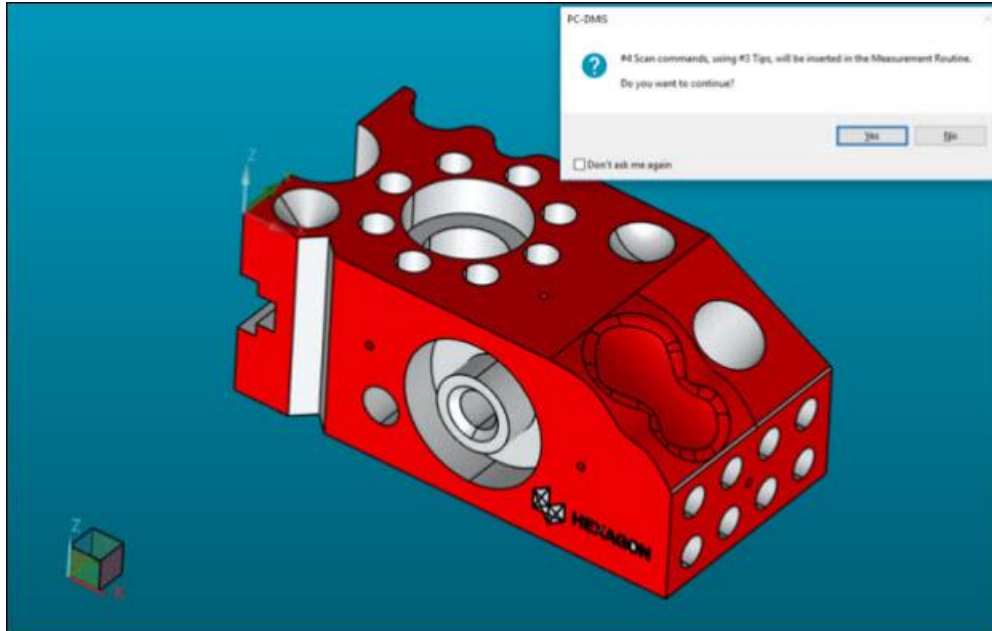
6. In the **Hits along A** and **Hits along B** boxes, specify how many hits in the A and B directions will get spaced and dropped onto the selected surface or surfaces.
7. Click and drag a rectangle on the screen over the surface or surfaces you want to include in your scan. This rectangle defines the boundary of the grid, which will be projected onto the CAD surface or surfaces. PC-DMIS draws points on the CAD model on any surfaces that were selected when you drew the rectangle.
8. Mark the **Select** check box if you want to deselect some surfaces. PC-DMIS highlights the selected surfaces and draws points only on them. It does not draw points on any deselected surfaces, even if they are included in the boundary of the rectangle.
9. If you mistakenly select a surface, press Ctrl and click on that surface a second time to deselect it. To deselect all of the highlighted surfaces at once, click the **Deselect All** button.
10. To recompute grid points (that is, to apply different A and B values on the selected surfaces), select the **Calculate Grid** button at any time.
11. In the **Point cloud reference feature** box, type the ID of the COP object from which to extract the surface data.
12. In the **Hit Type** list, **Surface Point** is the only option available since the scope of dialog box is to convert grid data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.
13. Click the **Create** button. PC-DMIS inserts the Surface Point laser commands into the Edit window in a collapsed **Group** command.

Performing a Surface Advanced Scan



Surface Scan dialog box

The **Surface Scan** dialog box creates a series of scans to cover a selection of surfaces. For example:

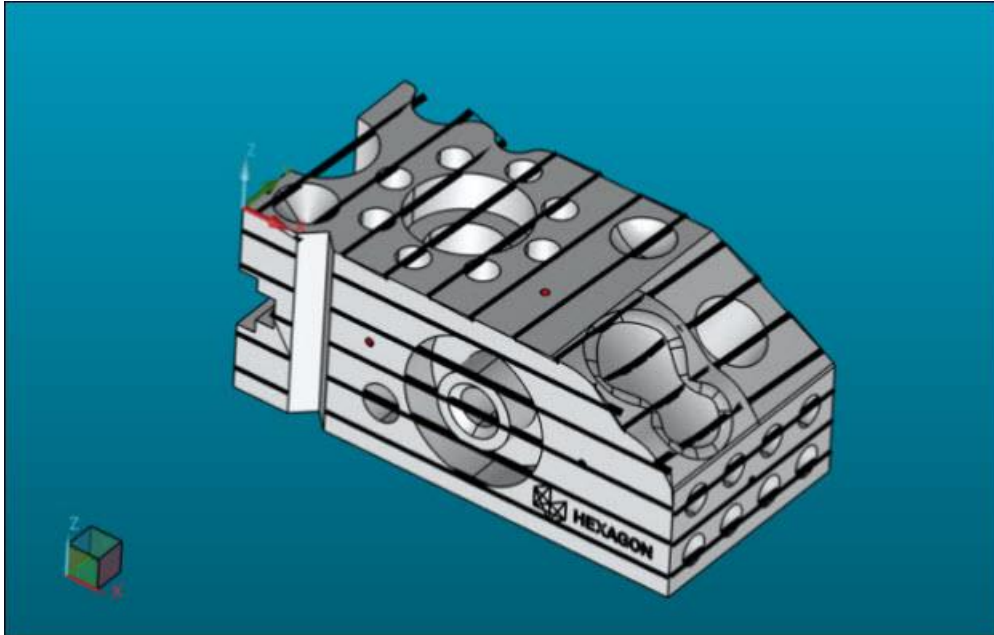


Example of a Surface Advanced scan

Creating a Surface Advanced Scan

1. Ensure that you have a Laser probe enabled.
2. Place PC-DMIS into DCC mode.
3. Verify that the CAD vectors on the surfaces are correctly defined. If necessary, you can adjust them in the **CAD Vectors** dialog box (**Edit | Graphic Display Window | CAD Vectors**). For more information, see the "Editing CAD Vectors" topic in the "Editing the CAD Display" chapter in the PC-DMIS Core documentation.
4. Create a COP in the **Pointcloud** dialog box (**Insert | Pointcloud | Feature**). For details on pointcloud features and how to create a COP, see the "Using Pointclouds" chapter.
5. Set the appropriate scan speed. For details, see the "Getting Started" chapter.
6. Select the **Insert | Scan | Surface Scan** menu item. The **Scan** dialog box appears with the **Surface Scan** already selected in the **Scan type** list.
7. In the **Filter Angle** box, type the appropriate filter angle.
8. In the **Point cloud reference feature** list, select the pointcloud reference feature.
9. Open the Probe Toolbox (**View | Probe Toolbox**). Do the following:
 - On the **Laser Scan Properties** tab, select the appropriate scan settings.
 - On the **Laser Filtering Properties** tab, select the appropriate filter settings for the scan.
 - On the **Laser Clipping Region Properties** tab, select the appropriate clipping parameters for the scan.

10. In the Graphic Display window (**View | Graphic Display Window**), select the surfaces on the CAD that the surface scan should cover. For help, see the "Editing the CAD Display" chapter in the PC-DMIS Core documentation.
11. Click the **Create** button. PC-DMIS inserts the necessary scans into the Edit window. For example:



Example of a Surface Advanced scan in the Edit window

Note the following:

- The surface scan is a one-time operation to create a list of scans.
- You cannot edit these scans.
- A verification if the scans can execute collision-free is necessary.
- A full coverage on the selected surfaces depends on their complexity. Full coverage is detectable if you execute the scans offline.
- The computation time for the scans depends on the complexity of the selected surfaces.

Performing a Manual Laser Scan on DCC Machines

Manual laser scans on DCC machines only work on FDC controllers and therefore only on Bridge machines with indexable heads. Manual laser scanning functionality is not available on horizontal arms with CW43L wrists.

To create a manual laser scan on a DCC machine:

1. Start PC-DMIS online with a laser sensor.

2. From the main menu, select **File | New** to start the machine in **Manual** mode.
3. Press the **Probe Enable** button on the jog box (you only need to press the button once, regardless of the button's state). The sensor initializes and the **Laser** tab appears in the Graphic Display window. The software automatically creates a COP command.



If the **Probe Toolbox** was already open, you can still change the sensor **Zoom** settings as needed.

4. Use the **Laser** tab and position the probe over the part in range as needed.
5. On the jog box, change the **Probe Enable** option to the "Enable" state. If not, it does not collect data.
6. On the jog box, press the **Record** button to start scanning. The **Laser** tab closes immediately, and the scanned data populates the COP object and the Graphic Display window in real time.
7. Use the jog box and move the probe over the part to scan it until you are satisfied with the data coverage.
8. To stop scanning, press the **Record** button again.
9. If necessary, press the **Probe Enable** button again to scan more data. You'll be prompted to empty the existing COP command or add new data to what is already there.
10. Repeat from Step 6 above to continue scanning.

You can also create a manual scan on a DCC machine by:

1. Follow steps 1-4 above.
2. On the jog box, change the **Probe Enable** button to the "Disabled" state.
3. On the jog box, press the **Record** button.
4. On the jog box, use the **Probe Enable** button to toggle data collection "On" and "Off".
5. On the jog box, press the **Record** button a second time to stop scanning and finalize the COP data.

Setting the Machine Speed for Scanning

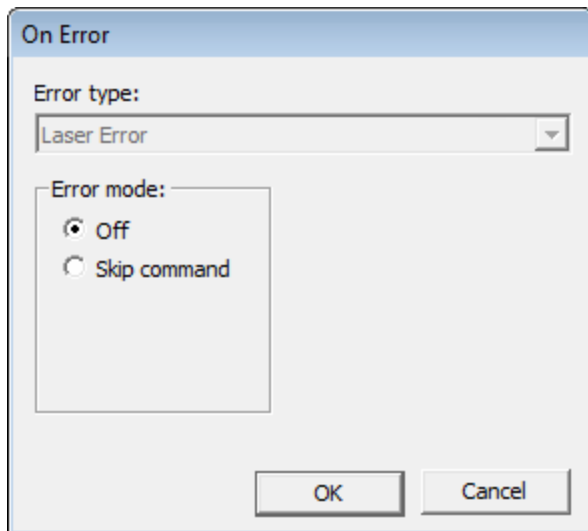
To properly define your machine's speed for scanning with your laser, you need to do the following:

- VHSS must be supported by your controller. PC-DMIS uses this high speed mode by default when supported by the CMM.

- The `ScanSpeed` registry entry, found in the **Leitz** section of the PC-DMIS Settings Editor, limits the maximum scan speed value that you can send to the controller. By default, this is set to 50 mm/sec. Any value set by a SCANSPEED/Edit window command is limited to the value of the `ScanSpeed` registry entry. This value can be increased accordingly the CMM limits.
- By default, PC-DMIS's **Acceleration** value, located in the **Opt. Probe** tab of the **Parameter Settings** dialog box, is set very low (10 mm/sec). To get higher scan speeds, you should increase this value to a desired value up to the limits allowed by your machine. To access this tab, select the **Edit | Preferences | Parameters** menu item and then click on the **Opt. Probe** tab.

Handling Laser Sensor Errors with On Error Command

The **Insert | Flow Control Command | On Error** option opens the **On Error** dialog box:



On Error dialog box

You can tell PC-DMIS to skip commands that generate certain laser sensor-related errors during execution by using the On Error command. The command only applies to the default Asynchronous Execution mode.

The information in this topic is specific to Laser configurations. For more information on this dialog box and how it applies to tactile probes, see "Branching on an Error" in the "Branching by Using Flow Control" chapter in the PC-DMIS Core documentation.

Error type - PC-DMIS Laser tracks these error conditions:

- Laser error
- Temperature out of threshold - The Temperature Compensation command in the measurement routine raises this error if one or more of the temperatures for the X axis, Y axis, or Z axis scale or part are above the upper threshold limit or below the lower threshold limit that the Temperature Compensation command defines. For more information, see "Branching on an Error" in the "Branching by Using Flow Control" chapter in the PC-DMIS Core documentation.



The On Error command must be placed above the Temperature Compensation command in the measurement routine.

Error mode - PC-DMIS can take these possible actions depending on the error type:

- **Off** - The command is not skipped. If PC-DMIS encounters an error in this mode, execution stops completely.
- **GoTo label** - The measurement routine flow moves to a defined label (see "Using Labels" in the "Branching by Using Flow Control" chapter in the PC-DMIS Core documentation). These options become available:
 - **Label ID** - In this box, type a reference to a label that doesn't exist yet.
 - **Current labels** - Lists all the labels in the measurement routine.
- **Set variable** - Sets a variable's value to one.
- **Skip command** - The execution continues, and PC-DMIS skips commands if they generate any of the following errors:
 - No laser stripes found for feature execution
 - No scan data
 - Feature calculation error

If PC-DMIS encounters any other laser errors, it stops execution and ignores the On Error command.

The command has the following syntax in the Edit window in Command mode:

`ONERROR/LASER_ERROR, TOG1`

TOG1 = This switches between SKIP or OFF.


Using the Mesh Commands

All Mesh commands are available from the **Mesh** toolbar (**View | Toolbars | Mesh** menu).


The Mesh commands are:

- **Mesh:** Click the **Mesh** button to display the **Mesh Command** dialog box to create a mesh feature from any number of pointclouds. You do not need to have any COPs defined to create a Mesh. If no COPs are defined, an empty Mesh object is created in the Edit window.

This option is available from the main menu (**Insert | Mesh | Feature**). It is also

accessed by clicking the **Mesh** button () on the **Pointcloud**, **QuickCloud**, or **Mesh** toolbar. When you select the option or button, the **Mesh Command** dialog box appears.

For details, see the "Creating a Mesh Feature" topic.


- **Mesh Operator:** This option is available from the main menu (**Insert | Mesh | Operator**) or from the **Mesh Operator** button () on the **Mesh** toolbar. This displays the **Mesh Operator** dialog box. Use the dialog box to create a Mesh operator.

For details, see the topic "Creating a Mesh Operator".

The operators are:

- Mesh CROSS SECTION Operator
 - Mesh EXPORT Operator
 - Mesh IMPORT Operator
 - Mesh COLORMAP Operator
 - Mesh EMPTY Operator
- **Import Mesh in STL Format:** This button opens the **Mesh Import Data** dialog box that you can use to import an STL mesh data file. If a Mesh object does not exist in the PC-DMIS Edit window, then a new Mesh object is created, and the software imports the STL data. If a Mesh object already exists in the PC-DMIS Edit window, then the software adds the STL data to the Mesh object.


For details, see the "Mesh IMPORT Operator" topic.

This option is available from the main menu (**File | Import | Mesh**). It is also accessed by clicking the **Import Mesh in STL format** button () from the **Mesh** toolbar.


For details, see the topic "Import Mesh in STL Format".

- **Export Mesh in STL Format:** This button opens the **Export Mesh Data** dialog box that you can use to export a Mesh in an STL ASCII or STL Bin file format.

For details, see the "Mesh EXPORT Operator" topic.

This option is available from the main menu (**File | Export | Mesh**). It is also accessed by clicking the **Export Mesh in STL format** button () from the **Mesh** toolbar.


For details, see the "Export Mesh in STL Format" topic.

- **Empty a Mesh:** Click the **Empty a Mesh** button () to empty a Mesh. To use this function, position the cursor in the Edit window directly ON the Mesh object you want to empty, then click the button. If your cursor is not on a Mesh, the Mesh immediately above your cursor position is emptied.

For details on the Empty a Mesh command, see the topic "Empty a Mesh".



Note that this is different from inserting the Empty command operator. In this case, the empty command is placed above the Mesh to empty. For details on the Empty command operator, see the "Mesh EMPTY Operator" topic.

- **Mesh Alignment:** Click the **Mesh Alignment** button () to display the **Mesh/CAD Alignment** dialog box. Use the dialog box to align the Mesh to a CAD model.

For details, see the "Mesh Alignment" topic.

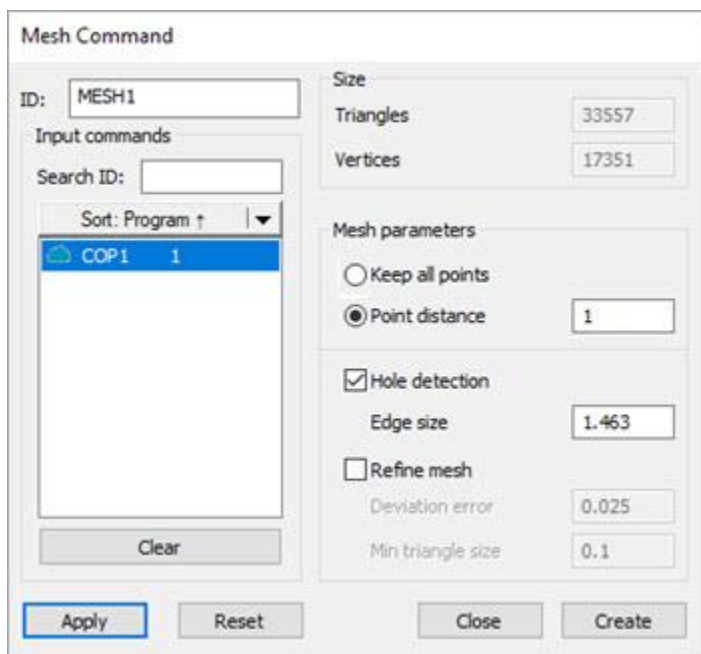
- **Receive a Mesh from OptoCat:** When clicked ON, PC-DMIS is placed in a state where it is waiting and ready to receive a mesh from the OptoCat application.

For details, see the "Receive a Mesh from OptoCat" topic.

Creating a Mesh Feature




The Mesh license must be enabled to use or view this option.



Mesh Command dialog box

The **Size** section details the number of triangle and vertices defined in your Mesh feature.

To create a Mesh feature:

1. Select **Insert | Mesh | Feature** from the main menu to display the **Mesh Command** dialog box. You can also access this option from the **Mesh** button  on the **Mesh** toolbar (**View | Toolbars | Mesh**).
2. Select the features and pointclouds to be meshed together from the list.
3. Update the options in the **Mesh parameters** section as needed:

- **Keep all points** - When you select this option, PC-DMIS uses all the points in the pointcloud to create the mesh.

When you select the **Keep all points** option, PC-DMIS requires more processing time to mesh the pointcloud.

- **Point distance** - This value defines the minimum distance between neighboring points that the software uses to create the vertices of each triangle in the mesh.

The **Point distance** option is the default and preferred setting. When you select this option, PC-DMIS projects a "grid" of this size onto the mesh and takes ONLY the best points in each grid element.

- **Hole detection** check box - When this check box is selected, PC-DMIS determines when to exclude points based on the **Edge Size** value.
 - **Edge size** - The value entered is used to determine when two points of the pointcloud are going to be included in the Mesh being created. If the distance is greater than the **Edge size** value, it's considered a hole, and the point is excluded. A value of -1 defines an edge size without limit.
 - **Refine mesh** check box - When this check box is selected, the following parameters refine the mesh being created:
 - **Deviation error** - The value entered determines how far points can deviate from the mesh construction and still be included in the mesh.
 - **Min triangle size** - The value entered determines the minimum size a triangle can be based on the points being evaluated.
4. Click **Apply** to apply any changes made in the **Mesh Command** dialog box. Click **Create** to generate the new Mesh command.

Click **Reset** to remove the created mesh from the Edit window and Graphic Display window.

Click **Close** to close the dialog box and cancel the mesh operation if the **Create** button wasn't clicked.


Creating a Mesh Operator

The Mesh operator commands listed below perform different operations on a Mesh object. The units for these commands are defined by the measurement routine.




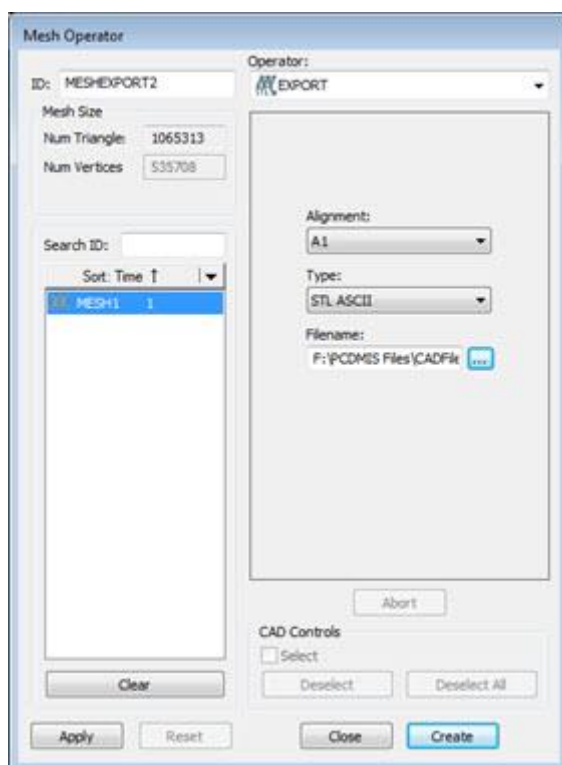
The Mesh license must be enabled to use or view this option.

To create a Mesh operator:

1. Click the **Mesh Operator** button () from the **Mesh** toolbar (**View | Toolbars | Mesh**) to access the **Mesh Operator** dialog box. The dialog box is also accessible from the menu (**Insert | Mesh | Operator**).



The **Mesh Operator** button is disabled if no Mesh object exists. You can create an empty Mesh object using the **Mesh** button ().



Mesh Operator dialog box

2. Select the type of operator to create from the **Operator** list.
3. Select the mesh from the **Feature List** box.

4. Select the options to use. The available options depend on the type of operator selected.
5. Click **Create**. The appropriate command inserts into the Edit window. For example, the EXPORT operator command is `MESH/OPER,EXPORT`.



An example of the command for a Mesh EXPORT operator is:

```
MESHEXPORT1=MESH/OPER,EXPORT,FORMAT=STL
ASCII,FILENAME=F:\TRAINING\TEST1_STL.STL,
REF,MESH1,,
```

Mesh CROSS SECTION Operator

The screenshot shows the 'Mesh Operator' dialog box with the 'Operator' set to 'Cross Section'. The 'ID' is 'MESHSECTION1'. The 'Mesh Size' section shows 'Num Triangles' and 'Num Vertices' both set to 0. The 'Search ID' field is empty. The 'Sort' dropdown is set to 'Program'. The 'Vector' section shows 'Start point' coordinates (X: 66.384, Y: 55.884, Z: -0.025) and 'Direction' coordinates (I: -0.9991207, J: -0.0419086, K: 0.0012373). The 'Width' is 142.812, 'Height' is 264, 'Delta' is 0.05, 'Step' is 10, 'Length' is 62.65649, 'Smoothing Tol' is 0, 'Gap Fill Distance' is 2, 'Point Spacing' is 1, and 'Max Distance to CAD' is 2. The 'Profile Dimension', 'Analysis View', and 'Annotation Min/Max' sections each have a '+' button. The 'CAD Controls' section has a 'Select' checkbox and 'Deselect' and 'Deselect All' buttons. The 'Create' button is highlighted in blue.


Mesh Operator dialog box - CROSS SECTION operator


The Mesh CROSS SECTION operation generates a subset of polylines determined by the defined intersection of a set of parallel planes with the Mesh object. The software defines the set of planes by the start point, direction vector, step distance between the planes, and length. The software determines the number of planes by the **Step** distance divided into the **Length** plus one.



You can evaluate the Mesh CROSS SECTION operator by the profile dimension.

To apply the CROSS SECTION operation to a Mesh:

1. From the **Mesh** toolbar (**View | Toolbars | Mesh**), click the **Cross Section a Mesh** button () to open the **Mesh Operator** dialog box. You can also click the **Insert | Mesh | Operator** menu option.
2. From the **Mesh Operator** dialog box, select **Cross Section** from the **Operator** list.

From the **Mesh** toolbar, click the **2D Section Slide Show** button  to display cross sections in 2D view. For details, see the "Cross Section Slide Show" section of the "Show and Hide Cross Section Polyines" topic.

The list underneath the **Operator** list contains these options: **Vector**, **Axis**, **Curve**, and **2 Points**. For details on how the **Curve** function works, see the "Creating a Cross Section along a Curve" topic. For details on the **2 Points** option, see the "Creating a Cross Section between 2 Points" topic.

The Mesh CROSS SECTION operator uses the following options:

- **Start point:** This option indicates the coordinates of a point belonging to the first plane cutting the mesh. The software displays the start point as a blue ball in the Graphic Display window. You can use the ball as a handle to drag to a new location. The software defines the start point by the first click in the Graphic Display window. In the actual Edit window command, the start point value is held in the START PT parameter.
- **Direction** (applies only to the **Vector** and **2 Points** options): This value indicates the direction of the normal vector. You can define it by the first click in the Graphic Display window. In the actual Edit window command, the software holds the **Direction** value in the NORMAL parameter.

- **Axis** (applies only to the **Axis** option): Use this option to create a cross section along the X, Y, or Z axis. Select the desired axis (the default is X), set a start point in the Graphic Display window, and set an end point. The section plane cuts the part at a given step value over the length of the cross section.
- **Width**: This value indicates the width of the section under consideration. If the value is 0, the system calculates the value as the CAD bounding box value.
- **Height**: This value indicates the height of the section under consideration. If the value is 0, the system calculates the value as the CAD bounding box value.
- **Delta**: The software does not use this value for Mesh cross sections.
- **Step**: This value indicates the distance between the planes. In the actual Edit window command, the software holds the step value in the INCREMENT parameter.



If the **Step** value is greater than the **Length** value, the software creates only one section cut at the start point.

- **Length**: This value indicates the maximum distance between the first and last plane. The software displays the length value in the **Length** parameter of the dialog box. PC-DMIS displays it as a purple line in the Graphic Display window.
- **Smoothing Tol**: Set to 0 (zero) to turn off smoothing (the default value).


Use **Smoothing Tol** to remove small steps in the cross section and create a smoother measured polyline. This setting filters out the points within the smoothing tolerance value and then fits a polyline to the data using the **Point Spacing** value.

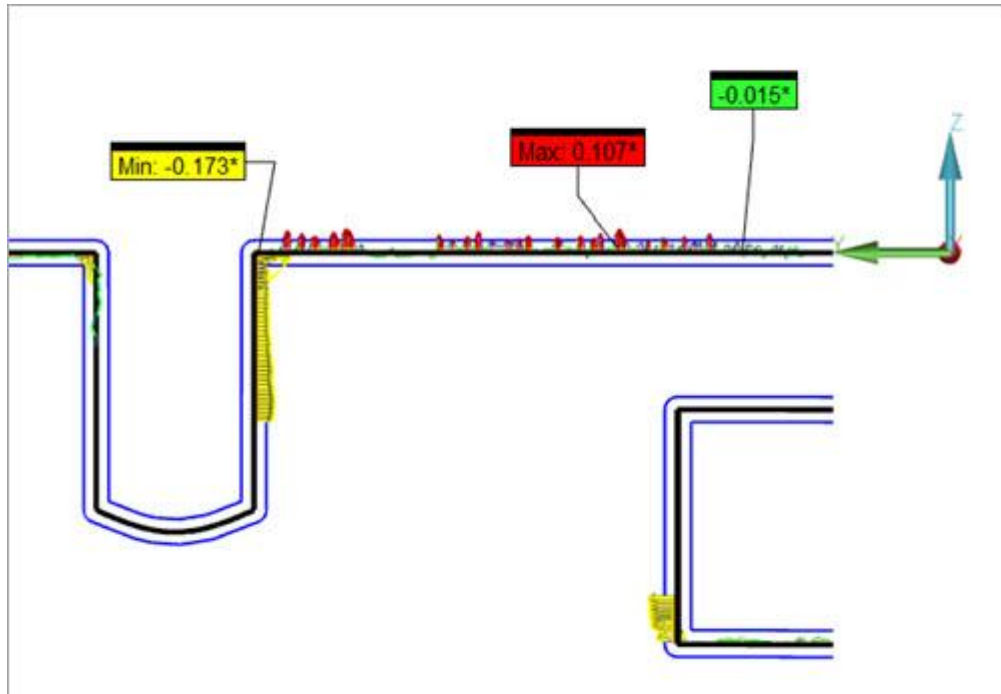


You can also define **Point Spacing** with the `CrossSectionCopCadCrossSectionStep` registry entry. For details on this registry entry, see "`CrossSectionCopCadCrossSectionStep`" in the PC-DMIS Settings Editor documentation.



You should set the **Smoothing Tol** very small so that the measured cross section does not deviate greatly from the actual data. Except for extreme situations (for example, a very large CAD model and/or a very poor density of points), you should set this parameter between a few tenths of a mm (maximum) and a few thousandths of a mm (minimum).

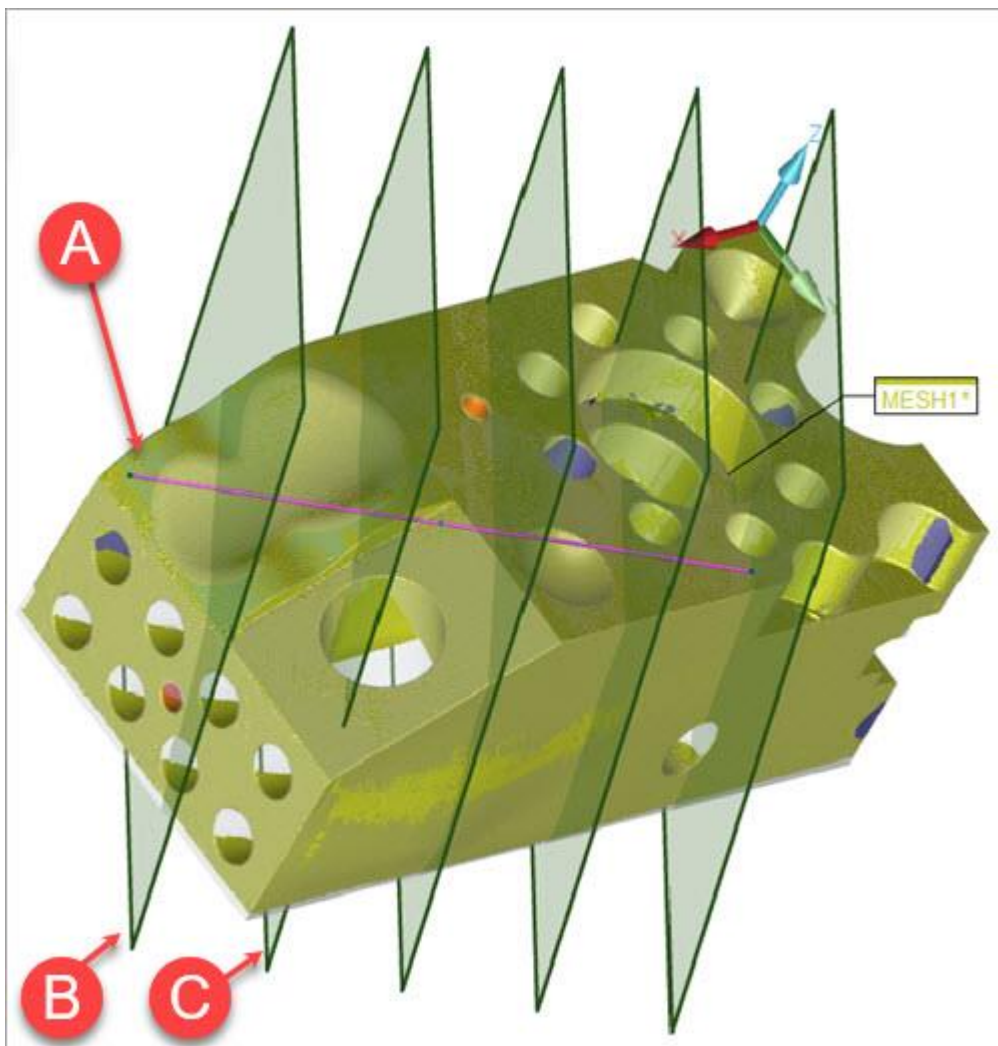
- **Gap Fill Distance:** This value defines the maximum gap distance along the yellow measured polylines of a cross section. If gaps equal to or smaller than this value appear, the gaps are filled in with calculated points. You can also set this value in the PC-DMIS Settings Editor. For details, see the `"CrossSectionMaximumEmptyLength"` topic in the PC-DMIS Settings Editor documentation.
- **Point Spacing:** Use this entry only when the `CrossSectionCopCadCrossSectionDrivenByCad` registry entry is set to 1 (True). This value is the step used along the CAD polylines to look for the best interpolated Mesh point. For greater accuracy, or if the CAD model is very small, you can set this value to a smaller value. You can also set this value in the PC-DMIS Settings Editor. For details, see the `"CrossSectionCopCadCrossSectionStep"` topic in the PC-DMIS Settings Editor documentation.
- **Max Distance to CAD:** The value defines the maximum distance of the Mesh data relative to the nominal CAD model. The default value is 2 mm. If the Mesh data object deviates more than the **Max Distance to CAD** value from the CAD model, the software may not compute the yellow measured cross section. You can adjust this value to account for large deviations of the Mesh data relative to the CAD model.
- **Profile Dimension:** Click the **Add** button  to create a new profile dimension for each cross section. For details on Profile Dimension, see "Dimensioning Profile - Line or Surface" in the "Using Legacy Dimensions" chapter of the PC-DMIS Core documentation.
- **Analysis View:** Click the **Add** button to create the `ANALYSISVIEW` command in the Edit window. For details on the `ANALYSISVIEW` command, see "Create Analysis View Command" in the "Inserting Report Commands" chapter in the PC-DMIS Core documentation.
- **Annotation Min/Max:** Click the **Add** button to create the minimum and maximum values in the form of annotation labels for the active cross section.



PC-DMIS calculates the minimum and maximum points each time you execute the measurement routine.

- **CAD Controls:** Mark the **Select** check box to select CAD surfaces in the Graphic Display window. PC-DMIS filters out any cross sections that do not pass through the selected surfaces when you click **Create**.

For example, if you select surface A after you define the start and end points, the software only creates the cross sections at B and C:



Example of a selected surface (A) limiting the cross sections to only (B) and (C)

Selected surfaces do not affect what you see when you click the **View** button.

When the cutting planes are visible in the Graphic Display window, it is possible to manipulate them as follows:

- Select a plane's edge handle and drag to resize the height and width of the cutting planes.
- Select a plane's corner handle and drag to rotate the set of planes around their axis.
- Select the first or last purple length line's blue point handle, and then drag to redefine the purple line's **START** or **END** definition. With the change in direction, the software updates the values in the dialog box and the number of planes in the Graphic Display window. In the case of Axis mode, the direction of the planes does not change.

- Select and drag the purple length line's middle blue point handle to move the set of planes.



When you create or edit a cross section, the cutting planes appear in a transparent view as shown above.

Click **Create** to:

- Insert a `MESH/OPER, CROSS SECTION` command for each plane into the Edit window.



For example:

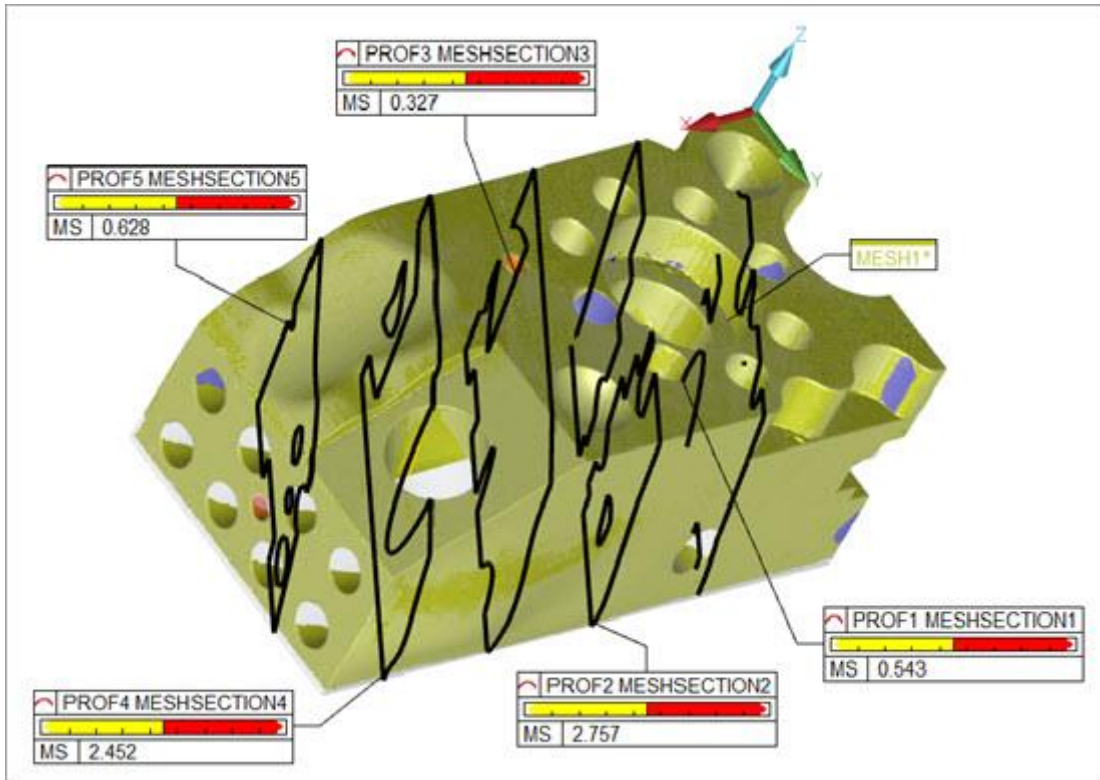
```
MESHSECTION3=MESH/OPER,Cross
Section,TOLERANCE=0.05,WIDTH=117.715,HEIGHT=227.086,
```

```
START PT = -6.439,60.097,6.276,NORMAL = 0.9684394,-
0.2221293,-0.1130655,SIZE=76
```

```
REF,MESH1,,
```

The black polylines represent the nominal CAD, the yellow polylines represent the measured polyline.

- Insert a label for each plane into the Graphic Display window as shown below:



Finished cross sections showing five planes

Defining the Cross Section by Typing Values

Use the **Mesh Operator** dialog box to enter any of these values:

- **START PT:** This value specifies the cross section's starting point using the **Start Point X, Y and Z** boxes.
- **NORMAL:** This value specifies the cross section's vector using the **Direction I, J and K** boxes.
- **WIDTH:** This specifies the value of the cross section's width property in the **Width** box.
- **HEIGHT:** This specifies the value of the cross section's height property in the **Height** box.
- **TOLERANCE:** This specifies the value PC-DMIS uses to determine the maximum distance from the plane a point must be in order for PC-DMIS to consider it as part of the cross section in the **Delta** box.
- **INCREMENT:** This specifies the value between cutting planes in the **Step** box.
- **LENGTH:** This specifies the value between the first and last cutting planes in the **Length** box.
- **SMOOTHING TOLERANCE:** This specifies the tolerance value to refine the points associated with the generated cross section in the **Smoothing Tol** box.

Defining the Cross Section by Using the Graphic Display Window

To define some of the cross section parameters, click the CAD model in the Graphic Display window to select the **Start Point**. A pink line appears. Click a second point on the CAD model to determine the **Direction** vector and the **Length**.

Creating a Profile Dimension from the Graphic Display Window

When you double-click a cross section label, PC-DMIS creates a new profile dimension that evaluates the selected cross section.

Exporting Mesh Cross Sections in IGES Format


Once you create the Mesh cross sections, you can export them in IGES format from the **Mesh Operator** dialog box.

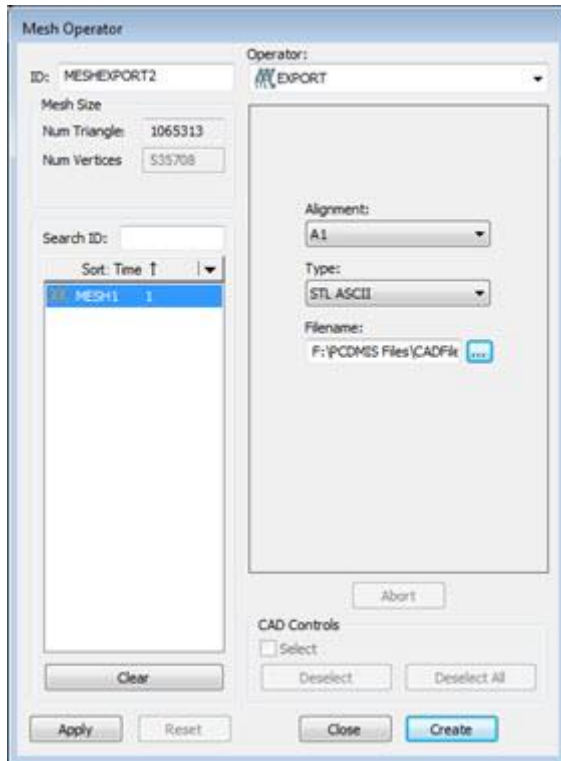
For details on the Mesh Export operator, see "Mesh EXPORT Operator" in this documentation.

Mesh EXPORT Operator

Exporting Mesh Data

To mesh data with the Mesh EXPORT operator:

1. Click the **Mesh Operator** button () from the **Mesh** toolbar (**View | Toolbars | Mesh**) to access the **Mesh Operator** dialog box.



Mesh Operator dialog box - EXPORT operator

2. Select the EXPORT operator from the **Operator** list.
3. Select the mesh from the **Feature List** box.
4. Select the options to use. The Mesh EXPORT operator uses these options:

Alignment: This option indicates the type of alignment to include when exporting the data.

Type: This list provides the options to determine the file type for the EXPORT operator. The file type options for the EXPORT operator are STL ASCII and STL Bin to export the Mesh data object.

If you have Mesh Cross Sections defined, the list contains a third option called IGES. You can also click the **Export Mesh Cross Section to IGES** button from the Mesh toolbar (**View | Toolbars | Mesh**) to perform this operation. For details on the Mesh Cross Section operator, see "Mesh CROSS SECTION Operator" in this documentation.

Filename: Indicates the name of the export file. Enter the path and file name, or use the **Browse** button to navigate to it.

5. Click **Create** and PC-DMIS inserts the EXPORT command into the Edit window. The command is `MESH/OPER, EXPORT`. The mesh data exports to the file location defined in the **Filename** box.



For example:

```
MESHEXPORT1=MESH/OPER,EXPORT,FORMAT=STL
ASCII,FILENAME=F:\PCDMIS FILES\STL\TEST1_STL.STL,
REF,MESH1,,
```

Exporting Mesh Cross Section Data in IGES Format


To export Mesh cross section data in IGES format:

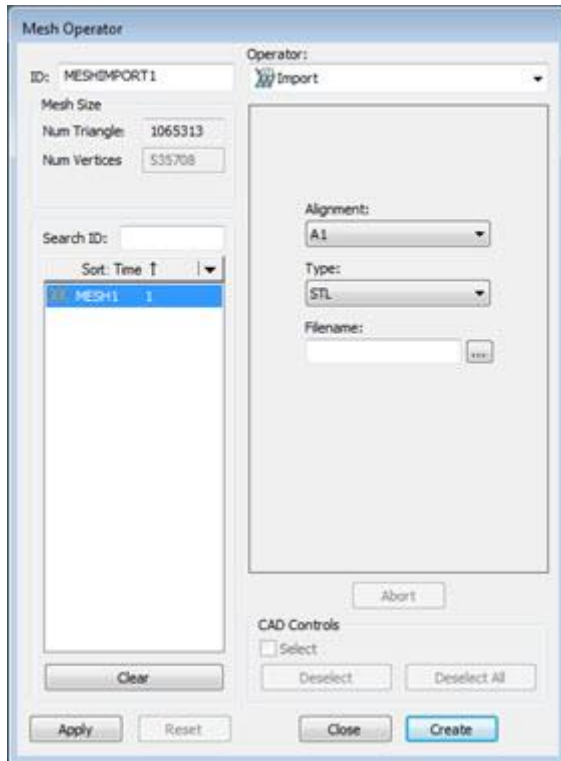
1. Click the **Mesh Operator** button from the **Mesh** toolbar (**View | Toolbars | Mesh**), or select it from the menu (**Insert | Mesh | Operator**) to open the **Mesh Operator** dialog box.
2. Select the **Mesh Export** operator from the **Operator** list.
3. From the **Type** list, select the **IGES** option. The software display all Mesh cross sections in the **Feature List** box.
4. Select the cross sections in the **Feature List** box you want to export.
5. Click **Apply** and then click **Create** to complete the export of Mesh Cross Sections in IGES format.
6. Click **Close** to return to the main PC-DMIS screen.

Mesh IMPORT Operator

To create a Mesh IMPORT operator:



1. Click the **Mesh Operator** button () from the **Mesh** toolbar (**View | Toolbars | Mesh**) to access the **Mesh Operator** dialog box.



Mesh Operator dialog box - IMPORT operator

2. Select the IMPORT operator from the **Operator** list.
3. Select the mesh from the **Feature List** box.
4. Select the options to use. The Mesh IMPORT operator uses these options:

Alignment: Indicates the type of alignment to include when exporting the data.

Type: The option for the IMPORT operator is **STL**.

Filename: Indicates the name of the import file. Enter the path and file name, or use the **Browse** button to navigate to it.

5. Click **Create**. The IMPORT command is inserted into the Edit window. The command is `MESH/OPER, IMPORT`. The mesh data imports from the file location defined in the **Filename** box.



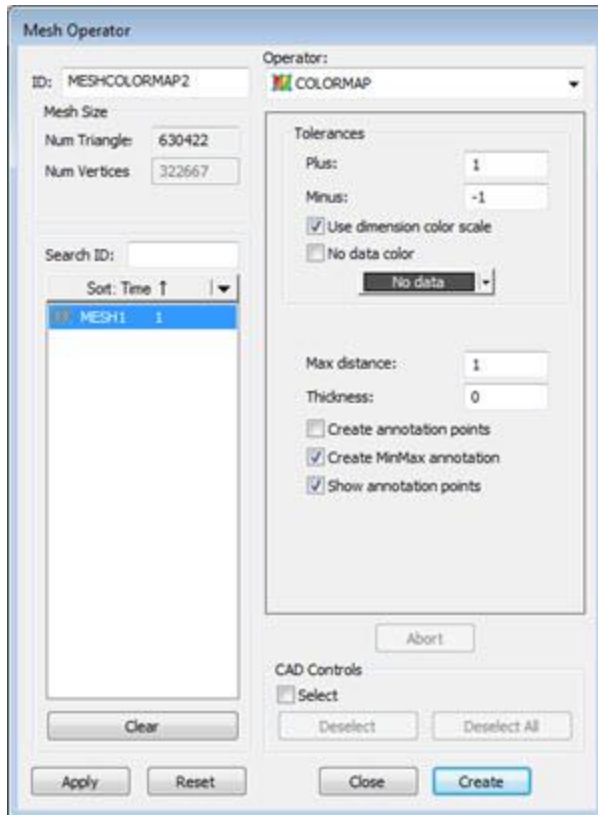
For example:

```
MESHIMPORT1=MESH/OPER, IMPORT, FORMAT=STL, FILENAME=F:\PCDM
IS FILES\STL\Test2_STL.STL,
```

REF, MESH1, ,

Mesh COLORMAP Operator


Mesh COLORMAP Operator



Mesh Operator dialog box - COLORMAP operator

The Mesh COLORMAP operation applies a colored shading to the selected mesh. The software shades the colormap according to the deviations of the mesh compared to CAD. PC-DMIS uses the colors defined in the **Edit Dimension Color** dialog box and the tolerance limits specified in the **Upper tolerance** and **Lower tolerance** boxes discussed below.


Because the Mesh colormap shows the color deviations on the Mesh object, when you apply the colormap, the software hides the CAD model. In comparison, the Pointcloud colormap colors the deviations on the CAD model, and PC-DMIS does not hide the

CAD model. To show or hide the CAD model, click the **Show CAD** button () on the **Graphic Items** toolbar. For details, see "Graphic Items Toolbar" in the "Using Toolbars" chapter in the PC-DMIS Core documentation.

The colors used for the colormap are defined in **Edit Dimension Color** dialog box (**Edit | Graphic Display Window | Dimension Color**).

Select **View | Other Windows | Dimensions Colors** to view the color scale from the **Dimensions Color Bar**.

To apply the Mesh COLORMAP operation to a Mesh:

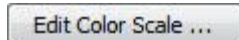
1. Click the **Color a Mesh** button () on the **Mesh** toolbar (**View | Toolbars | Mesh**), or select **Insert | Mesh | Colormap**.
2. Update these options based on your needs:

Tolerances - Use this option to set the upper (plus) and lower (minus) tolerance values:

Plus - This is the upper tolerance value.

Minus - This is the lower tolerance value.

Use dimension color scale check box - Select this check box to use the **Dimensions Color Bar** to define the color bar used for the Mesh Colormap color properties. For details, see "Using the Dimensions Colors Window (Dimensions Color Bar)" in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.



Edit Color Scale - If you do not select the **Use dimension color scale** check box, the software enables the **Edit Color Scale** button. When you click this button, the functionality to dynamically change the color, scale and threshold of the surface and point colormap properties becomes available through the **Color Scale Editor** dialog box. For details, see the "Edit the Color Scale" topic.

No data color check box - When you select this check box, the software maps the selected color to areas on the selected surfaces where no data is found.

Max distance - The software only includes points that fall within the **Max distance** value as part of the colormap. Note that if this value is too small, you may not see all the expected colored deviations. A good rule of thumb is to set this value slightly larger (10%, for example) than the largest deviation.

Thickness - This adds a thickness value to deviations on the colormap. This is useful if you want to add a material thickness to the Mesh surface model.

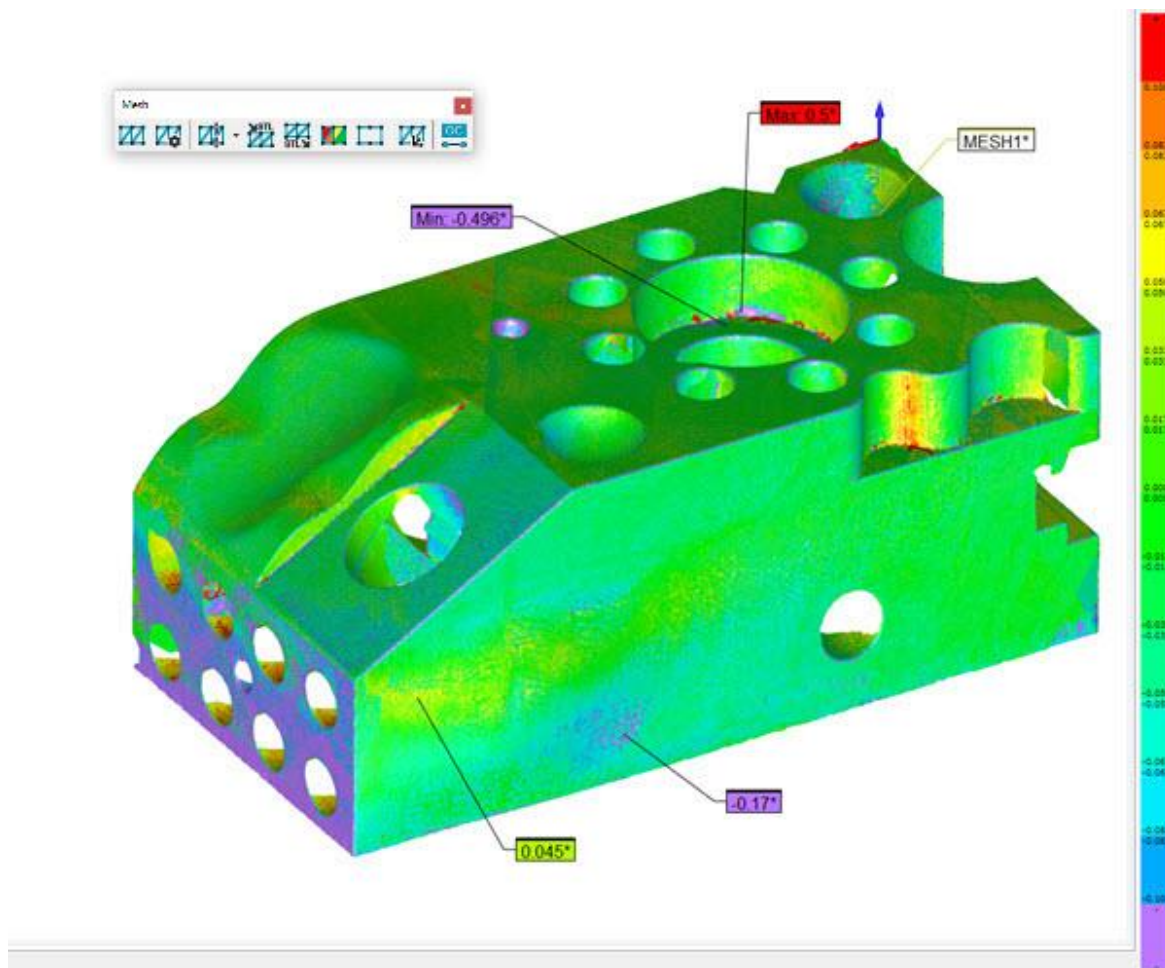
Create annotation points check box - Annotations are a way to display the deviation for a specific location on a surface colormap with its associated color. To create an annotation:

1. Select the **Create annotation points** check box. This clears the **Select** check box in the **CAD Controls** area and disables most of the options on the right side of the dialog box.
2. Select a point on the colormapped Mesh in the Graphic Display window. PC-DMIS evaluates and creates an annotation label in the same background color as the Mesh deviation point with the deviation value. The label can be moved around in the Graphic Display window as any other label.



Once created, the annotation labels remain in the same position and have the same characteristics if you restart the measurement routine, or if you restart PC-DMIS and reload the same measurement routine.

Create MinMax annotations check box - When you select this check box, PC-DMIS creates the minimum and maximum values in the form of annotation labels for the active Mesh Surface Colormap.



Example of Mesh Colormap with Min, Max, and various point annotation labels displayed

The software calculates the minimum and maximum points each time you execute the measurement routine.

Show, Hide, or Delete Annotation Labels

To show, hide, or delete annotation labels, right-click on a label to display the shortcut menu, and then select the appropriate option.

Delete Annotation - The software deletes the selected annotation label.

Show All Annotations - The software displays all annotation labels.

Hide All Annotations - The software hides all annotation labels.

Delete All Annotations - The software deletes all annotation labels.

Show annotation points check box - When you select this check box, the software displays all annotation points.

3. Click **Create** to insert a `MESH/OPER, COLORMAP` command into the Edit window.



For example:

```
MESHCOLORMAP1=MESH/OPER, COLORMAP, PLUS  
TOLERANCE=0.5, MINUS TOLERANCE=-0.5, THICKNESS=0, MAX  
DISTANCE=1,
```

```
REFINE  
FACTOR=0.1, TRIANGLES=401063, VERTICES=206625,  
REF, MESH1, ,
```

Colormaps in the Report

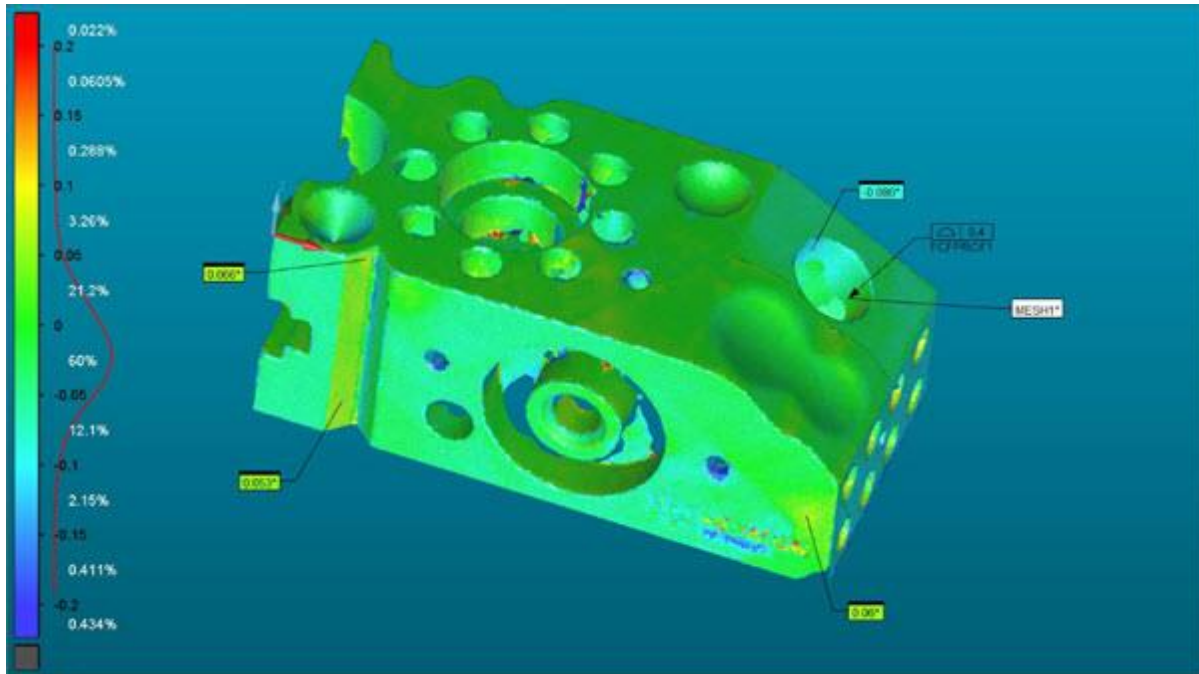
For information on how the software shows colormaps in the report, see "Colormaps and the CadReportObject" in the "Reporting Measurement Results" chapter of the PC-DMIS Core documentation.

More:

Dimensioning Surface Profile Using the Mesh COLORMAP

Dimensioning Surface Profile Using the Mesh COLORMAP

You can use a Mesh COLORMAP to create a Dimension Surface Profile.



Example of a Dimension Surface Profile created using a Mesh COLORMAP

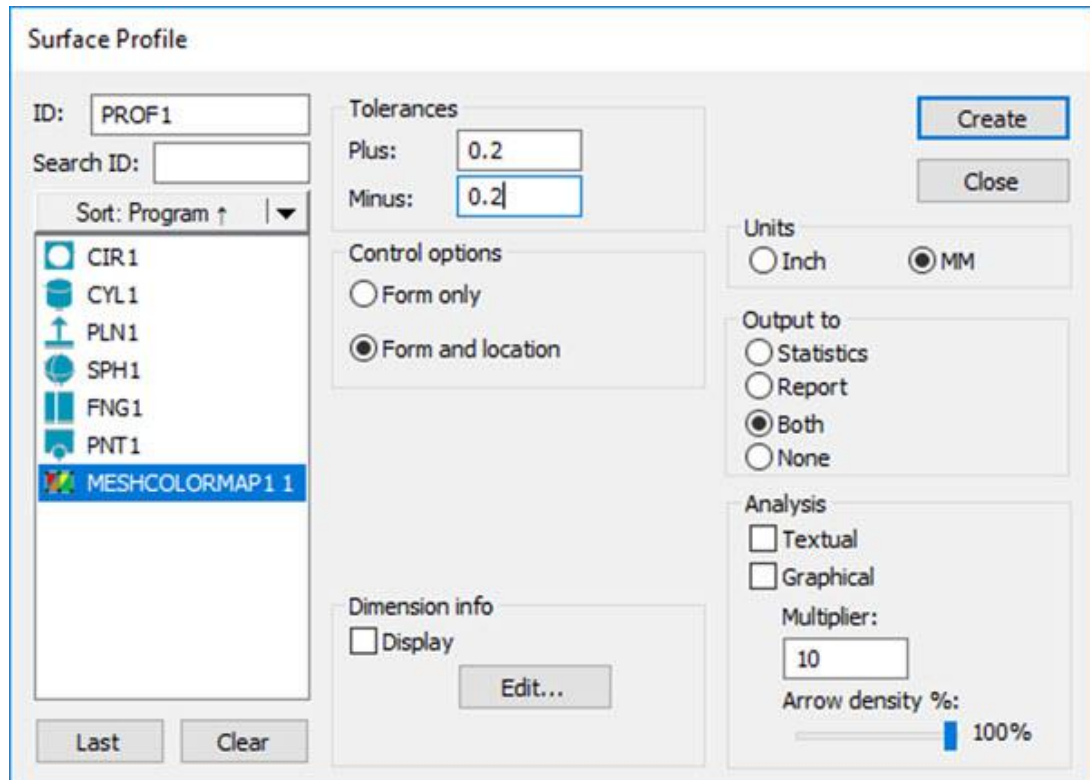
To create a Dimension Surface Profile from a Mesh COLORMAP, do the following:

1. Create a Mesh COLORMAP. For details, see "Mesh COLORMAP Operator".
2. Use one of these dimensioning methods to create the Dimension Surface Profile:

Legacy Dimension

To create the Dimension Surface Profile for legacy dimensions:

- a. Make sure you have the **Use Legacy Dimensions** option selected (**Insert | Dimension | Use Legacy Dimensions**).
- b. Click the **Profile Surface Dimension** option from the **Dimension** toolbar (**View | Toolbars | Dimension**), or select it from the menu (**Insert | Dimension | Profile | Surface**). The **Surface Profile** dialog box opens.



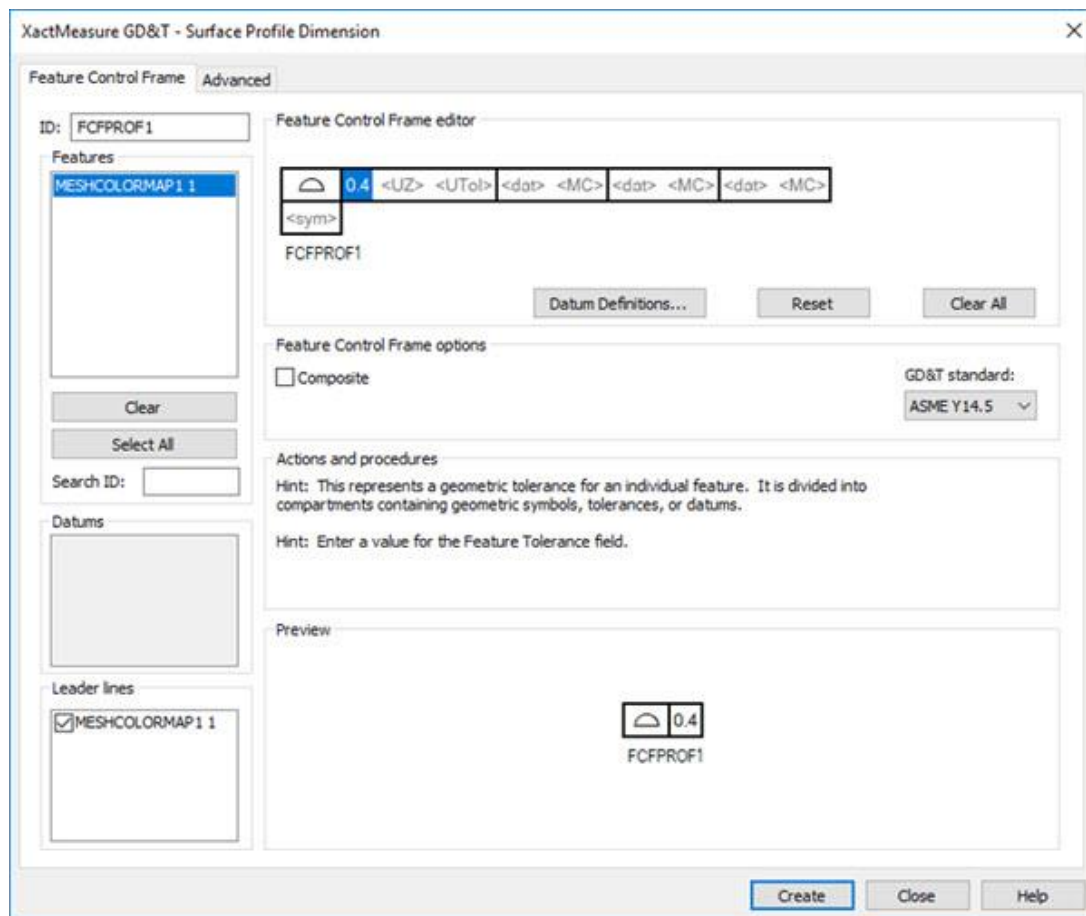
Surface Profile legacy dialog box for Mesh COLORMAP

For details on creating a legacy Surface Profile, see "To Dimension a Feature Using the Surface Profile Option" in the "Using Legacy Dimensions" chapter of the PC-DMIS Core documentation.

XactMeasure Dimension

To create the Dimension Surface Profile for XactMeasure dimensions:

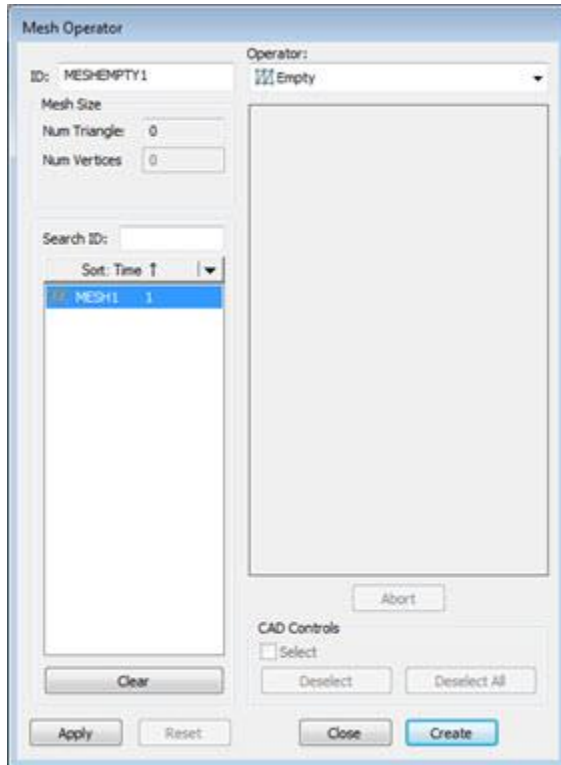
- Make sure the **Use Legacy Dimensions** option is NOT selected (**Insert | Dimension | Use Legacy Dimensions**).
- Click the **Profile Surface Dimension** option from the **Dimension** toolbar (**View | Toolbars | Dimension**), or select it from the menu (**Insert | Dimension | Profile | Surface**). The **XactMeasure GD&T - Surface Profile Dimension** dialog box opens.



XactMeasure GD&T - Surface Profile Dimension dialog box for Mesh COLORMAP

3. Select the desired Mesh COLORMAP from the **Features** list box.
4. Set the other options as needed.


Mesh EMPTY Operator



Mesh Operator dialog box - EMPTY operator

When this command is executed, PC-DMIS removes all the data from the Mesh.

To apply the Mesh EMPTY operation to a Mesh:

1. In the Edit window, position your cursor just above the Mesh you want to empty.
2. Click **Empty a Mesh** () on the **Mesh** toolbar, or select the menu option **Operation | Mesh | Empty**. The **Mesh Operator** dialog box appears.
3. Click **Create** to insert a `MESH/OPER, EMPTY,` command into the Edit window. The software inserts it just above the Mesh you want to empty. This is the Mesh that the Empty command acts on.



For example:

```
MESHEMPTY1 =MESH/OPER, EMPTY,  
REF, MESH1, ,
```



Once you apply this command to a Mesh, there is no way to restore the Mesh data. You cannot click **Undo** to restore the lost data.


Import Mesh in STL Format

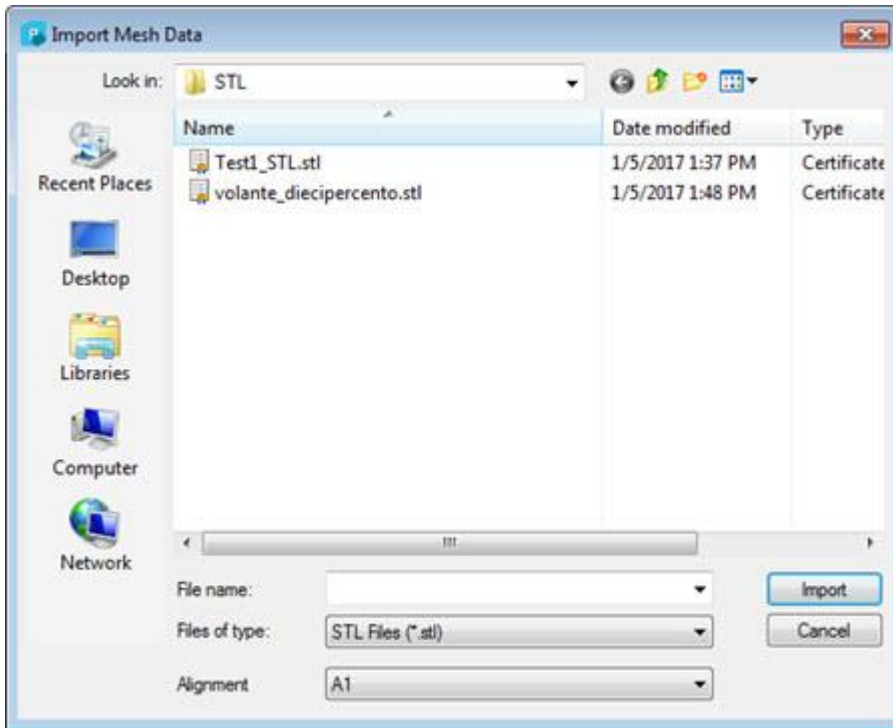


If a Mesh object does not exist in the PC-DMIS Edit window, then a new Mesh object is created and the STL data is imported. If a Mesh object already exists in the PC-DMIS Edit window, then the STL data is added to the Mesh object. If the data needs to be separated, you must create an empty Mesh object and then import the mesh STL data into that one.

The Mesh license must be enabled to use or view this option.

To import mesh data from an STL file:

1. Click the **Import Mesh in STL format** button () found on the **Mesh** toolbar (**View | Toolbars | Mesh**) to open the **Import Mesh Data** dialog box. You can also import a Mesh STL file from the menu (**File | Import | Mesh**).



Import Mesh Data dialog box


2. Use the dialog box to navigate to the location of the file containing the mesh data. Select the file type from the **Files of type** list to filter the list of files displayed in the dialog box. Left-click the file you want to import the mesh data from.
3. Select the alignment type from the **Alignment** list.
4. Click the **Import** button to import the mesh data. Click **Cancel** to exit the dialog box without importing any data.

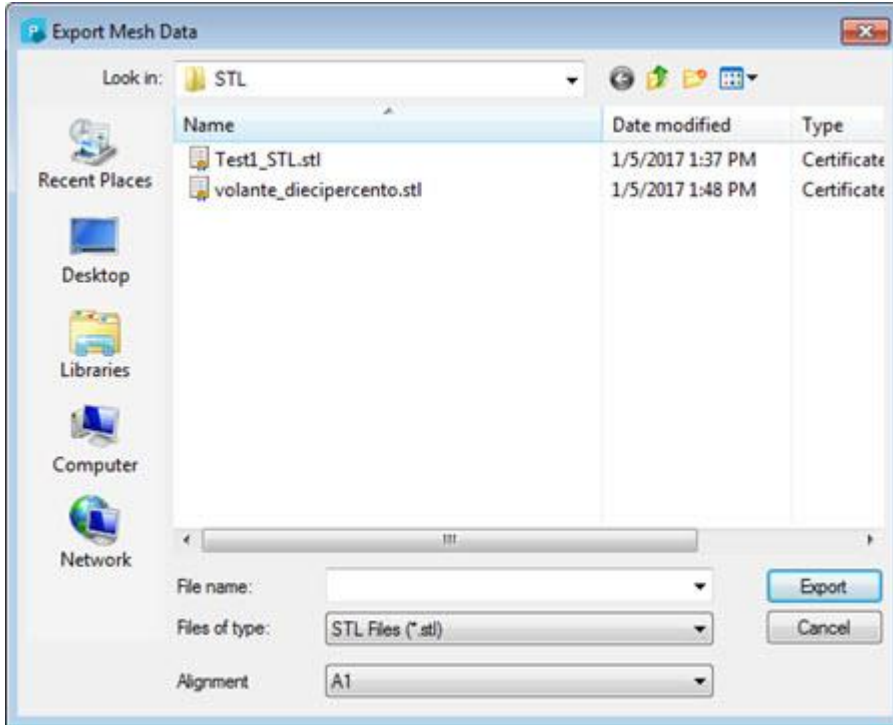
Export Mesh in STL Format



The Mesh license must be enabled to use or view this option.

To export mesh data to an STL file:

1. Click the **Export Mesh in STL format** button () on the **Mesh** toolbar (**View | Toolbars | Mesh**) to open the **Export Mesh Data** dialog box. You can also export a Mesh in STL format from the menu (**File | Export | Mesh**).



Export Mesh Data dialog box

2. Use the dialog box to navigate to the location where you want to export the mesh data.
3. Enter a unique name for the file in the **File name** box.
4. From the **Alignment** list, select the alignment you want to apply to the mesh data.
5. Click the **Export** button to export the mesh data. Click **Cancel** to exit the dialog box without exporting the data.


Empty a Mesh



The Mesh license must be enabled to use or view this option.

To empty a mesh:

1. In the Edit window, position your cursor on or just below the Mesh you want to empty. If you have two consecutive meshes defined in the Edit window, you must be on the mesh you want to empty.

2. Click the **Empty a Mesh** button  from the **Mesh** toolbar, or select **Operation | Mesh | Empty** from the menu.

The Mesh is emptied of all its data.



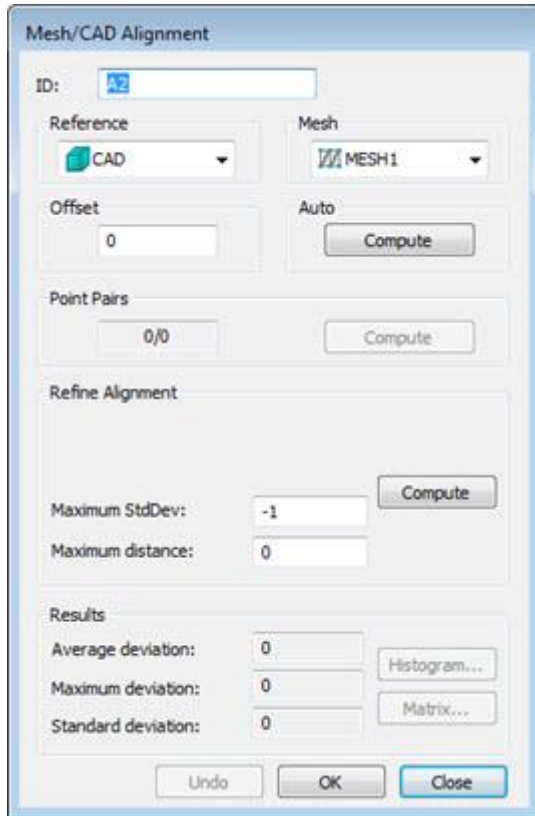
Once you apply this command to a Mesh, there is no way to restore the Mesh data. You cannot click **Undo** to restore the lost data.

Mesh Alignment

In order to use the data you've collected in your mesh properly, you need to create an alignment between the mesh and the CAD data of your part model or between meshes. This is done using the **Mesh/CAD Alignment** dialog box.

You can access this dialog box from the **Mesh Alignment** button () on the **Mesh** toolbar (**View | Toolbars | Mesh**).

Mesh/CAD Alignment Dialog Box Description



Default view of Mesh/CAD Alignment dialog box

The **Mesh/CAD Alignment** dialog box contains these options:

ID - This option displays the identification label for the alignment.

Reference - Select the reference object for your alignment, usually either the CAD itself or a defined Mesh. The mesh is aligned to the selected Reference.

Mesh - This list lets you choose the mesh to use in the alignment.

Offset - This option defines an offset value for a surface CAD model and is typically used with sheet metal parts. Applying an offset value essentially gives the surface CAD model a thickness so that you can align the mesh data to a different face that isn't represented in the surface CAD model. For example, if you have a surface CAD model for the top of a part but you want to align to a corresponding bottom surface, you could apply an offset value of the part's thickness to align the scanned data to the bottom side. Use a positive value if you want to apply a thickness in the same direction as the surface normal vector; use a negative value if you want to apply a thickness opposite the surface normal. It is only available for Mesh to CAD alignments.

Auto - This area lets you automatically align the CAD with the mesh with the **Compute** button. It is only available for Mesh to CAD alignments.

Point Pairs - This area lets you create a rough alignment based on selected points from the CAD that correspond to selected points from the mesh. Once you have the needed pairs selected, click **Compute** to perform the rough alignment.

Refine Alignment - This area allows for a more refined alignment. Only the **Maximum Distance** option is available for Mesh to Mesh alignments.

Depending on the alignment being made, the **Refine Alignment** area of the dialog box may consist of the following items:

Total points - This box defines the number of random sampled points used to refine the alignment. This number must be a value of at least 3. A good number is around 200 points.

Maximum iterations - This box defines the number of repetitions the process makes in order to refine the alignment.

Compute - This button begins the refined alignment process. A progress bar on the status bar shows the progress as the process moves through the alignment iterations.

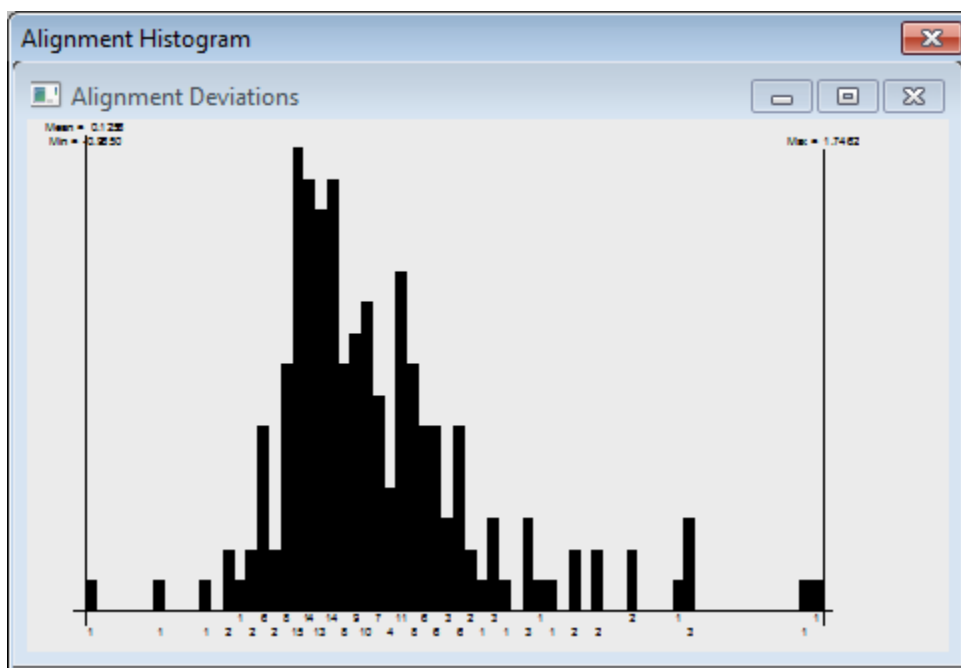
Maximum StdDev - Maximum StdDev is the maximum standard deviation used during the execution of an auto alignment. If the entered value is exceeded during the command execution, you are prompted to optionally pick point pairs on the CAD/Pointcloud. A value of -1 disables the Maximum StdDev functionality.

Maximum Distance - Defines the maximum distance PC-DMIS looks from the CAD for valid Mesh points. If no value is entered, the default value of 0 (zero) is used and the maximum distance becomes half the distance of the CAD bounding box.

Results - This area contains the following items:

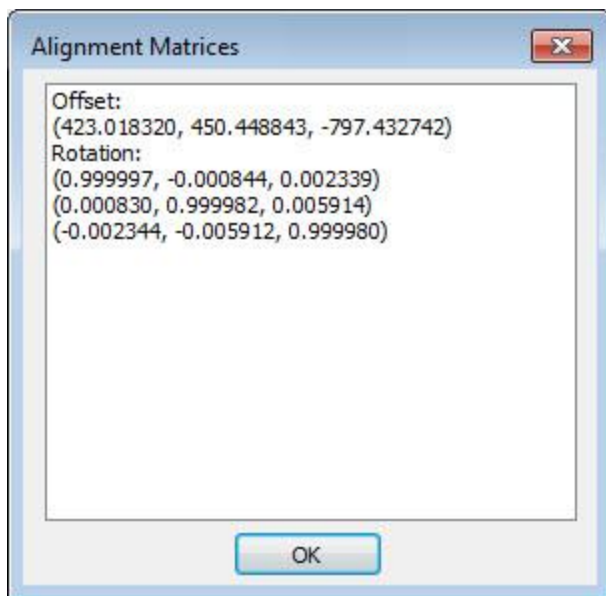
Information boxes showing the **Average Deviation**, **Maximum Deviation**, and **Standard Deviations** of the mesh data compared to the CAD model's data.

Histogram - This button takes a random sample of points from the mesh and projects them onto the CAD. The **Alignment Histogram** dialog box shows the deviations for that sample.



Sample Alignment Histogram dialog box for the selected Mesh

Matrix - This button displays the **Alignment Matrices** dialog box for the mesh alignment. The numerical values of the mesh alignment in offset and rotation matrices are listed.



Sample Alignment Matrices dialog box for the alignment

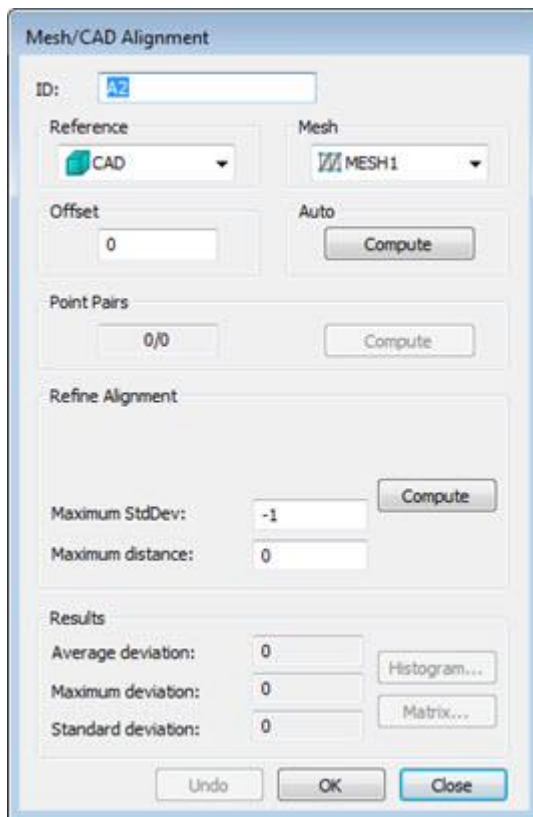
Creating a Mesh/CAD Alignment

To create a Mesh to CAD alignment, do the following:

1. Ensure that you have an imported CAD model in the Graphic Display window and a [MESH](#) command in the measurement routine. These elements are required to align a mesh to the CAD.
2. Select the **Insert | Mesh | Alignment** menu option or select the **Mesh**



Alignment button () on the **Mesh** toolbar. You can also access this dialog box by typing the [MESHCADBF](#) command in the Edit window's Command mode between the [ALIGNMENT/START](#) and the [ALIGNMENT/END](#) commands. The **Mesh/CAD alignment** dialog box appears:



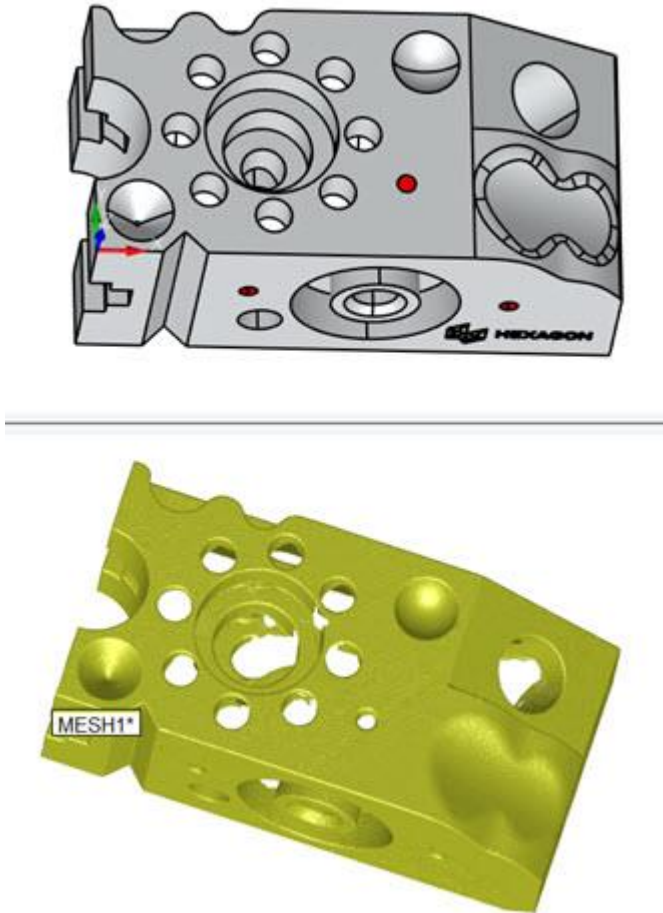
Mesh/CAD Alignment dialog box



For a complete description of the **Mesh/CAD Alignment** dialog box, see the topic "Mesh/CAD Alignment Dialog Box Description" in the PC-DMIS Laser documentation.

3. A temporary and split-screen view of the CAD model and the mesh appears in the Graphic Display window. You can use this split-screen view to visually see the alignment taking place. Select your point of reference from the **Reference**


drop down list; usually, either the CAD model itself or a defined Mesh is available. The Mesh is aligned to the selected reference.

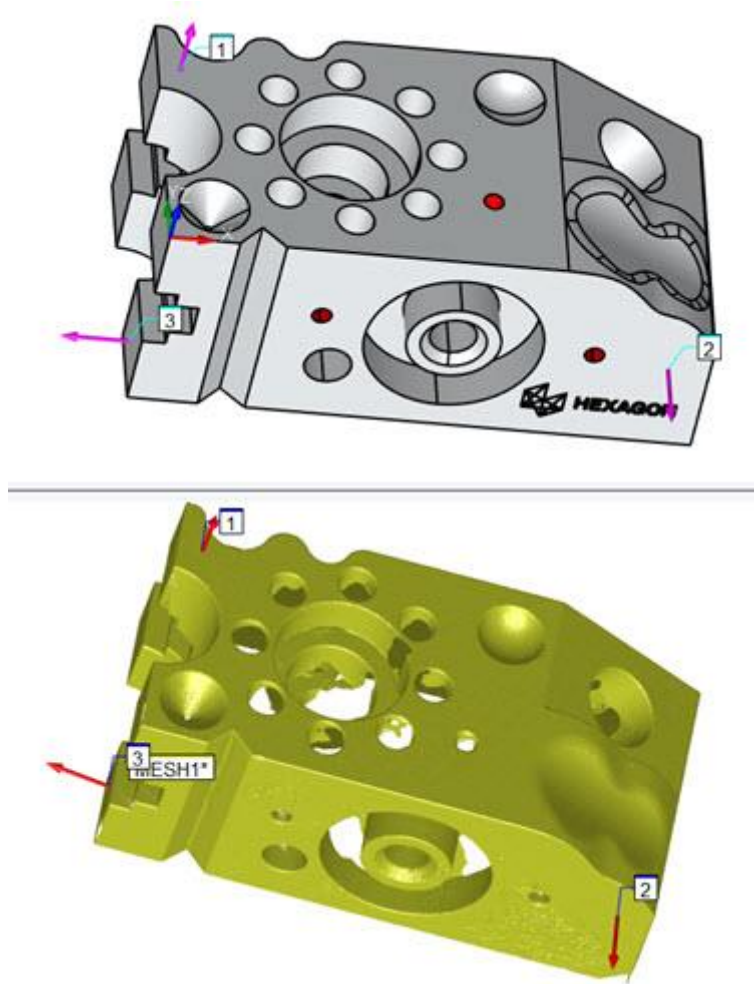


Split-screen view showing the CAD model on the top view, and the mesh on the bottom view


4. If you have more than one mesh in your measurement routine, choose the mesh from the **Mesh** list.
5. Perform the alignment:
 - a. Click the **Compute** button in the **Auto** section. You should only use this when you have a full scan of the external faces of the part. This automatically performs an alignment of the Mesh to the CAD and also a refinement on the alignment as it's generating.
 - b. If the auto compute does not compute a good alignment, use the **Point Pairs** area to perform a rough alignment. This brings the mesh close enough to the CAD if it's not already close. You can then refine the alignment further if needed. You should use this type of alignment if the

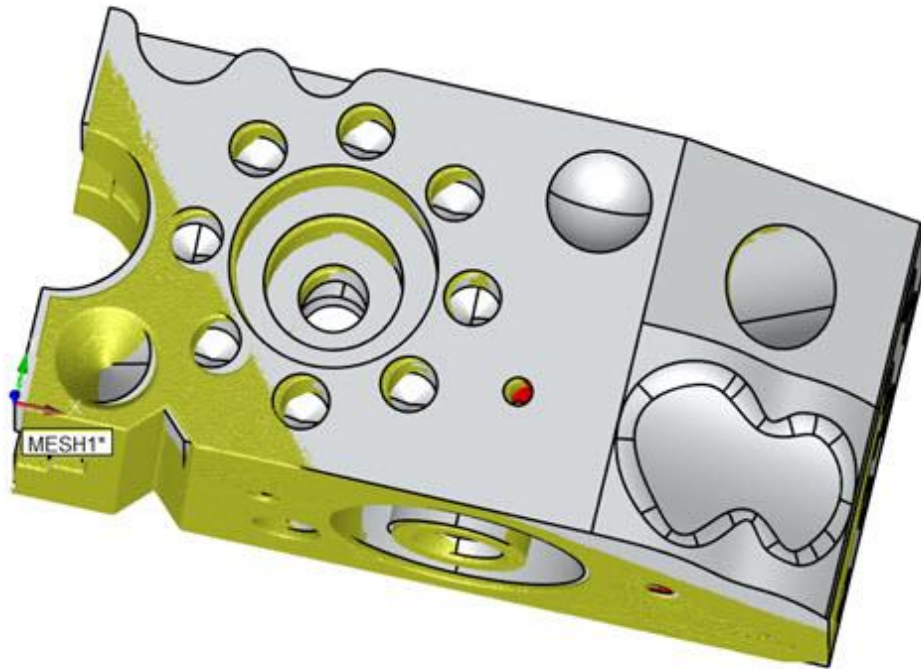
mesh is not complete or it contains scanned data belonging to a fixture, table, or other similar feature.

- i. Click a desired number of points on the mesh.
- ii. Click corresponding locations on the CAD model. 



Split-screen view showing selected CAD (top) and corresponding mesh (bottom) points

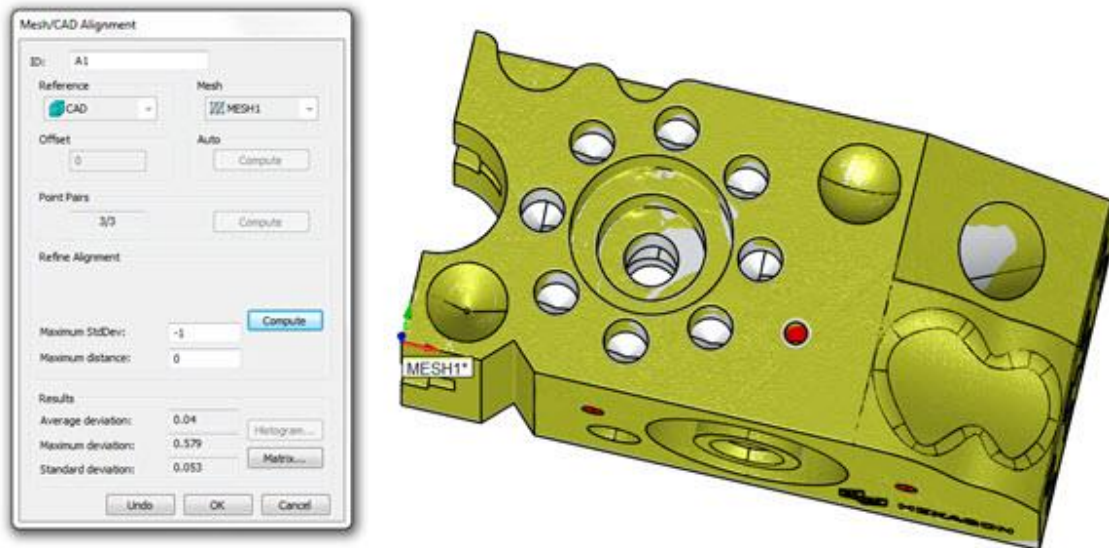
- iii. The more points you take around the different areas of the model and mesh, the better the rough alignment.
 - iv. Click **Compute** to create the rough alignment.
- c. Next, use the **Refine Alignment** area whenever you want to refine your alignment, thereby bringing the mesh closer to your CAD model. In order to be able to obtain a good refined alignment, the mesh points should be close enough to the CAD points through an initial rough alignment. 



A sample rough mesh-to-CAD alignment that needs a refinement

- i. Define the total number of random sample points to use in each iteration in the **Total Points**.
 - ii. Define the number of iterations in the **Maximum Iterations** box.
 - iii. Define the maximum standard deviation for the auto alignment execution between the points in the mesh and the CAD model using the **Maximum StdDev** box. When the auto alignment command is executed, if the standard deviation of the Mesh/CAD deviations is greater than the maximum value defined, you can select point pairs to get a better alignment. The default value of -1 is equivalent to an infinite allowed standard deviation.
 - iv. Define the maximum distance of the points from the CAD in order for use in the best fit routines. The default value is 0. In this case, an internal max distance based on the size of the mesh is used.
 - v. Click **Compute** to refine the alignment.
6. If a portion of the mesh doesn't align nicely with the CAD, you can click the **Undo** button and recompute the alignment using the same type of alignment with additional parameters, or you can try a different alignment.
 7. If you have a surface model representing a sheet metal part, and you want to align to the offset faces, define an **Offset** value representing the constant thickness of the sheet metal part.

8. Use the **Results** area to see how well the mesh aligned with the CAD. Make any changes to the **Offset** or **Refine Alignment** values to improve the alignment if necessary. If any changes are made, be sure to click the **Compute** button to regenerate the alignment with the new values.
9. Once you're satisfied with the alignment, click **Create**. PC-DMIS closes the temporary split-screen view and places the **MESHCADBF** command in the Edit window. See the "MESHCADBF Command Mode Text" topic.



Example of a completed Mesh to CAD alignment

MESHCADBF Command Mode Text

The MESHCADBF command allows you to perform a best fit alignment of the mesh data with the CAD data.

Below is an example code snippet for a MESHCADBF alignment:

```
A1 =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
    MESHCADBF/REFINE=n1,n2,n3,SHOWALLPARAMS=TOG1,
    ROUGH ALIGNPAIR/
        THEO/<x,y,z>,<i,j,k>,
        MEAS/<x1,y1,z1>
    REF,TOG2,,
ALIGNMENT/END
```


n1 represents the offset value for applying a thickness.

n2 represents the maximum standard deviation value.

n3 represents the maximum distance value.

TOG1 lets you show or hide the parameters used for the rough alignment. It can be set to YES or NO.

```
ROUGH ALIGNPAIR/  
    THEO/x,y,z,i,j,k,  
    MEAS/x1,y1,z1
```

These rough alignment pairs of points are defined/selected using the Graphic Display window. The values next to **THEO/** represent the point on the CAD. The values next to **MEAS/** represent the corresponding point on the Mesh. These pairs are used to determine a rough transformation between the CAD and the Mesh which allows the Mesh to come close enough to the CAD to allow further refinements of the alignment.

TOG2 lets you choose the mesh to use for the alignment.

Creating a Mesh to Mesh Alignment

The Mesh to Mesh alignment functionality allows you to best fit align one mesh to another mesh which have been collected in two different reference frames that have some overlap. A typical example is two scans in two mesh commands, representing areas of a part that cannot be scanned in the same part orientation.

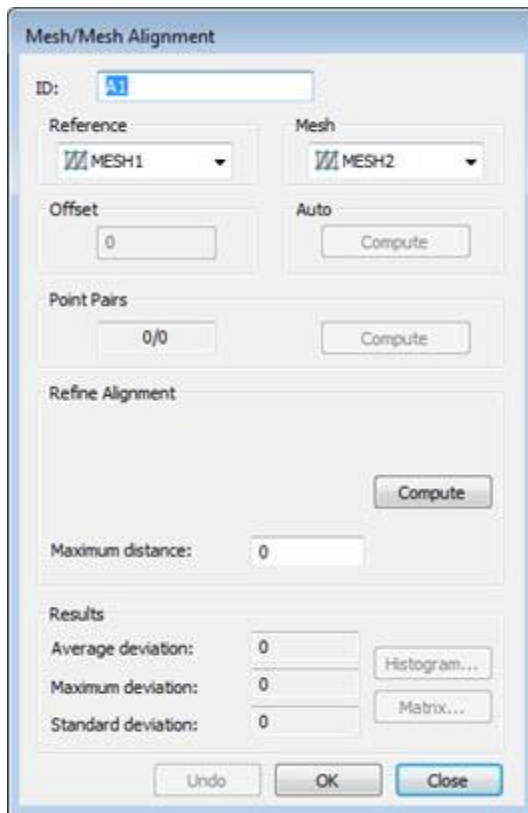
The alignment is done in two steps:

- A rough alignment, where pairs of points in the overlapping area of the two meshes are selected.
- A refined bestfit, which tries to bring the second mesh as close as possible to the reference mesh.

To create a Mesh to Mesh alignment, do the following:

1. Ensure that you have two or more Mesh commands in the measurement routine that you are using to align. These elements are required to align the two meshes.
2. Select the **Insert | Mesh | Alignment** menu option. You can also access this dialog box by typing the **MESHMESHBF** command in the Edit window's Command

mode between the [ALIGNMENT/START](#) and the [ALIGNMENT/END](#) commands.
The dialog box appears:

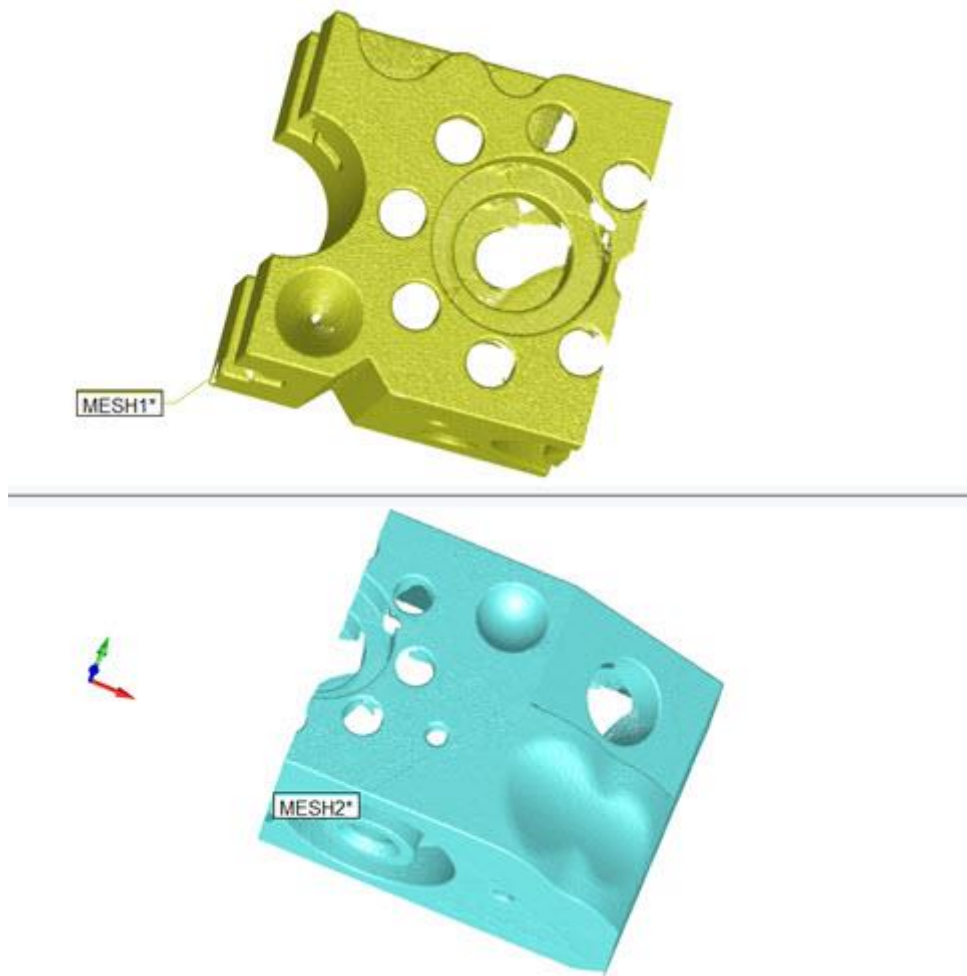


Mesh/Mesh Alignment dialog box



For a complete description of the dialog box, see the topic "Mesh/CAD Alignment Dialog Box Description".

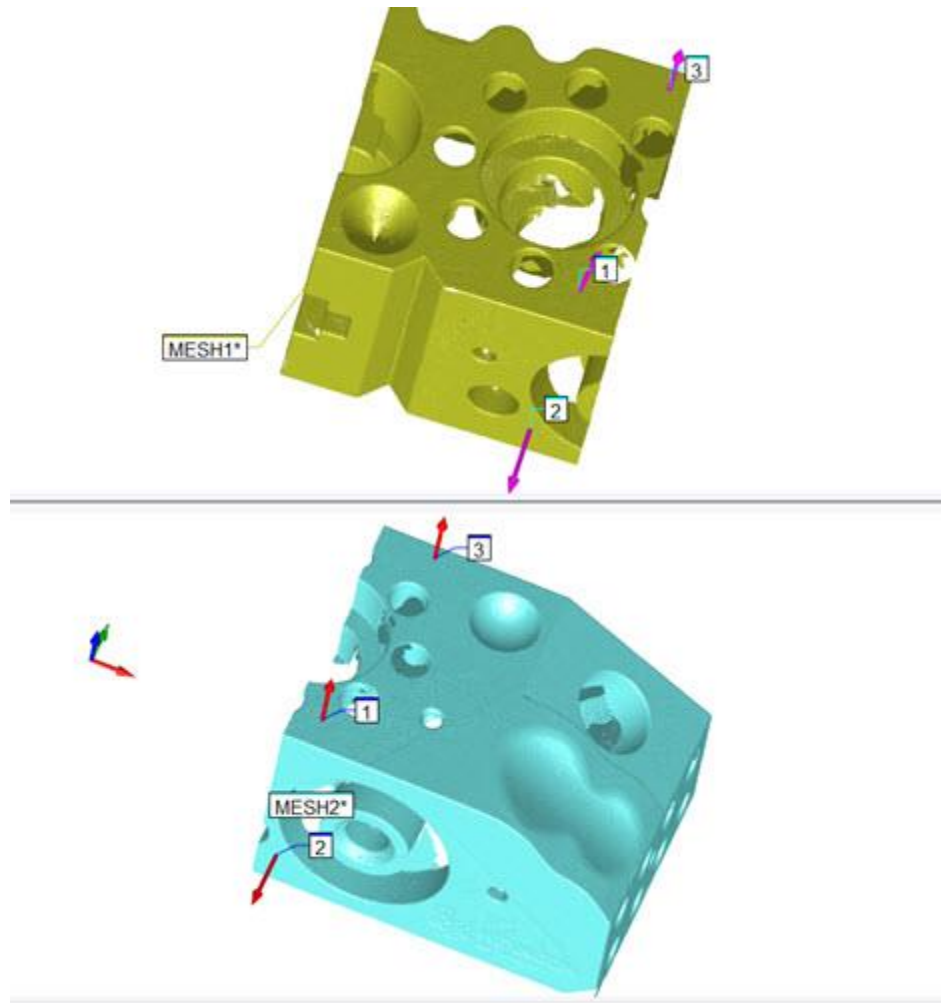
3. A temporary split-screen view of the two meshes appears in the Graphic Display window. You can use this view to visually see the alignment taking place. Select the first Mesh used as a point of reference from the **Reference** drop down list.



Split-screen view showing a mesh to mesh alignment

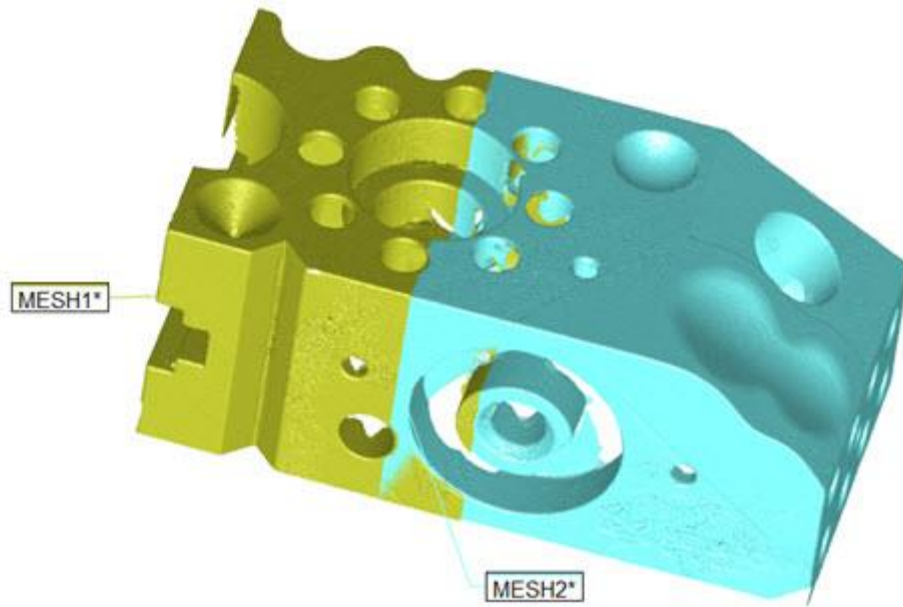
4. Use your mouse to manipulate and orient each view as needed to create the Point Pairs.
5. Perform the alignment:
 - a. Click the **Compute** button in the **Auto** section. You should only use this when you have a full scan of the external faces of the part. This automatically performs an alignment of the Mesh to the reference Mesh and also a refinement on the alignment as it's generating.
 - b. If the auto compute does not compute a good alignment, use the Point Pairs area to perform a rough alignment that brings the meshes close enough to each other. You can then refine the alignment further if needed. You should use this type of alignment if the mesh is not complete or it contains scanned data belonging to a fixture, table, or other similar feature.

- Click a desired number of points (at least three pairs) on each of the meshes on the overlap area. ONLY click points on the overlap area of the two meshes. ⓘ



Split-screen view showing selected MESH1 and MESH2 meshes

- The more points you take around the overlap area of the meshes results in an improved alignment. Click **Compute** to create the rough alignment.
- c. Next, use the **Refine Alignment** area whenever you want to refine your alignment, thereby bringing the two meshes closer to each other. In order to get a good refined alignment, the two mesh points should be close enough to each other through the initial rough alignment. ⓘ



A sample rough mesh to mesh alignment that needs a refinement

- i. Define the maximum distance between the points in the two meshes using the **Maximum Distance** box. The default value is 0 (zero). If the default value is used, PC-DMIS uses an internal default value related to the dimensions of the meshes.
 - ii. Click **Compute** to refine the alignment.
6. If a portion of one mesh doesn't align nicely with the other, you can click the **Undo** button and recompute the alignment using the same type of alignment with additional parameters, or you can try a different alignment.
7. Once you're satisfied with the alignment, click **Create**. PC-DMIS closes the temporary split-screen view and places the [MESHMESHBF](#) command in the Edit window. For details on the [MESHMESHBF](#) command, see the "MESHMESHBF Command Mode Text" topic in the PC-DMIS Laser documentation.

MESHMESHBF Command Mode Text

The [MESHMESHBF](#) command allows you to perform a best fit alignment of the reference mesh with a second mesh.

Below is an example code snippet for a [MESHMESHBF](#) alignment:

```
A1 =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
    MESHMESHBF/REFINE,SHOWALLPARAMS=TOG1,
```

```

ROUGH ALIGNPAIR/
    THEO/<x,y,z>,<i,j,k>,
    MEAS/<x1,y1,z1>
    REF,TOG2,TOG3,,
ALIGNMENT/END

```

TOG1 lets you show or hide the parameters used for the rough alignment. It can be set to YES or NO.

```

ROUGH ALIGNPAIR/
    THEO/x,y,z,i,j,k,
    MEAS/x1,y1,z1

```


These rough alignment pairs of points are defined/selected using the Graphic Display window. The values next to **THEO/** represent the point for the reference Mesh. The values next to **MEAS/** represent the corresponding point on the second Mesh. These pairs are used to determine a rough transformation between the reference Mesh and the second Mesh which allows the two meshes to come close to allow further refinements of the alignment.

TOG2 determines the reference Mesh used for aligning to the second Mesh.

TOG3 determines the second Mesh used for the alignment back to the reference Mesh.

Receive a Mesh from OptoCat



Use the **Receive a mesh from OptoCat** button () on the **Mesh** toolbar to place PC-DMIS in a state where it is waiting and ready to receive a mesh from the OptoCat application.

When mesh data is received:

- If the PC-DMIS measurement routine already contains a mesh command, the existing mesh data is replaced with the new mesh data.
- If the PC-DMIS Inspection plan does not contain a mesh command, a mesh command containing the new mesh data is inserted into the measurement routine.

- After the received mesh data is inserted into the measurement routine, the measurement routine is automatically executed.

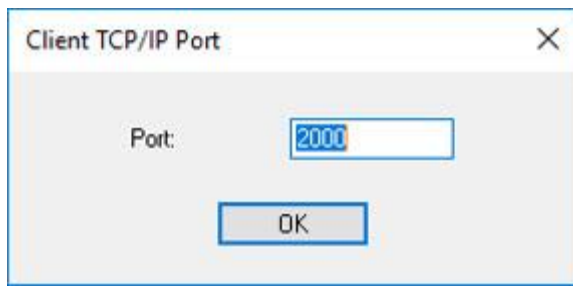
When clicked ON, the **Receive a mesh from OptoCat** button has a darker background

color: .

Click the button to toggle this function ON and OFF.

To use this function:

1. Open the measurement routine you are importing the OptoCat Mesh data into.
2. From the **Mesh** toolbar (**View | Toolbars | Mesh**), click the **Receive a mesh from OptoCat** button. The **Client TCP/IP Port** dialog box appears.



3. Update the **Port** field if necessary. The port assignment on the computer must match the port assignment of the OptoCat application.
4. Click **OK**. PC-DMIS is ready to receive mesh data from the OptoCat application.

Index

2

2 Points 192

2D Features 93

 Max Incidence Angle 93

2D Radius Gage 262, 264, 267

 Dialog Box 264

3

3D Features 87

 Max Incidence Angle 87

A

Aligning Clouds of Points 270, 271, 280

Animation Parameters 150

Annotation

 Opposing Points 163

 Show Opposing Points 163

Annotation Opposing Points 163

Annotations 162

 Thickness Colormap 162

Auto Feature (Laser) 87, 93, 298, 302, 303, 304, 307, 312, 314, 315

 Advanced Measurement Options 306

 Best Fit Math Type 306

 Command Buttons 307

 Feature Properties 304

 Measurement Properties 305

 Relative To 306

 Scan 293

Auto Feature Extraction 87, 93, 290, 295, 298

 without CAD Data 291

B

Big COP 127

Boundary Points 388

 Adding and Deleting 391

 Clearing 391

 Editing 390

 Setting by Typing 389

 Setting Using the CAD Data Method 390

 Setting Using the Measured Point Method 389

C

Calculation methods for laser surface point 312, 314, 315

Calibrate 3

 Laser Sensor 23

Calibration Sphere 23

 Manually Bisecting 36

Caliper 247, 248, 254, 259

 End Point 259

Mid Point 259	CLEAN 231
Start Point 259	CROSS SECTION 174, 183, 186, 192, 196, 199, 208
Caliper End Point 259	EMPTY 243
Caliper Mid Point 259	EXPORT 238
Caliper Start Point 259	FILTER 236
Changing a Zone's Color 171	IMPORT 244
Circle, Laser Auto 303, 328	POINT COLORMAP 164, 171, 225
Cloud of Points 51, 127, 131, 171	Pointcloud EXPORT 238
CMS 9	Pointcloud IMPORT 244
Eagle Eye 2 9	PURGE 235
CMS Sensor 9	RESET 242
Eagle Eye 2 9	SELECT 171
Color Bar Levels Area 166	SURFACE COLORMAP 164, 171, 210
Color Bar Profiles Area 169	COPALIGN Command 271
Color Scale Area 168	COPCADBF Command 271
Colormap 155, 157	COPCOPBF Command 271
Thickness 155, 157, 163	Creating a Mesh Operator 420
Compare to CAD Thickness Colormap 157	Creating a Mesh to Mesh Alignment 455
COP 51, 127, 131, 145, 150	Creating a Pointcloud to Pointcloud Alignment 271, 280
Big 127	Cross Section 186, 192, 196, 199, 208, 422
Graphical Representation 131	2 Points 192
Small 127	2D View 183
COP Command 150, 192	Distance Gage 199
COP/OPER Command 150, 153, 183	Hide 196
BOOLEAN 246	Reports 208

PC-DMIS 2019 R2 Laser Manual

Show 196

Cut Plane Vector 393

CWS Laser Probe 34

CWS Parameter 95

CWS/WLS Laser Probe 34

Cylinder, Laser Auto 303, 360, 364

Command Mode Text 364

Parameters 361

Paths 365

D

Data Filtering Section 138

DCC Machines 413

Manual Laser Scan 413

DCC Mode 307

Density Type 70

Dimensioning Surface Profile 222, 438

Display in Scene Area 170

Distance Gage 199, 208

Reporting 208

Viewing Labels in Reports 208

E

Eagle Eye 2 9

Edge Point, Laser Auto 318

Command Mode Text 323

Edit Color Scale 164

EMPTY a Mesh 445

EMPTY a Mesh Command 445

EMPTY Mesh 445

EMPTY Mesh Command 445

End Touch Vector 394

Error Handling 415

Example File Format

Thickness Colormap 163

Exclusion Plane Section 141

Execution Mode 105

Export COP/OPER 238

Export Mesh in STL Format 444

Extended Surface Point Calculation Method 315

Extracting a Surface Point from a Mesh 295, 298

F

Feature Extraction 79, 295, 298

Filters 85, 137

Flush and Gap, Laser Auto 339

Freeform Advanced Scan 405

G

Gage 247, 262, 264, 267

Caliper 247

Gages 247, 262, 264, 267

Caliper 247

Getting Started 3

Graphical Overlays 113

Gray Sum Settings 74

Grid Advanced Scan 408

H

HP-L-10.6 (CMS106) Sensor 1

Compared to HP-L-5.8 Sensor 19

Compared to Zeiss Eagle Eye 2 15

HP-L-20.8 Sensor 1

HP-L-5.8 Sensor 1, 19

HP-L-5.8A-SYSTEM (AJ) Sensor 1

HP-L-5.8T-SYSTEM (TKJ) Sensor 1

I

IDM 70

Import Mesh in STL format 443

Initial Touch Vector 393

Intelligent Density Management 70

L

Laser Data Collection Settings 137, 138, 141, 142

Data Filtering Section 138

Exclusion Plane Section 141

Pointcloud Display Section 142

Laser probe auto feature 312

Laser Probe Toolbox 40, 93

Laser AF Multiple Creation 99

Laser Clipping Region Properties 76

Laser Filtering Properties 54, 87, 93

Long Line Filter 57

Median Filter 60

Weighted Average Filter 63

Laser Pixel Locator Properties tab 73

Laser Scan Properties 44, 145

Position Probe tab 42

Controls 43

Positioning Your Laser Sensor 43

Laser Sensor Tab 6

Laser Surface Point 298, 314

Calculation methods 312, 314, 315

Using to measure 309

Laser View 107

Levels Area 166

Linear Open Advanced Scan 395

Creating 395

Parameters 396

M

Manual Laser Scan 413

DCC Machines 413

Max Incidence Angle 87, 93

Measure CWS/WLS Laser Probe 34

Measure Laser Probe Options 32

Measured Thickness Colormap 156

PC-DMIS 2019 R2 Laser Manual

Measuring Cross Section Distances 199

Mesh 248, 254, 295, 417, 419, 422, 434, 442, 445, 446

Aligning 455

Alignment 446, 447

Auto Feature Extraction 295

EMPTY Command 445

EMPTY Mesh Command 445

EMPTY Operator 442

Export in STL Format 444

EXPORT Operator 430

Extracting a Surface Point 295, 298

Import in STL format 443

IMPORT Operator 432

Operator 420, 422

OptoCat 460

Receive a Mesh from OptoCat 460

Thickness Colormap 155

Mesh Alignment 446, 447

Creating 449, 455

Mesh Alignment Dialog Box 447

Mesh Colormap 438

Dimensioning Surface Profile 438

Mesh COLORMAP Operator 434

Mesh CROSS SECTION 422

Mesh EMPTY Operator 442

Mesh EXPORT Operator 430

Mesh IMPORT Operator 432

Mesh Operator 420, 422, 432, 434, 442

EMPTY 442

EXPORT 430

Mesh to Mesh Alignment 455

Mesh Toolbar 124, 417, 422, 430, 432, 434, 445, 446

Alignment 446, 447

EMPTY Command 445

MESHALIGN Command 454

MESHCADBF Command 454

MESHMESHBF Align Command 459

MESHMESHBF Command 459

O

On Error Command 415

Operator 422, 430, 434, 442

Mesh EMPTY Operator 442

Mesh IMPORT 432

OptoCat 460

P

Patch Advanced Scan 398

PC-DMIS Laser 1

Perceptron Sensors 7

Perimeter Advanced Scan 401

Planar calculation method 312

Plane, Laser Auto 324	Pointcloud Export 238
Pointcloud	Pointcloud Import 244
Colormap 155	Purge 235
Pointcloud Alignment 127, 270, 271	Reset 242
Creating 280	Select 171
Pointcloud Alignment Dialog Box 271	Surface Colormap 164, 166, 210
Pointcloud Colormap 222	Toolbar 118
Dimensioning Surface Profile 222	Pointcloud Server 118, 285
Pointcloud Colors 116, 164	Pointcloud Surface Colormap 222
Pointcloud Display Section 142	Dimensioning Surface Profile 222
Pointcloud Export COP/OPER 238	Pointcloud to Pointcloud Alignment 271, 280
Pointcloud IMPORT 244	Pointcloud Toolbar 118, 271, 422
Pointcloud Mesh 248, 254, 419	Pointclouds 51, 118, 127, 131, 137, 145, 150, 170, 248, 254, 419
Pointcloud Operator 153	Graphical Representation 131
Animation Parameters for Pointcloud Simulation 150	Mesh 419
Boolean 246	Point Information 135
Clean 231	Simulate 145, 150
Cross Section 174, 183, 192, 196, 199, 208	Animation Parameters 150
Empty 243	Simulate Function 145, 150
Export 238	Animation Parameters 150
Filter 236	Probe Setup 15
Import 244	Zeiss Eagle Eye 2 15
Introduction 153	Zeiss I++ DME Server 15
Manipulating 154	Profiles Area 169
Point Colormap 164, 225	

Q

QuickCloud Toolbar 117, 124, 248, 254, 419

 Mesh 419

QuickFeature Implementation 303

QuickMeasure Toolbar 117

R

Receive a Mesh from OptoCat 460

Remove outliers 85

Reporting 208

Reports 208

Ring Band 84

Round Slot, Laser Auto 303, 333

 Parameters 334

S

Scan Line Indicator 111

Scanning 145, 377

 Auto Features 293, 307

 Boundary Points 388

 Colors 116

 Common Functions 378

 Freeform 405

 Graphical Representation of Vectors 393

 Grid 408

 ID 378

 Linear Open 395

Manual Laser 413

Manual Laser on DCC Machines 413

Measure 394

Patch 398

Perimeter 401

Point Cloud Reference Feature 394

Scan Parameters 378

Speeds 414

Surface 411

To Points conversion 381

Vectors Area 392

Select COP/OPER 171

Sequential Execution Mode 105

Show Annotation Opposing Points 163

Simulate 145, 150, 307

 Animation Parameters 150

 Scan Stripes 307

Simulate Pointcloud 145, 150

 Animation Parameters 150

 Function 145, 150

Simulate Pointcloud Function 145, 150

 Animation Parameters 150

Small COP 127

Sound Events 107

Sphere, Laser Auto 303, 373

- Paths 375
- Spherical calculation method 312, 314
- Spherical Surface Point Calculation Method 314
- Spline Points 386
 - Calculation Type 387
 - Curve Type 387
 - Increment 388
 - Point Spacing Type 388
 - Weight 388
- Square Slot, Laser Auto 303, 333
 - Parameters 334
 - Paths 337
- Surface Advanced Scan 411
- Surface Colormap 164, 166, 168, 210, 215
 - CAD model with Multiple Surface Profile Tolerances 215
- Surface Point, Laser Auto 298, 309, 314, 315
- Surface Profile 222, 438
 - Dimensioning 222, 438
- SurfacePointType registry entry 312
- T**
- TCP/IP Pointcloud Server 285
- Theoretical Points 385
 - Deleting 386
 - Editing 385
- Thickness Colormap 155, 156, 157, 162, 163
 - Annotation 163
 - Annotations 162
 - Compare to CAD 157
 - Example File Format 163
 - Method 159
 - Show in Report 162
 - Thickness Colormap Annotations 162
 - Thickness Colormap Method 159
 - Ray Based 159
 - Sphere 159
 - Toolbar 117, 118, 124
 - Mesh 124, 422, 430, 432, 434, 446
 - Mesh Alignment 446, 447
 - QuickCloud 117, 124, 248, 254, 419
 - Mesh 419
 - QuickMeasure 117, 248, 254
- U**
- Using the Mesh Commands 417
- Using the Simulate Pointcloud Function 145, 150
 - Animation Parameters 150
- V**
- Vectors 397
- W**
- WLS Laser Probe 34

Z

Zeiss I++ DME Server 15

Zeiss Eagle Eye 2 15

Glossary

C

CCD: Charge Coupled Device - This is one of the two main types of image sensors used in digital cameras.

Cloud of Points: The Cloud of Points command is a container for XYZ coordinate data. The data can be input from an external file, or it can come directly from a laser sensor through the referring scan command(s).

COP: The Cloud of Points command is a container for XYZ coordinate data. The data can be input from an external file, or it can come directly from a laser sensor through the referring scan command(s).

E

Exposure: This parameter controls the exposure of the Laser sensor.

G

Gauge side point: In a Flush and Gap Auto Feature, this is the point on the gauge surface side indicating where the flush should be measured. (also called the gauge point)

L

LWM: Laser Wrist Map

M

Master side point: In a Flush and Gap Auto Feature, this is the point on the master side surface where the flush is to be measured.

Mesh: A mesh is a set of vertices and triangles that are combined using a best fit algorithm to represent a 3D part shape.

milli-pixel: 1 milli-pixel = 0.001 pixel

O

Overscan: This parameter controls how far beyond the nominal feature's dimensions the probe will scan along both the major and minor axes of the feature.

P

Pointcloud: A Pointcloud is a collection of data points used to define a feature on a CAD model.

R

Row Overlap: This parameter controls how far each pass will overlap with the previous pass.

S

Sensor Frequency: This parameter controls the internal sensor frequency of the probe. The value that appears is the sensor pulses per second.

Surface CAD Model: A surface CAD model only has surfaces, and it does not create a solid. Some examples of this would be a plane feature, or a cylinder surface where there is no closed volume.