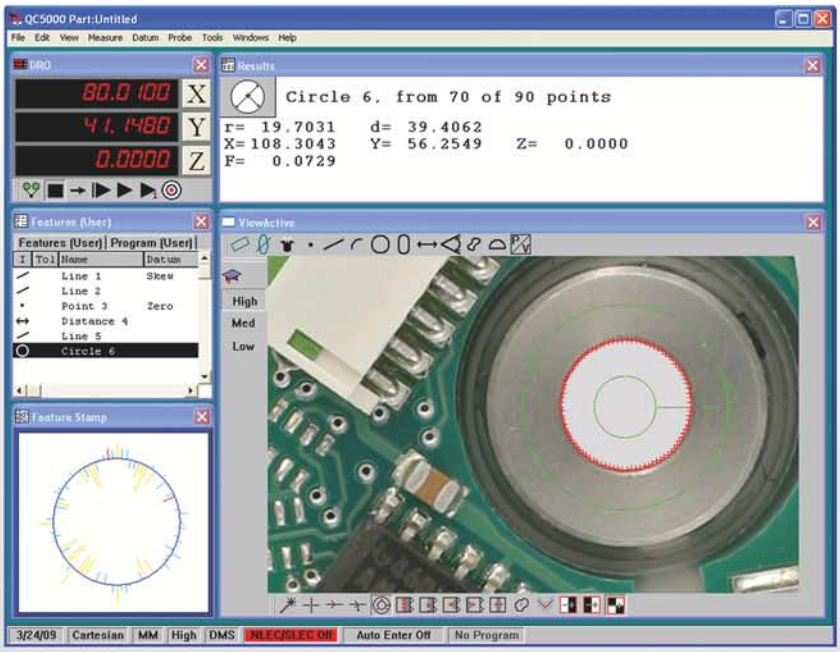
HEIDENHAIN



Addendum

IK 5000
QUADRA-CHEK

Software Version
3.4.x

English (en)
5/2017

Information Contained in this Manual

This addendum covers the operating and setup feature additions and improvements implemented in IK 5000 Software v3.0.x - v3.4.x. For OEM setup feature additions refer to the IK 5000 QUADRA-CHEK OEM Addendum (ID 1041353-2x), available by request from HEIDENHAIN.

Minimum System Requirements

Component	w/o 3D Profiling	w/ 3D Profiling
PC	2.66 GHz dual-core Pentium	2.8 GHz quad-core Pentium
Operating System (OS)	■ Windows Vista 32-bit ■ Windows 7 32-bit, Windows 7 64-bit ■ Windows 8 32-bit, Windows 8 64-bit ■ Windows 10 32-bit, Windows 10 64-bit	
RAM	1GB	2GB
Available Hard Disk	500MB	1GB
PCI	1 PCI slot and 1 to 3 additional empty slots (depending on the version)	
Video Display Unit	1024 x 768 resolution	
Windows users rights for installation and setup	Administrator	

Backward Compatibility

The IK 5000 QUADRA-CHEK v3.4.x software version is compatible with several previous versions down to version 2.93.0. It can be used to update any prior version starting at version 2.93.0 and up. To update a previous version refer to “Software Updates” below.

Software Updates

End users should only update the IK 5000 software after consultation with the machine manufacturer. Observe the system requirements as described in “Minimum System Requirements” above. For software updating procedures, refer to "Updating the IK 5000 Software on page 100”.

Fonts Used in these Instructions

Items of special interest or **concepts** that are emphasized to the user are shown in bold type.

`Software controls` and **`Windows`** are shown in letter gothic bold type.

1 Operation 11

2 Setup 101

1

Operation

1.1 Operation

About IK 5000 window information

The About window provides information about the system.

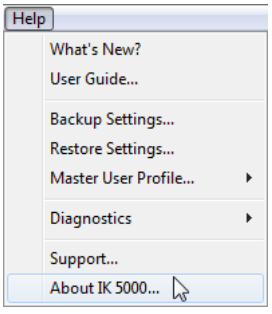
About window information includes:

- IK 5000 version number
- Configured options
- Temporary license time remaining
- IK 5000 ID and serial number (displayed with PCIe boards only)
- Axis board revision
- DSP software version
- Software copyright
- Website address

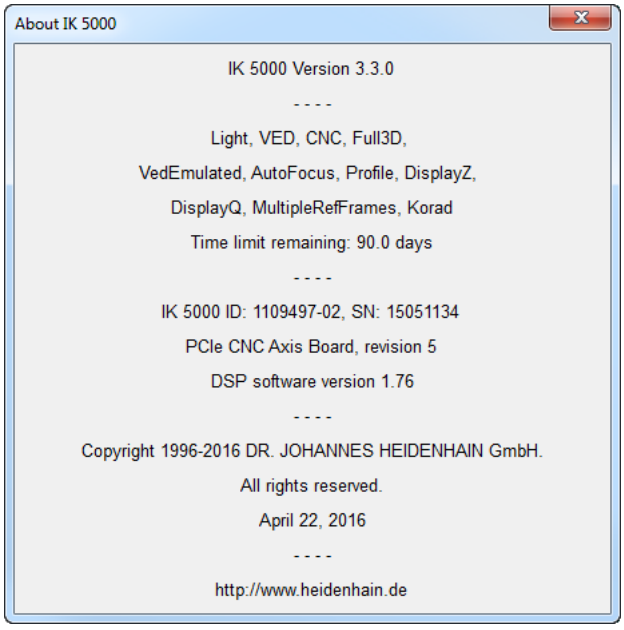
To open the About window:

▶ Click **Help>About IK 5000...**

The About window is displayed.



Click **Help>About IK 5000...**



The About window is displayed.

Adjusting for Change of Part Size

The location of features in a part under measure can change due to change in size of the part. A program can adjust to measure the features at the new locations by using the Prompt user for part scaling function. This function prompts the user for a scaling factor that repositions all features in a program. The new feature position is calculated using the recorded feature position multiplied by the scaling factor. The scaling factor is prompted for each time the program is run.

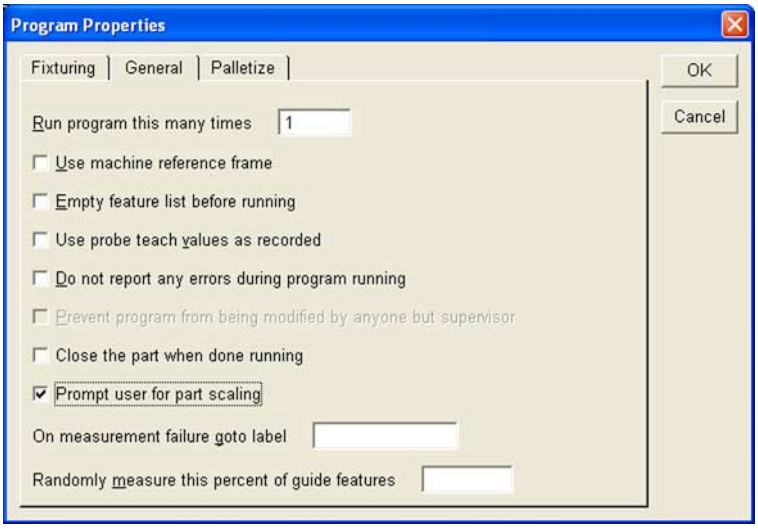
The Prompt user for part scaling function is located in the General tab of the Program Properties dialog.

The size of the features under measure in the program are not altered by the part scaling function, hence the system will probe the features at the recorded size but new scaled position.

To prompt a user for part scaling:

- Start recording a program
- Double click **Program Properties** in the **Program** template window
- Click the **General** tab
- Check the check box next to **Prompt user for part scaling** and then click **OK**

The operator will now be prompted to enter a scaling factor when the program is run.

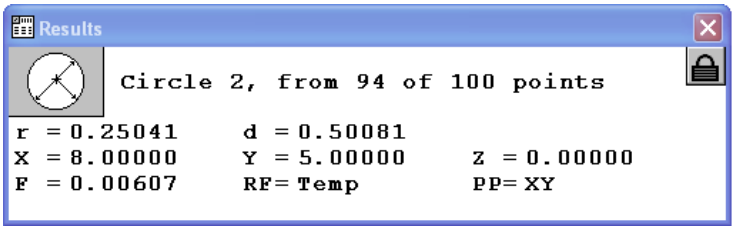


Selecting Prompt User for Part Scaling

Part Scaling Example

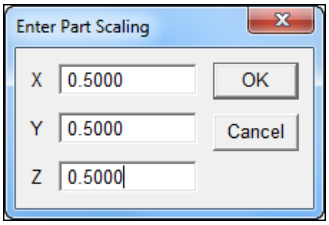
This example shows measurement results before and after part scaling has been applied.

- Start recording a program
- Measure a feature and note the position of the feature



Feature measurement results

- Stop recording the program
- Activate **Prompt user for part scaling** as described on page 11.
- Run the recorded program. A prompt will appear requiring the user to enter a part scaling factor for each axis.



Enter Part Scaling prompt

- Re-measure the feature and complete program execution

Note that the program feature CIRCLE 2 will now be probed at a position scaled by a factor of 0.500 (X=4.00000), 0.500 (Y=2.50000), 0.500 (Z=0.00000)

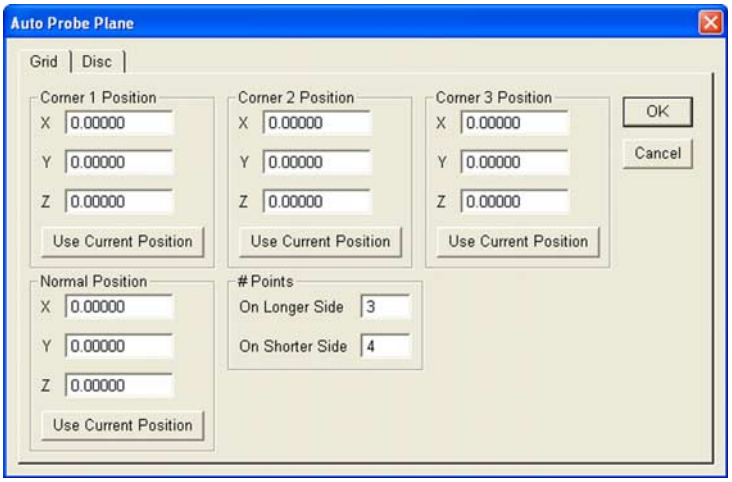
Auto Probe a Plane

Plane measurement accuracy is increased by probing more than the three required points. IK 5000 systems with CNC and touch probe options can automatically probe multiple points by using the **`Auto Probe Plane`** feature in the **`Measure Plane`** Dialog.

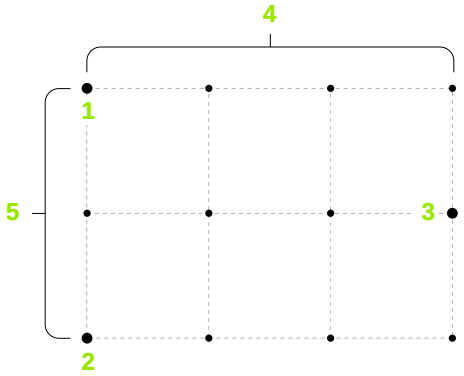
To auto probe a plane:

- Click the **`Auto Probe...`** button in the **`Measure Plane`** dialog.The **`Auto Probe Plane`** dialog appears.
- Select the **`Grid`** or **`DISC`** tab
- Probe the required positions and enter the required parameters
- Click **`OK`**

Grid Pattern Auto Probe



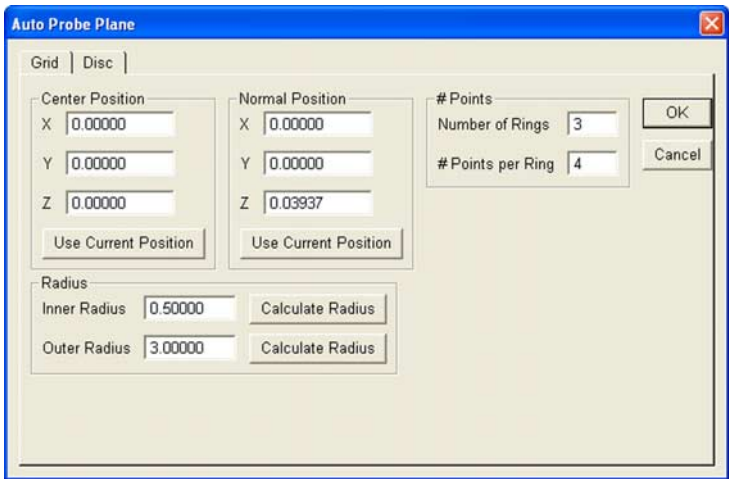
Grid dialog



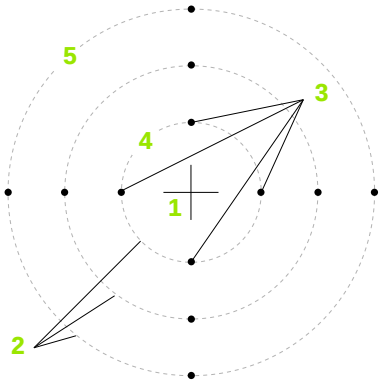
Grid pattern

	Parameter	Description
1	Corner position 1	The first corner position in the grid
2	Corner position 2	The second corner position in the grid
3	Corner position 3	The third corner position in the grid
4	# Points on longer side	The number of points on the longest side of the grid
5	# Points on shorter side	The number of points on the shortest side of the grid
	Normal position	A position above the plane for the probe to retract to after probing a point

Disc Pattern Auto Probe



Disc dialog



Disc pattern

	Parameter	Description
1	Center position	The center position of the disc pattern
2	Number of rings	The number of rings in the pattern
3	# Points per ring	The number of points probed per ring
4	Inner radius	The radius of the inner ring of the pattern
5	Outer radius	The radius of the outer ring of the pattern
	Normal position	A position above the plane for the probe to retract to after probing a point

Programming Changes

While recording a program that is intended to include a looping function (such as a loop, grid, polar grid, or random placement), it is important to record the entire program with the looping function before completing the program and running or editing it. This ensures proper handling of any datum steps in the loop, if any, and proper insertion of offset recall tags.

Tags are not displayed in the program.

When a feature is under measure during program playback, that feature is now removed from the feature list. Previously, the recorded feature was maintained in the feature list during measurement and replaced by the newly measured feature once the measurement was completed.

Grids, polar grids, and random placements are described in Programming Wizards on page 16.

Programming Wizards

Use programming wizards to create the following program steps:

- Grids
- Polar grids
- Random placements

Grids

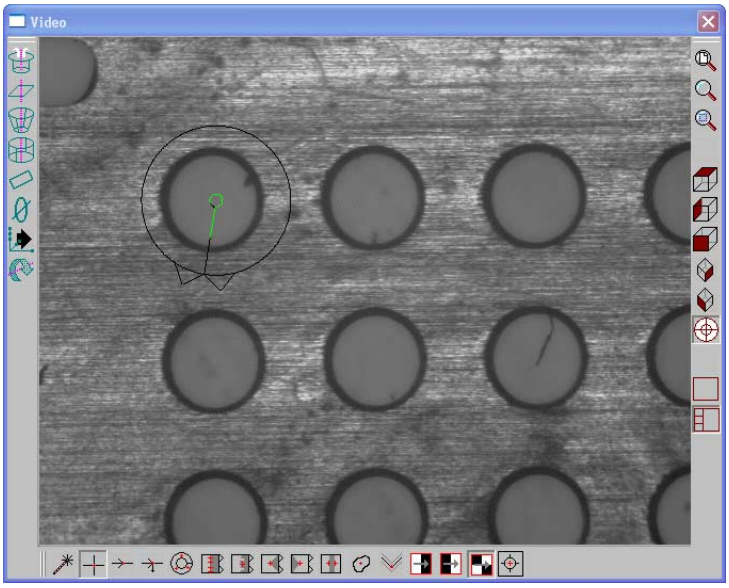
Use the Create a Grid wizard to measure arrays or a series of like features within a single part. For example, create a grid to measure an array of evenly spaced holes on a pc board. The IK 5000 requires the user to input the X and Y offset distance, the X and Y distance from one feature to the next.

All features in a grid must be evenly spaced.

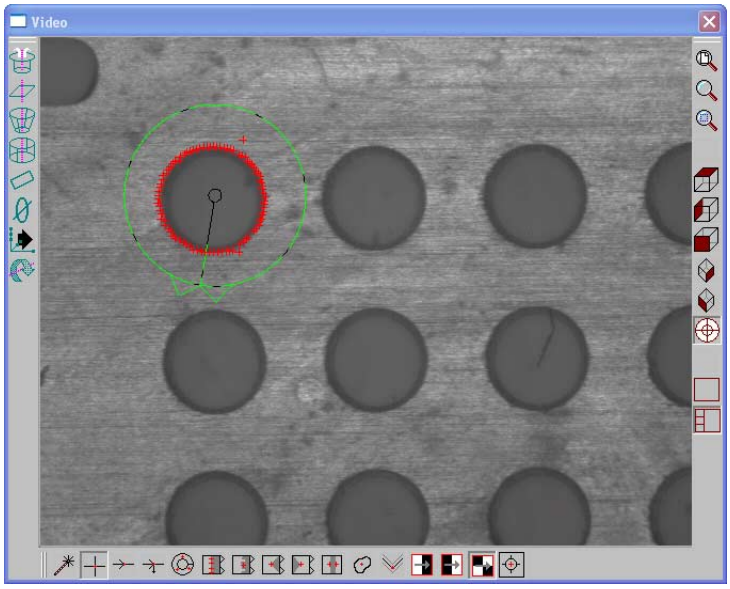
Before creating a grid begin a new part program with a datum.

To create a grid:

▶ Position the first array feature as shown

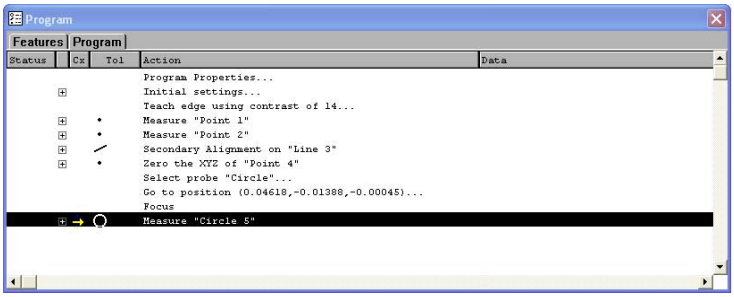


- Click the center mouse button to fire the probe tool

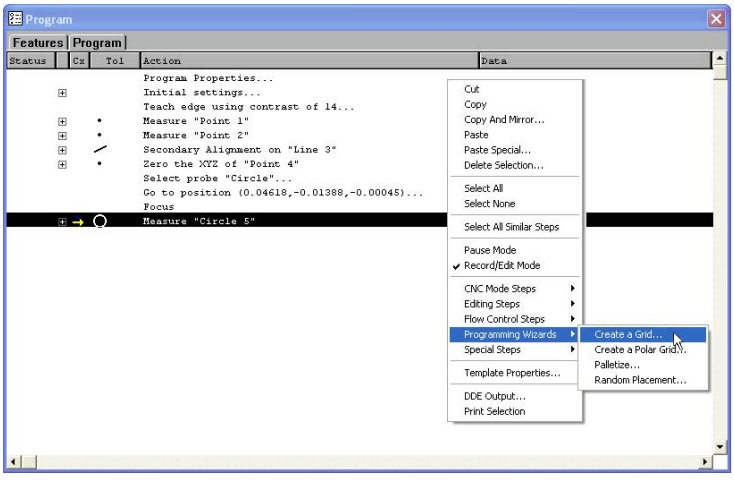


- Click **OK** in the dialog box

- Highlight the first array feature in the **Program** template as shown

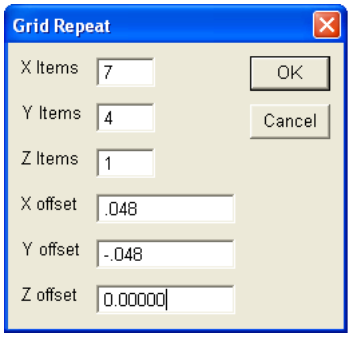


- Right-click in the **Program** template and select **Programming Wizards**, then select **Create a Grid**



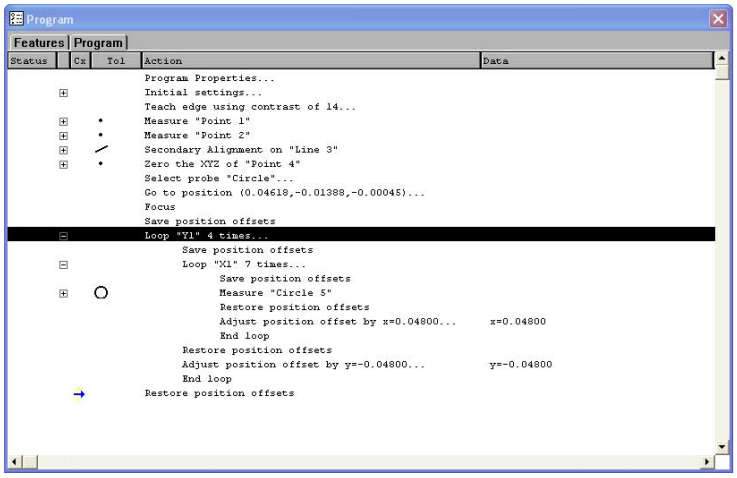
▶ Enter the number of features and type in the desired X and Y offsets as shown

Items are the number of features in each axis direction. Offsets are the distance to the next feature relative from previous feature.



▶ Click **OK** in the dialog box

The grid is created and the program is ready to run



▶ Click **New Run** to run the program

Polar Grids

A polar grid measures a part with like features that are in a rotational array. For example, create a polar grid to measure a bolt hole pattern. The programming wizard that creates a polar grid requires the user to input the number of features and the angle between the features.

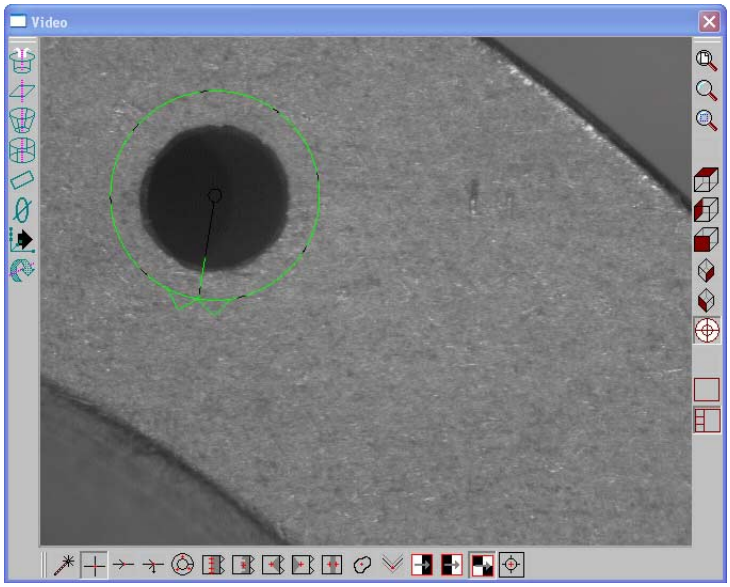
All features in a polar grid must be evenly spaced.

Before creating a polar grid, create a new part program. Use the center of the polar grid features as the zero point of the part.

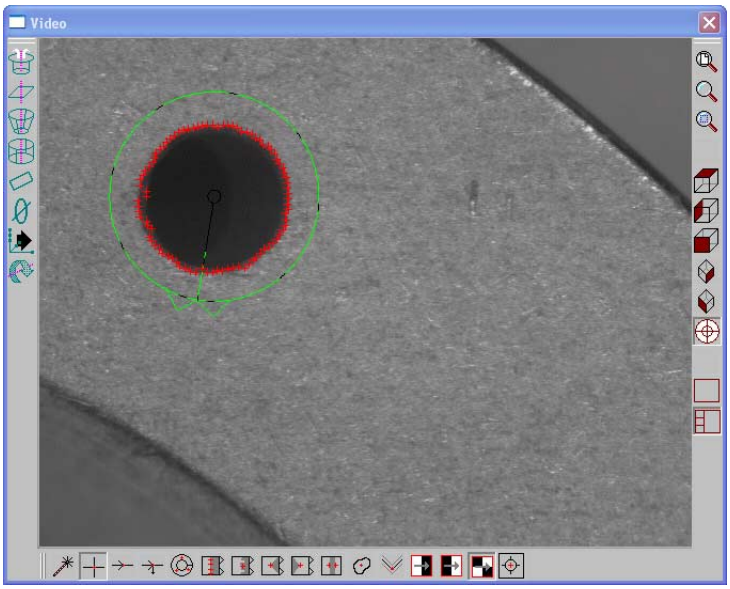
Make certain the program is recording before completing the following steps.

To create a polar grid:

- Position the first feature in the polar array in the center of the field of view as shown

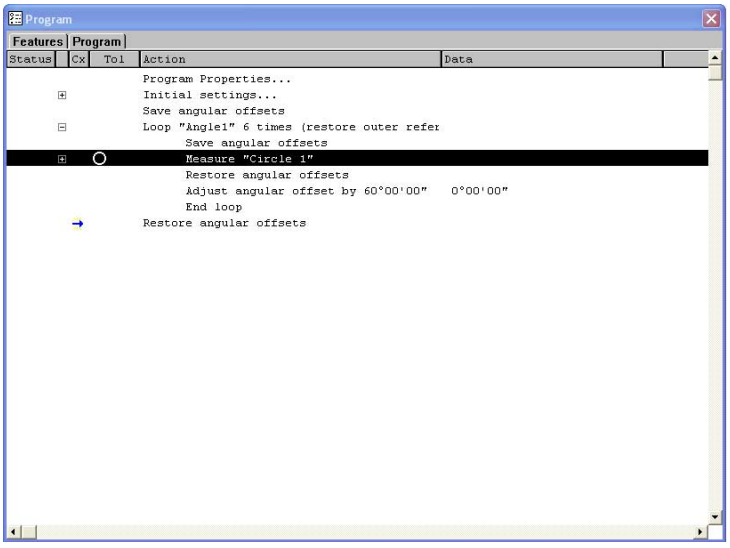


- Select **Measure>Circle**



- Click **Enter Point** in the **Measure Circle** dialog
- Click **OK**
- Select the measured **Circle** in the **Program** template
- Select **Tools>Programming>Programming Wizards>Create a Polar Grid**
- Enter the number of times to loop in the **Times to Repeat** field
- Enter the loop increment angle in the **Angle Increment** field
- Click **OK**

The polar grid is created and the program is ready to run.



Random placement

Use the Random Placement wizard for multiple parts on a stage without a fixture.

All parts must have the same orientation on the stage and be square to the axes of the machine.

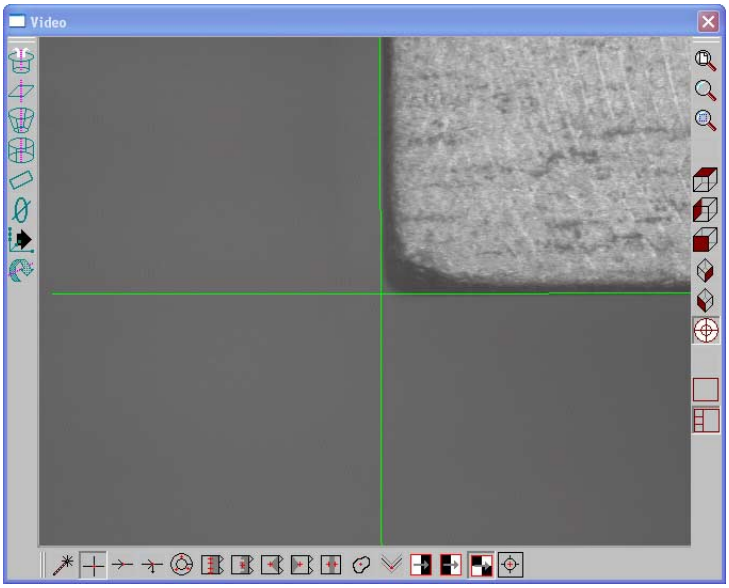
Before creating a random placement, create a new part program with reference steps.

Reference Steps

To insert reference steps in a part program:

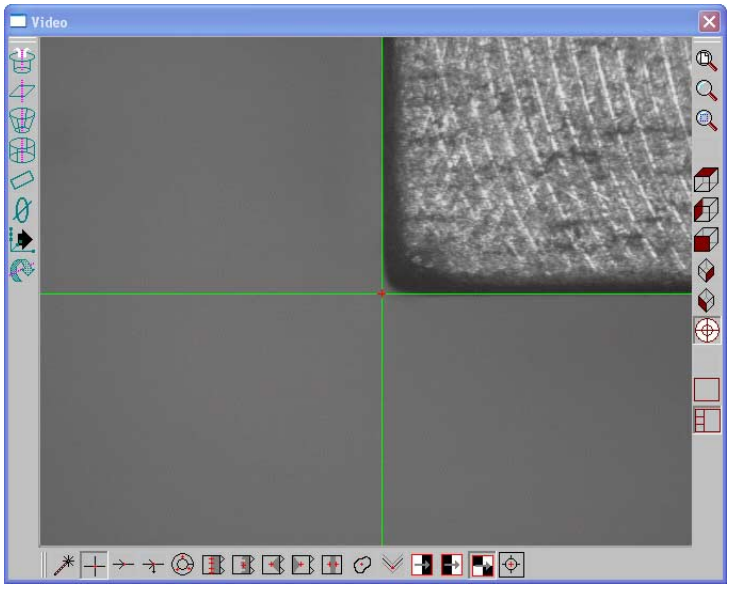
- Click the **Record/Edit** button on the **Program** toolbar

- Position the desired reference feature as shown

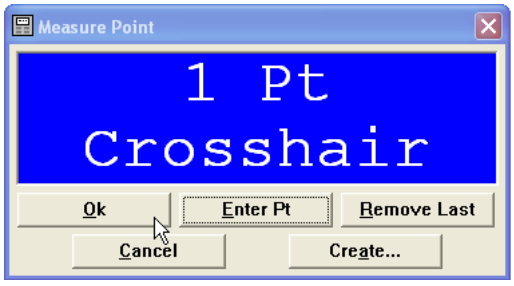


The reference feature is a repeatable point to help the IK 5000 locate the randomly placed parts.

▶ Click the center mouse button to fire the probe tool



▶ Click OK in the dialog



The IK 5000 will use this feature to locate the randomly place parts.

Create a the rest of the part program measuring all the features of the part.

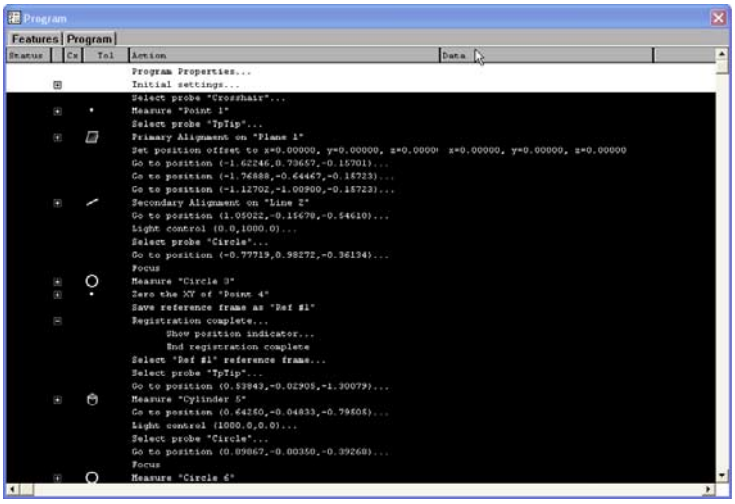
Save the reference frame in the part program immediately after creating the datum.

Random Placement

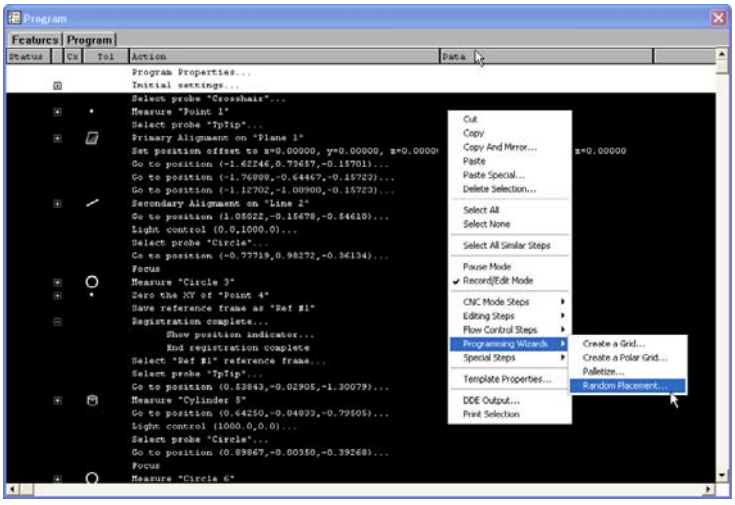
To create a random placement:

Make certain the program is recording before completing the following steps.

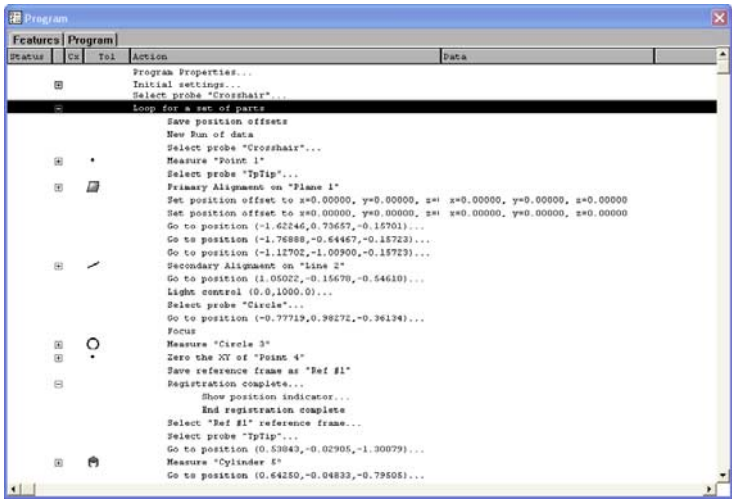
- Highlight the desired features, and the reference point as shown



- Right-click in the **`Program`** template and select **`Programming Wizards`**, then select **`Random Placement`**

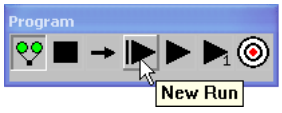


The IK 5000 inserts the required steps for the random placement.

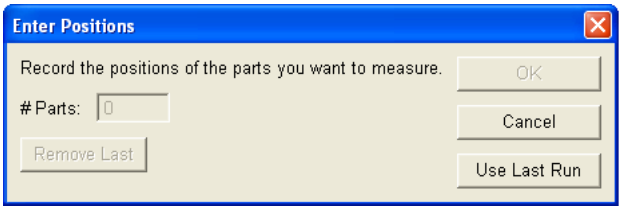


To run a parts program with a random placement:

- Click the **New Run** button on the **Program** toolbar

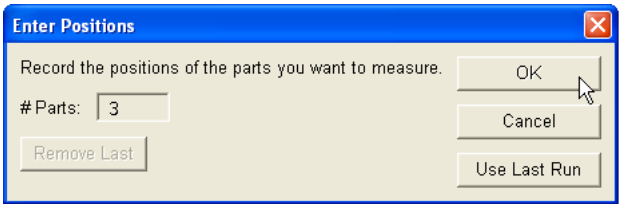


- Measure the reference point for the first part



Repeat these steps until all parts are entered.

- Click **OK** in the dialog box



Click the **Remove Last** button to remove the last point enter. Reenter the point for that part.

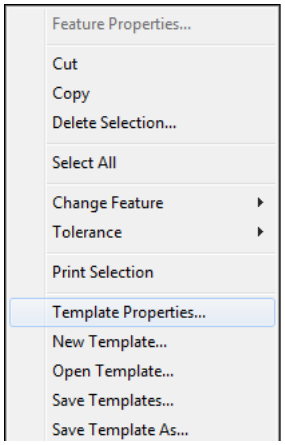
Click the **Use Last Run** button to use the same offsets as the previous run.

Adding a Program Variable to a Template Header or Footer

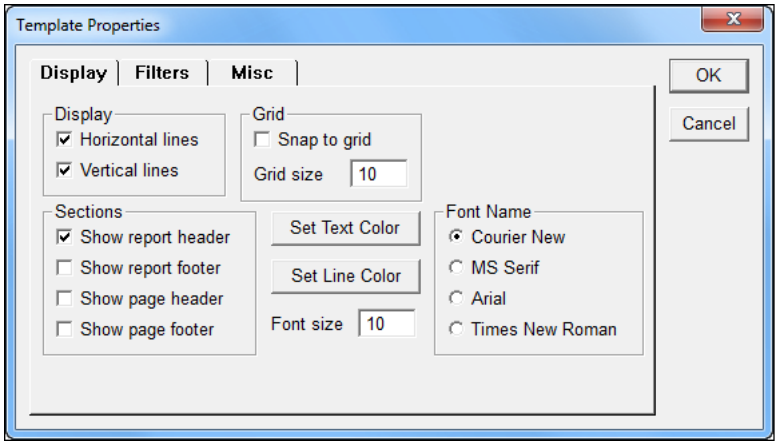
A program variable can be added to a report template header or footer. When a variable is added to the template it will be included in the header or footer when the report is displayed and printed.

To add a variable to a report template:

- Open or record a program that includes a variable
- Right click inside the Program window and select **`Template Properties...`**
- Check the **`Show report header`** or **`Show report footer`** checkbox

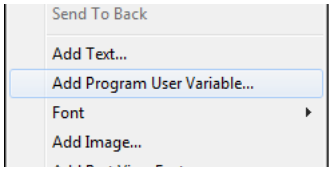


Select **`Template Properties...`**

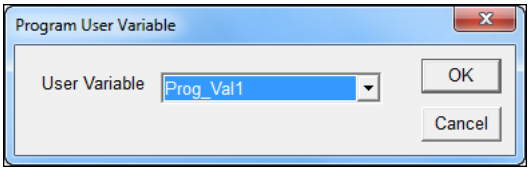


Check **`Show report header`** or **`Show report footer`**

- Right-click inside the header or footer area and select **`Add Program User Variable...`**
- Select the desired variable from the **`User Variable`** drop-down list and click **`OK`**

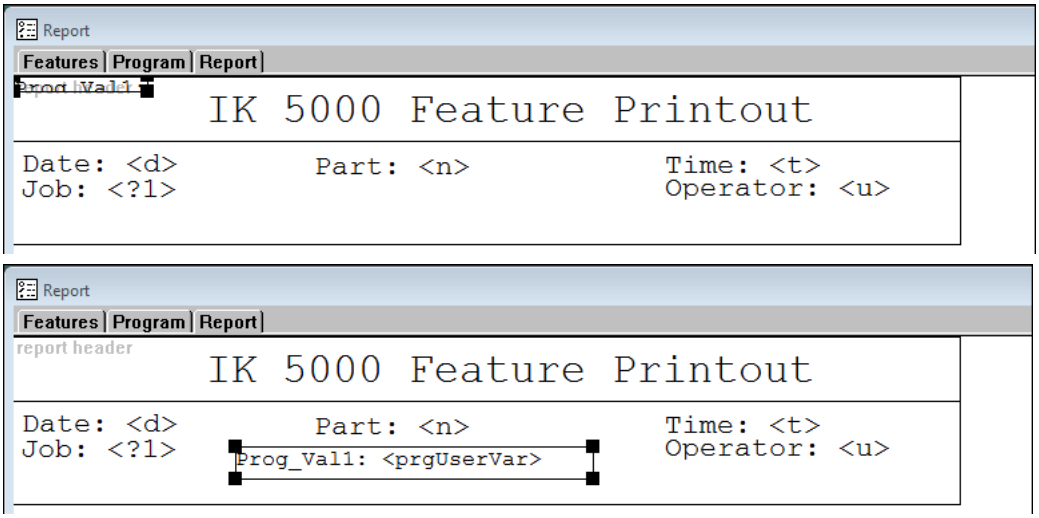


Select **`Add Program User Variable...`**



Select the desired variable

- Move the variable to the desired position in the header or footer



Move the variable to the desired position

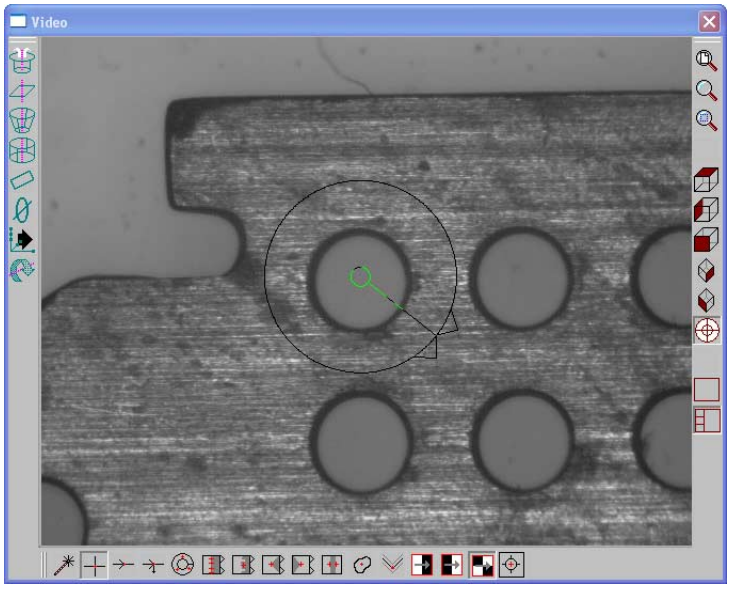
The program variable will be added to the header or footer when the report is printed.

Copy and Paste Special

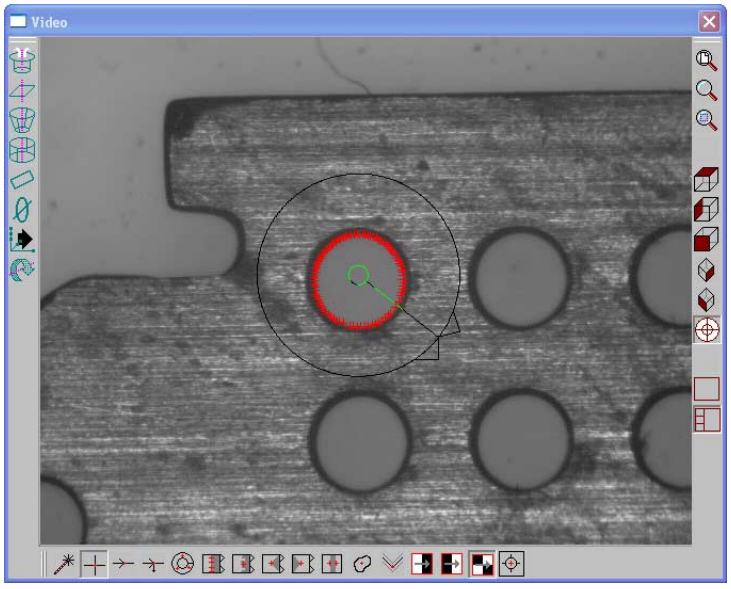
Before performing a copy and paste special, record a new part program with a datum.

To copy and paste special:

- Position the first array feature as shown

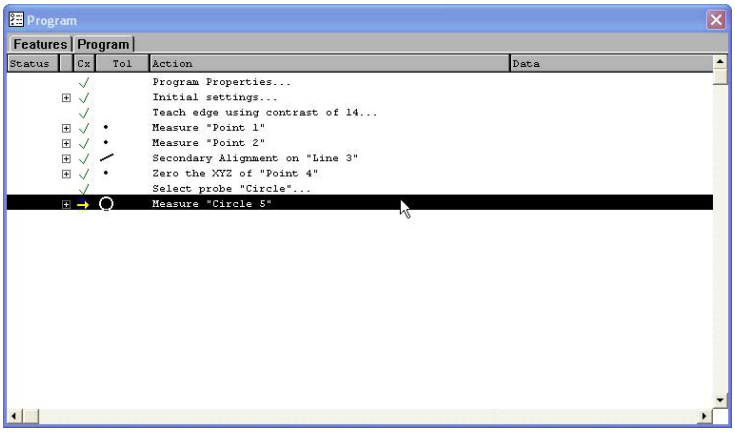


- Click the center mouse button to fire the probe tool

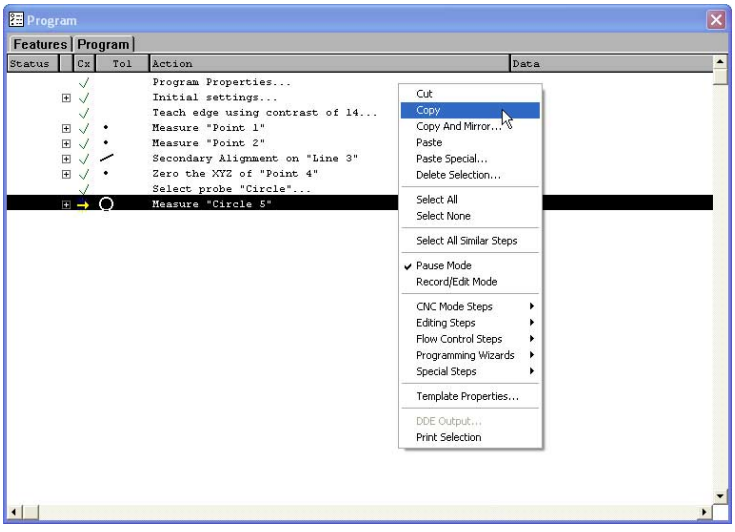


- Click **OK** in the dialog box

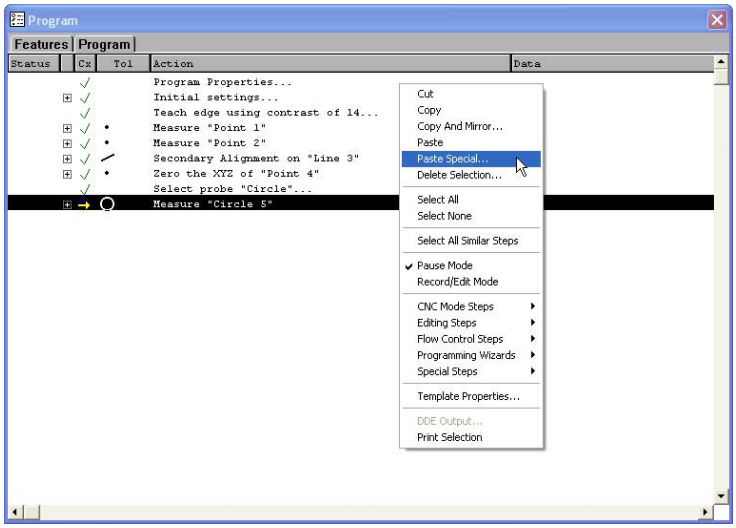
- Highlight the first array feature in the **Program** template as shown



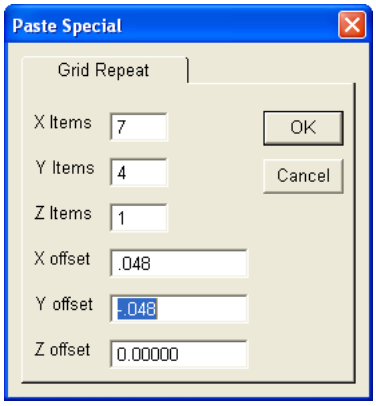
- Right-click in the **Program** template and select **Copy**



- Right-click in the **Program** template and select **Paste Special**

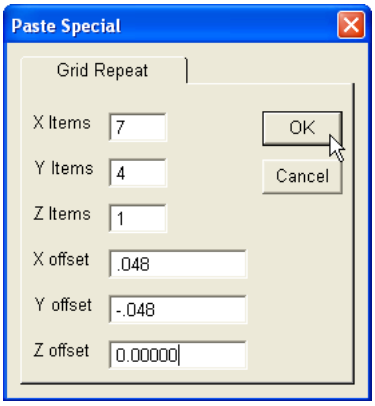


- Enter the number of **Items** and type the desired **X** and **Y offsets** as shown

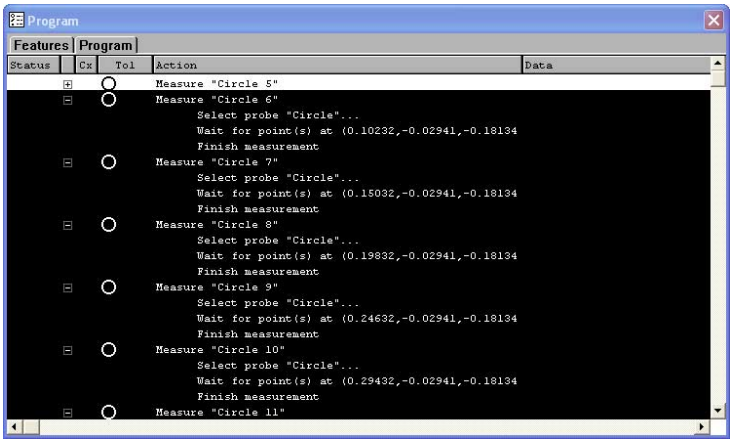


Items are the number of features in each axis direction.
Offsets are the distance to the next feature relative from previous feature.

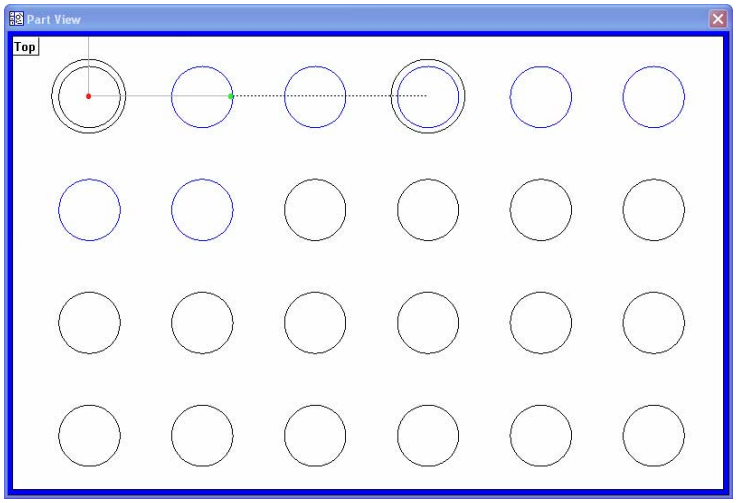
- Click **OK** in the dialog box



The IK 5000 creates a grid on the part view and assigns a numbered feature name to each feature in the grid.



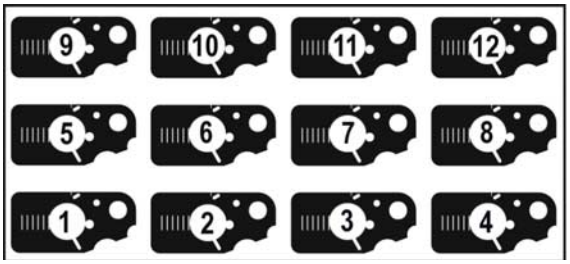
The copied and pasted parts appears in the **Part View** window as shown.



The program is ready to run.

Palletizing Multiple Parts

A pallet is a rectangular arrangement of identical parts organized into a consistent matrix of columns and rows. The X and Y offsets between parts in the matrix must be the same for all parts.

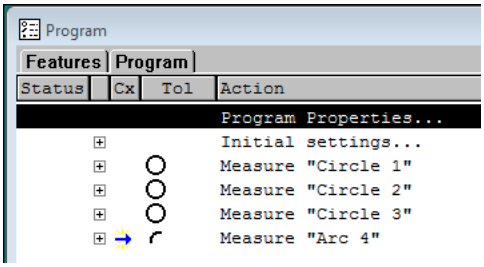


Pallet of 12 identical parts

Once a part program is created and tested for a single part, it can be run on each of the identical parts contained in a pallet. The IK 5000 will begin measuring parts at the lower-left corner of the pallet, and will proceed from left to right and from bottom to top.

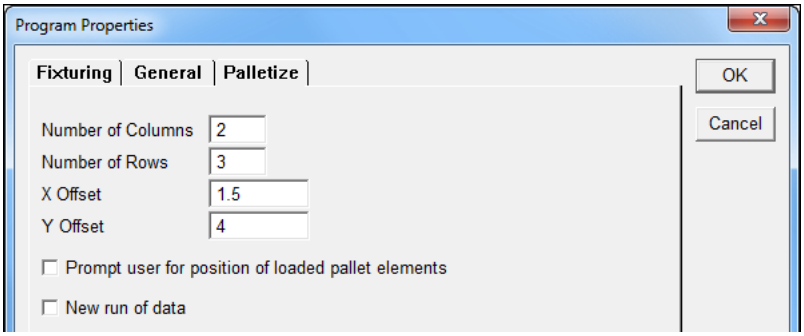
To specify the palletizing mode:

- Double click **Program Properties** in the **Program** template. The **Program Properties** dialog is displayed.



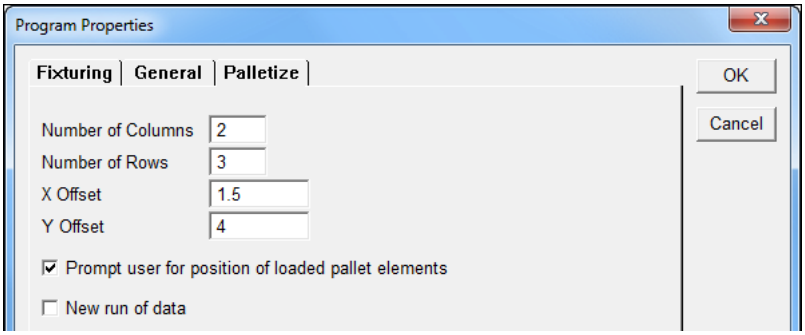
Double click **Program Properties** in the **Program** template

- Select the **Palletize** tab. The pallet setup parameters will be displayed.



Select the **Palletize** tab

- Enter the **Number of Columns** and **Number of Rows** into the fields provided
- Enter the **X Offset** and **Y Offset** between parts into the fields provided
- When it is known that some pallets will not be full, the program can prompt the operator at the beginning of each run to specify the location of parts contained by the current pallet. Check the **Prompt user for position of loaded pallet elements** check box to prompt the operator.
- Click **OK**



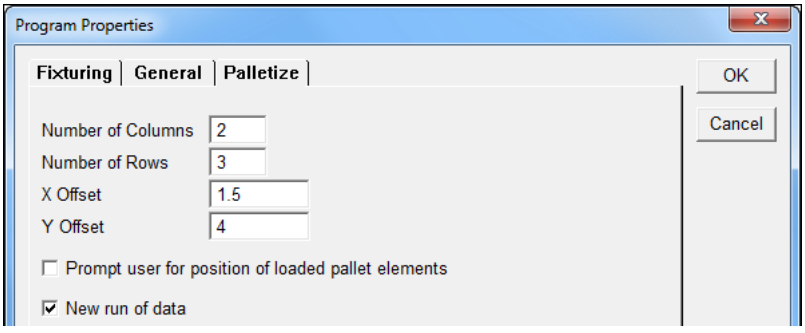
Enter pallet parameters and click **OK**

New run of data

New data can be entered into the runs database for every pallet instead of appending the data to the previous pallet's data.

To enter new data into the runs database:

- Check **New run of data** in the Program Properties window

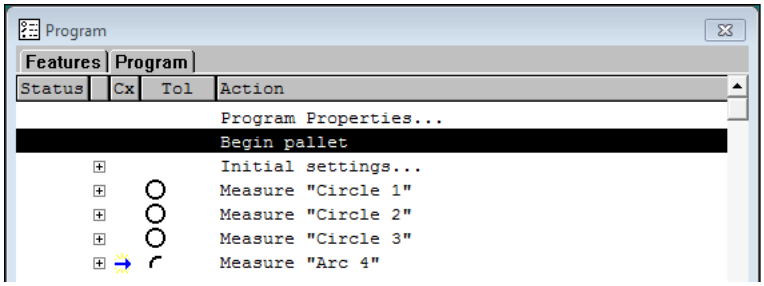


Check **New run of data**

- Click **OK** to save the changes and exit the Program Properties window

Begin pallet

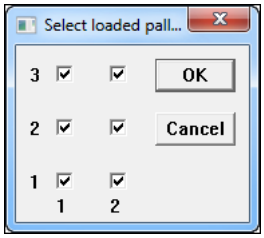
A **Begin pallet** step is inserted into the program when the palletize function is used.



Begin pallet step

Prompt user for position

When **Prompt user for position of loaded pallet elements** is enabled, the system will prompt you to enable or disable elements that are available on the pallet when the program is played back.



Enable or disable elements on the pallet

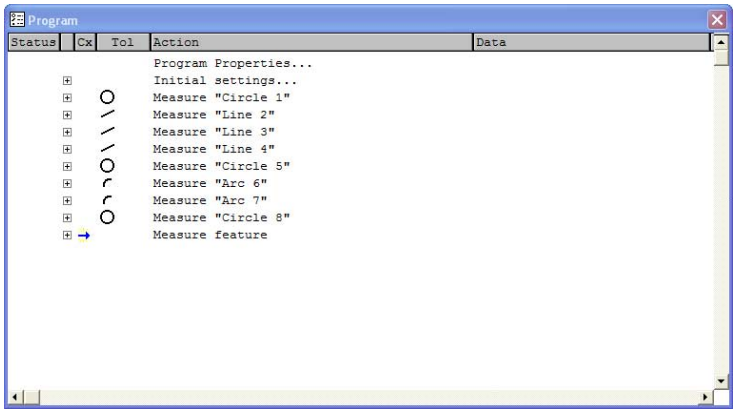
Profile Import Points

Probed points from features can be imported into a profile measurement when recording a program. This eliminates the need to re-probe points for contours that make up features already measured in the same program.

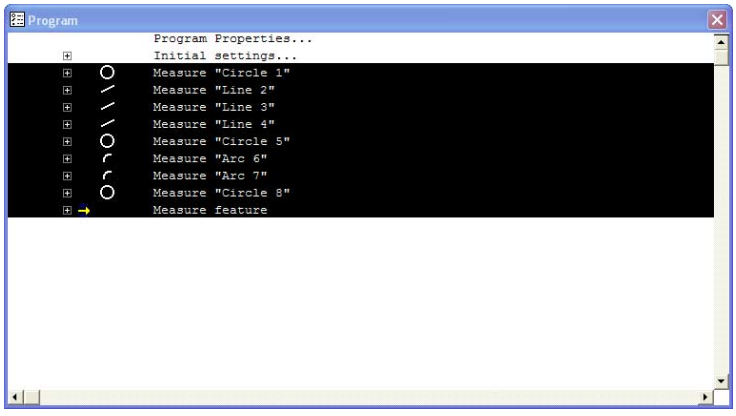
All features whose points are used in the import process need to exist during program playback or the import function will fail.

To import points into a Profile measurement:

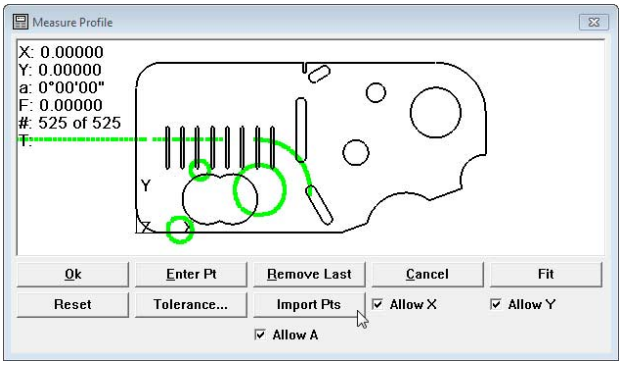
- Start recording a program
- Measure features into the program by probing them



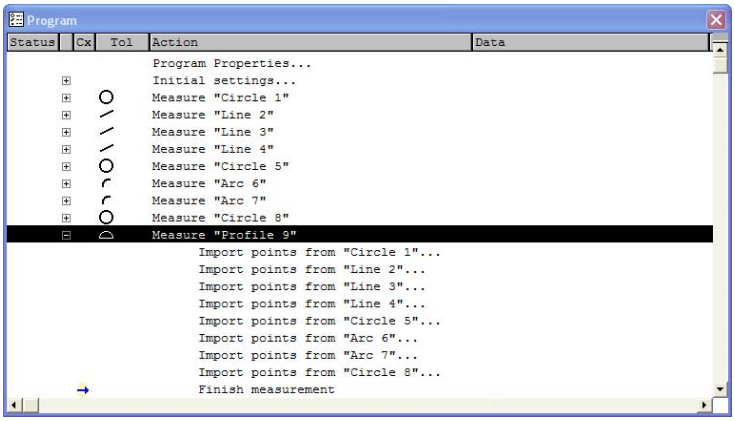
- Start a **`Profile`** measurement
- Select the features whose data cloud will be imported into the measurement as probed points



▶ Click **Import Pts**



Points are imported into the Profile measurement.



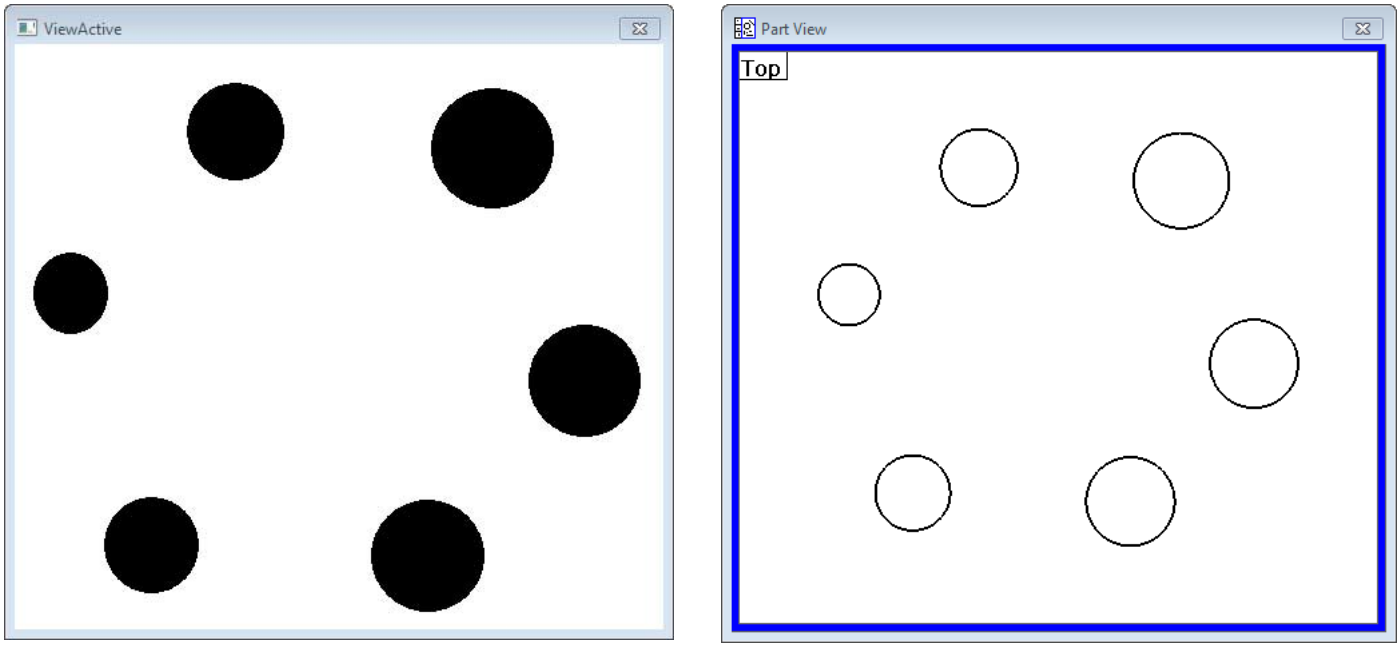
▶ Click **OK**

Measuring maximum inscribed and minimum superscribed tangent circles

The IK 5000 can measure maximum inscribed and minimum superscribed tangent circles. The circles can be measured by constructing a circle from a set of circles that are in a bolt-hole pattern.

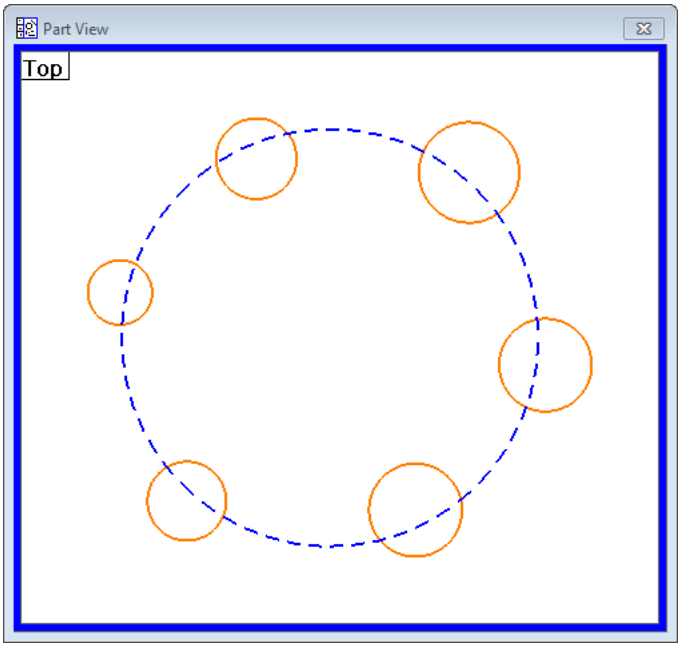
To measure a maximum inscribed or minimum superscribed tangent circle:

- Measure a set of circles that are in a bolt-hole pattern



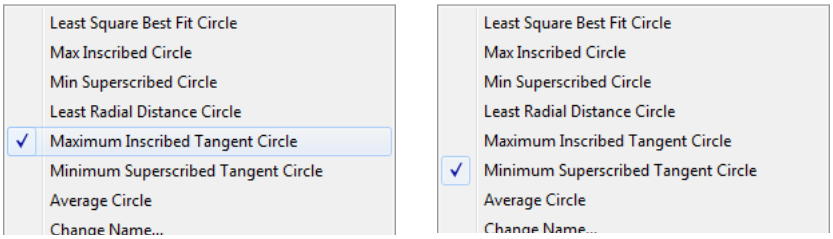
Measure a set of circles

- Construct a circle from the measured circles



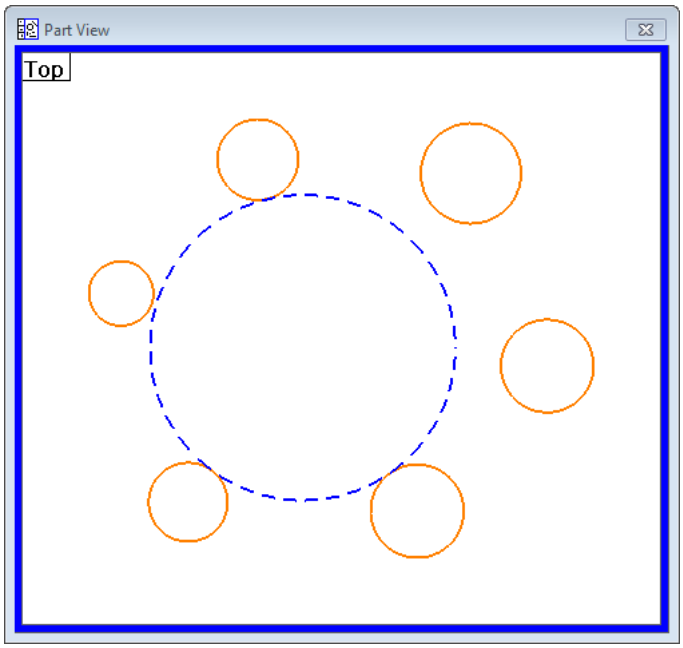
Construct a circle

- Right-click on the Results window
- Select **`Maximum Inscribed Tangent Circle`** or **`Minimum Superscribed Tangent Circle`**

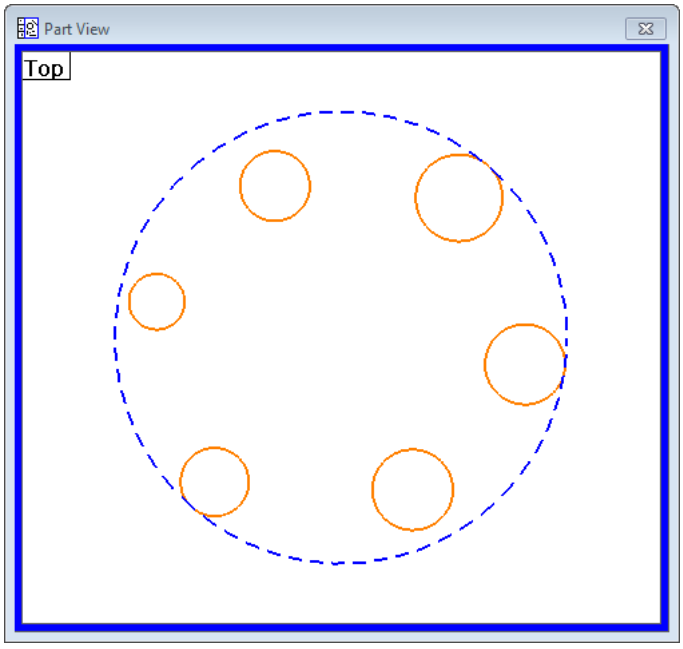


Select **`Maximum Inscribed Tangent Circle`** or **`Minimum Superscribed Tangent Circle`**

A Maximum Inscribed Tangent Circle or Minimum Superscribed Tangent Circle will be measured.



Maximum Inscribed Tangent Circle



Minimum Superscribed Tangent Circle

Measuring Rectangles

Precisely five probed points are required to measure a rectangle. The five points must be located in an exact pattern, and can be probed in either clockwise or counterclockwise order.

The correct pattern of points probed around the rectangle are:

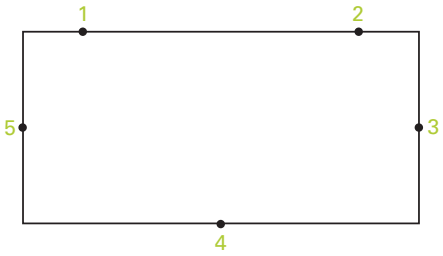
- Two points evenly distributed along one long side
- One point on the closest end
- One point on the approximate center of the second long side
- The last point on the remaining end

Probing a different pattern of points, or probing out of either clockwise or counterclockwise order will result in erroneous rectangle measurements.

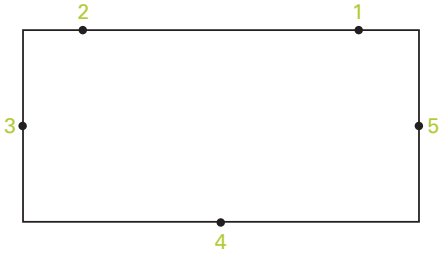
Measurement results are shown in the Features template, Part View window and Results window. The Results window shows:

- The feature identification number
- The number of points used in the measurement
- The length and width of the rectangle
- The angular orientation of the rectangle
- The coordinate location of the center of the rectangle

Only the rectangle shape and orientation are shown in the Feature stamp of the Results window.



Clockwise probing order



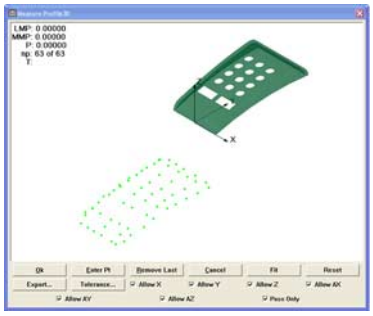
Counterclockwise probing order

Profile3D Measurements

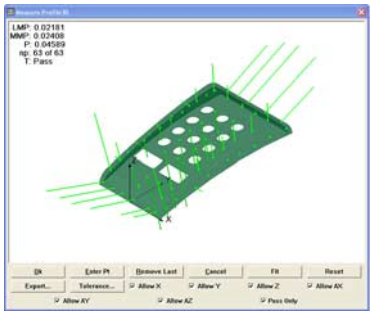
Profile3D measurements compare probed part surfaces to nominal part profiles from drawing or part data files. Probed surfaces and nominal profile surfaces are compared in the Measure Profile3D window.

The nominal part profile can be imported into the Measure Profile3D window from a drawing file in .IGS or .STP formats.

As surfaces are probed, their points are displayed in the Measure Profile3D window. When the desired surface measurements are complete, a fit operation is performed. During the fit operation, the system positions the probed data over the nominal profile and adjusts the X, Y, Z and rotational orientation of the probed points to achieve the best fit. Data points are shown as dots and form errors are shown as whiskers.



Probed points shown in Measure Profile3D window

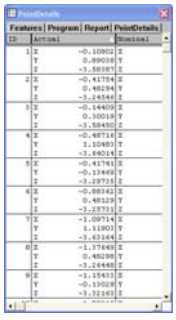


Fit operation performed and form error whiskers shown

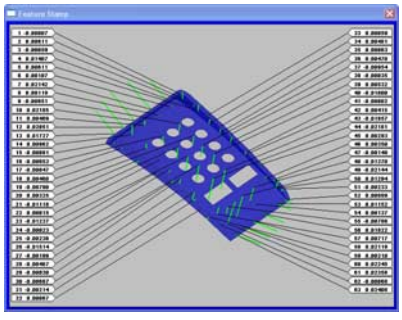
Tolerances can be applied to Profile3D measurements as bilateral limits or as unilateral pass/fail boundaries. Bilateral tolerances can be distributed equally or unequally on opposite sides of a surface.

The form errors, the number of data points and tolerance pass/fail results can be displayed in the Measure Profile3D window at the conclusion of the Fit operation. Upon completion of the Profile3D measurement, the Profile3D feature is listed in the Features template.

The form error of each surface point is listed in the Points Detail template and can be visually inspected using the Profile3D Feature Stamp.



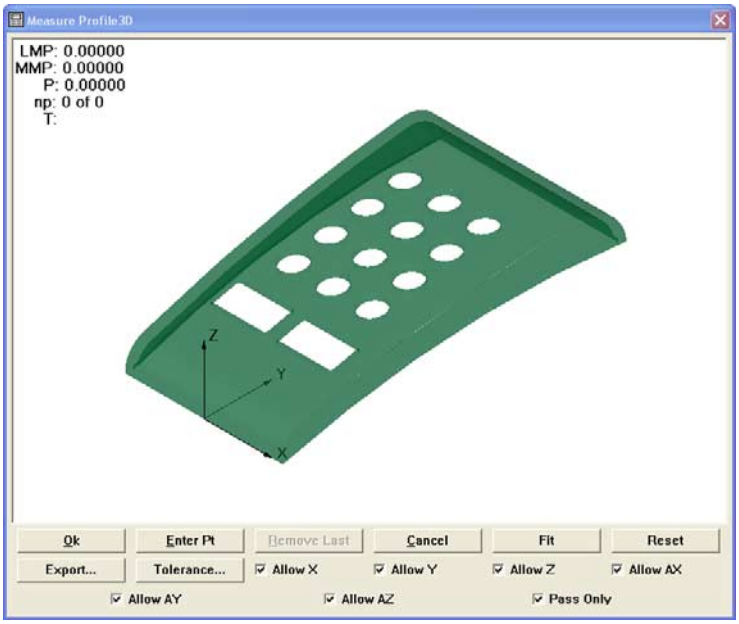
Points Detail template



Profile3D feature stamp

Profile3D Measurement Screen Functions

Functions for conducting Profile3D measurements are located across the bottom of the Measure Profile3D window. These functions are discussed in detail later in the Profile3D measurement instructions.



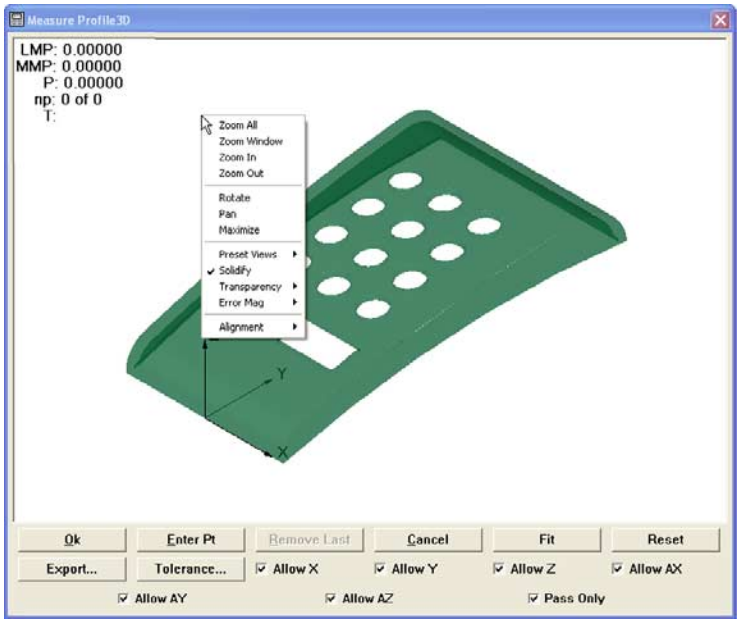
Measure Profile3D window

The Measure Profile3D window functions are shown below:

Functions	Function descriptions
OK	Complete the Profile3D measurement
Enter Pt	Probe part surfaces
Remove Last	Remove the last probed points
Cancel	Cancel the Profile3D measurement
Fit	Fit probed data to the nominal Profile3D
Reset	Reset the fit operation
Export	Export the adjusted Profile3D model
Tolerance	Assign surface tolerances
Allow X, Y, Z	Allow (or prohibit) data shifts during Profile3D fit analyses
Allow AX, AY, AZ	Allow (or prohibit) data rotation during Profile3D fit analyses
Pass Only	Enable (or disable) Pass Only option

Profile3D Window Menu

Functions for evaluating and displaying Profile3D measurement results are contained in the Measure Profile3D window menu. Right-click anywhere in the Measure Profile3D window to display the Measure Profile3D menu.



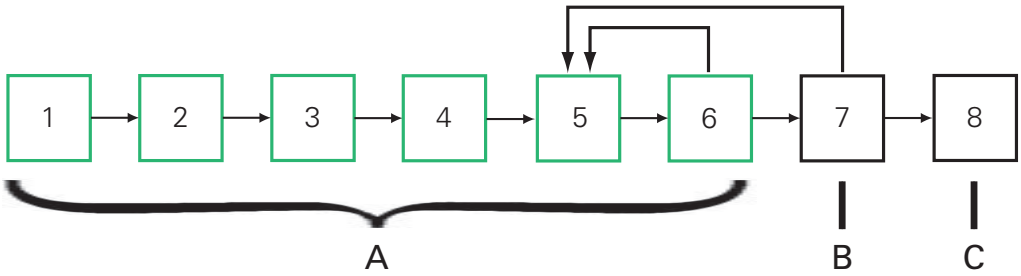
Measure Profile3D window menu

The Measure Profile3D window menu functions are shown below:

Functions	Function descriptions
Zoom All	Zoom to fit model in window
Zoom Window	Click and drag to select an area to zoom in on
Zoom In	Zoom in to show more detail
Zoom Out	Zoom out to show less detail
Rotate	Rotate the model around the datum
Pan	Move the model in the measure window
Maximize	Expand the measure window to the full size of the IK 5000 application window
Preset views	Set the viewpoint of the model to a predefined view
Solidify	Toggle between a wireframe and solid model
Transparency	Adjust the percentage of transparency for surfaces of a solid model
Error Mag	Magnify the display of form error whiskers
Alignment	Adjust the model to match the orientation of the part to the shape

The Profile3D Measurement Process

Typical steps required to complete a Profile3D measurement are shown below. Details regarding the use of measurement windows, menus and toolbar tools are included in the instructions.



Profile3D measurement process

Profile3D measurement process steps are shown below:

	Requirement	Step	System activity
A	Measure	**1**	Start Profile3D measurement
A	Measure	**2**	Import part profile
A	Measure	**3**	Enable data shifts for fit
A	Measure	**4**	Assign tolerances
A	Measure	**5**	Probe part surfaces
A	Measure	**6**	Perform Profile3D fit analysis. Go to step 5 and probe more points if needed.
B	Review and adjust results	**7**	Adjust display magnification of error whiskers. Go to step 5 and probe more points if needed.
C	Complete measurement	**8**	Add Profile3D feature to the Feature template

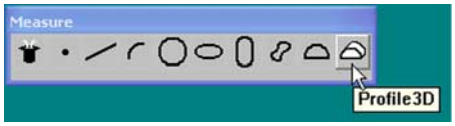
Start Profile3D Measurement

To begin a Profile3D measurement:

- Click **Measure>Profile3D**

or

- Click the **Profile3D** icon button on the **Measure** toolbar

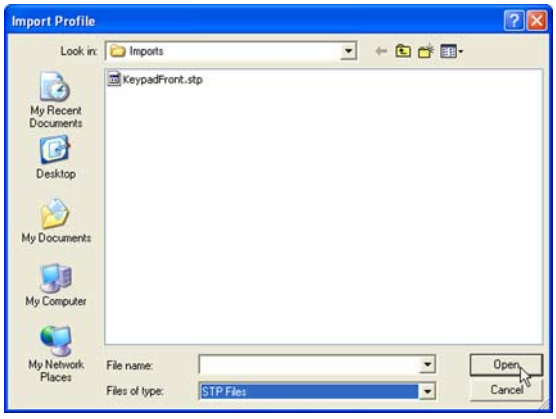


Start a Profile3D measurement from the Measure toolbar

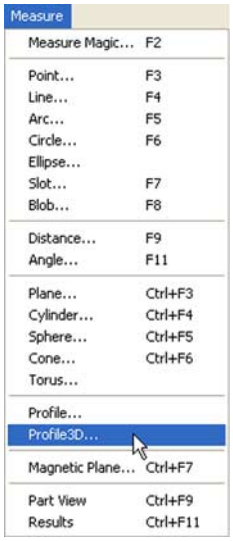
The **Import Profile** dialog will be displayed. A nominal part profile can be imported from a drawing file. Drawing files can be in the .IGS or .STP format.

Import Part Profile

- Select the desired .IGS or .STP file and then Click **Open**

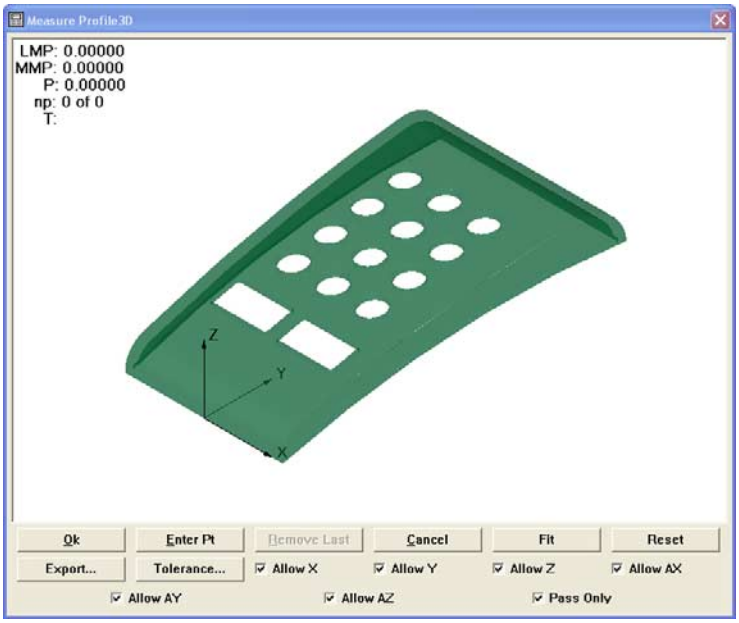


Select file



Start a Profile3D measurement from the Measure menu

The nominal Profile3D will be shown in the Measure Profile3D window.



Nominal Profile3D shown in Measure Profile3D window

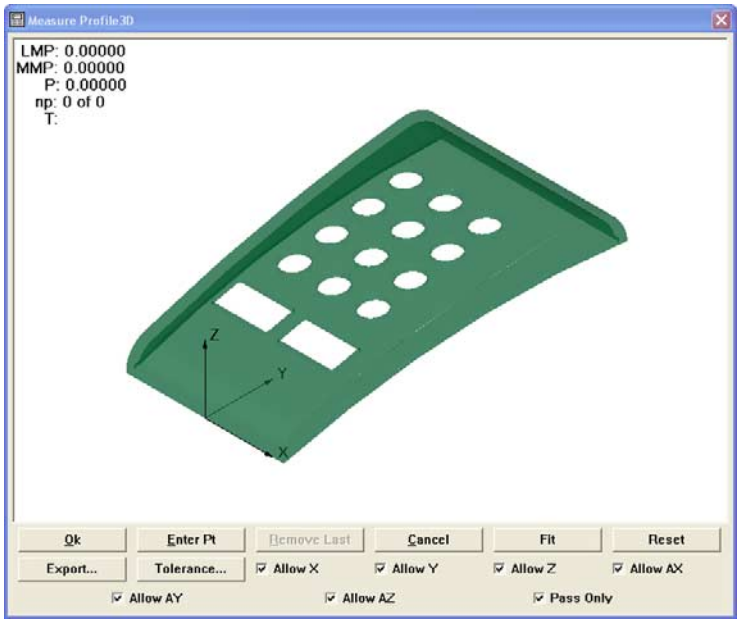
Enable or Prohibit Data Shifts for Profile3D Fit

The Profile3D fit algorithm shifts the probed data points in the cartesian or polar coordinate system to achieve the best fit between the probed points and the nominal part profile.

Maximum degrees of freedom are given to the fit algorithm by checking the **Allow X, Allow Y, Allow Z, Allow AX, Allow AY** and **Allow AZ** check boxes across the bottom of the window. Clearing a check box prohibits data movement in the indicated orientation.

To give maximum degrees of freedom to the Profile3D fit algorithm:

- Check the **Allow X, Allow Y, Allow Z, Allow AX, Allow AY** and **Allow AZ** check boxes across the bottom of the **Measure Profile3D** window



Check all check boxes for maximum degrees of freedom

- Clear a check box to prohibit movement of data in the indicated orientation

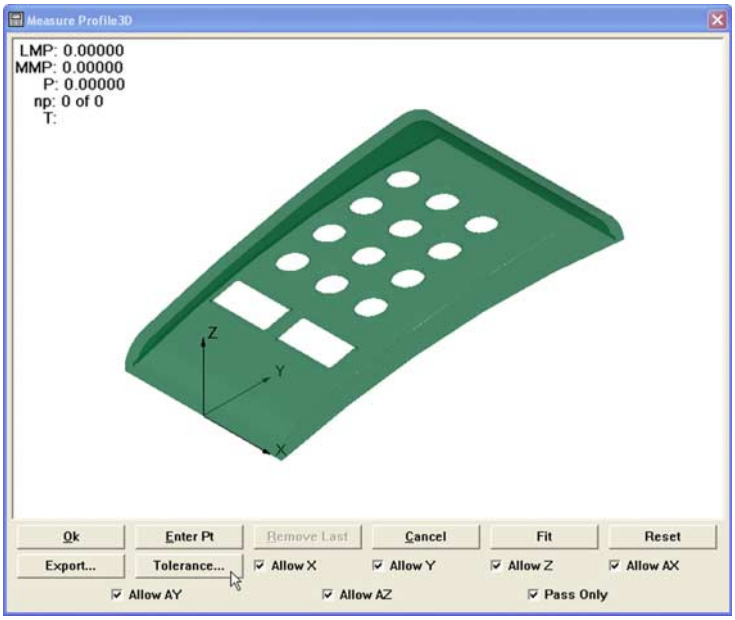
Allowing the maximum degrees of freedom for the Profile3D fit algorithm is recommended for most applications.

Assign Tolerances

Bilateral or unilateral tolerances can be assigned to Profile3D measurements. Bilateral tolerances can be equal or unequal.

To begin applying tolerances:

- Click the **`Tolerance`** button to display the **`Profile Tolerance Edit`** dialog



Begin applying tolerances

Equal Bilateral Tolerances

Bilateral tolerance values will be centered on the surface of the nominal profile. For example, an equal bilateral tolerance of 0.005 will be applied as ±0.0025.

To apply an equal bilateral tolerance:

- Check the **Bilateral** check box, enter the desired tolerance value into the **Tolerance** field for equal bilateral tolerances and then click **OK**

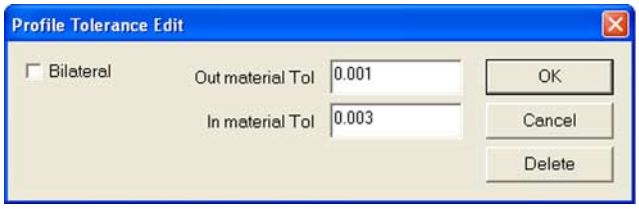
Apply equal bilateral tolerances

Unequal Tolerances

Unequal or unilateral tolerance values are specified by the user for In-material and Out-material tolerances.

To apply an unequal tolerance:

- Uncheck the **Bilateral** check box, enter the desired **In material Tol** and **Out material Tol** tolerance values in the fields provided and then click **OK**



Apply unequal tolerances

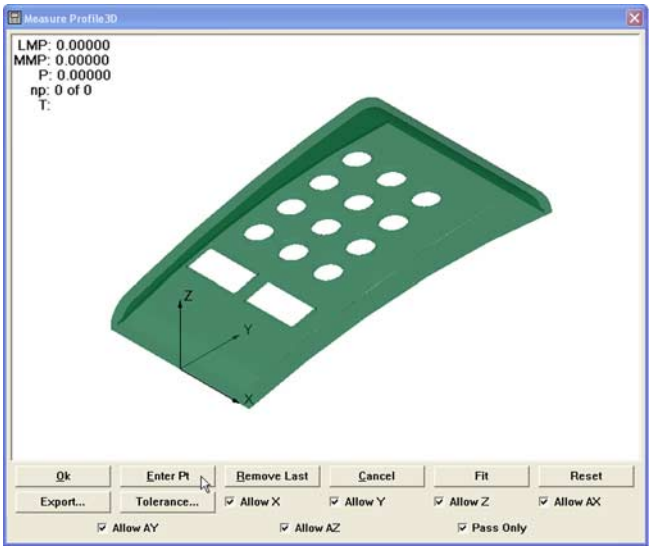
Probe Part Surfaces

As part surfaces are probed and points are collected, they appear in green in the Measure Profile3D window.

When Auto Enter is enabled points will be entered automatically as they are probed.

To enter a probed point manually:

- Click **Enter Pt** in the **Measure Profile3D** window



Enter points from the Measure Profile3D window

To remove the last point(s) collected:

- Click **Remove Last**

To cancel the Profile3D measurement:

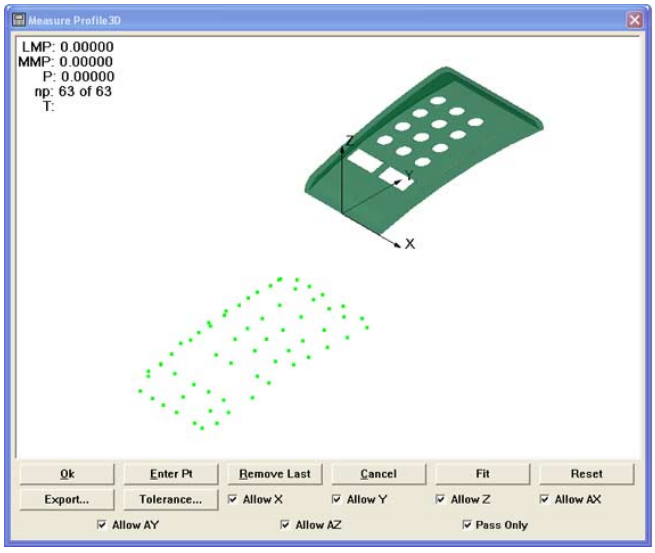
- Click **Cancel**

If a point was probed in a non-perpendicular direction to the surface during program recording, the system will probe that point orthogonal to the surface during program playback as long as the fit analysis was done and completed.

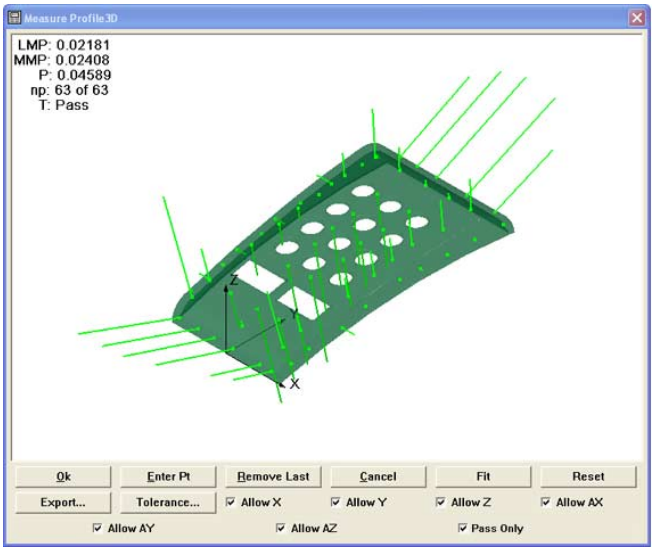
Perform Profile3D Fit Analysis

When all the required points have been probed, the points can be fit to the nominal part model.

When the fit operation is complete, the green probed points will be shifted to match the part model as closely as possible and form error whiskers will indicate differences between the probed locations and the nominal part surfaces.



Points shown before fit calculation



Form error whiskers shown after fit operation

To perform a fit analysis:

- Click **Fit**

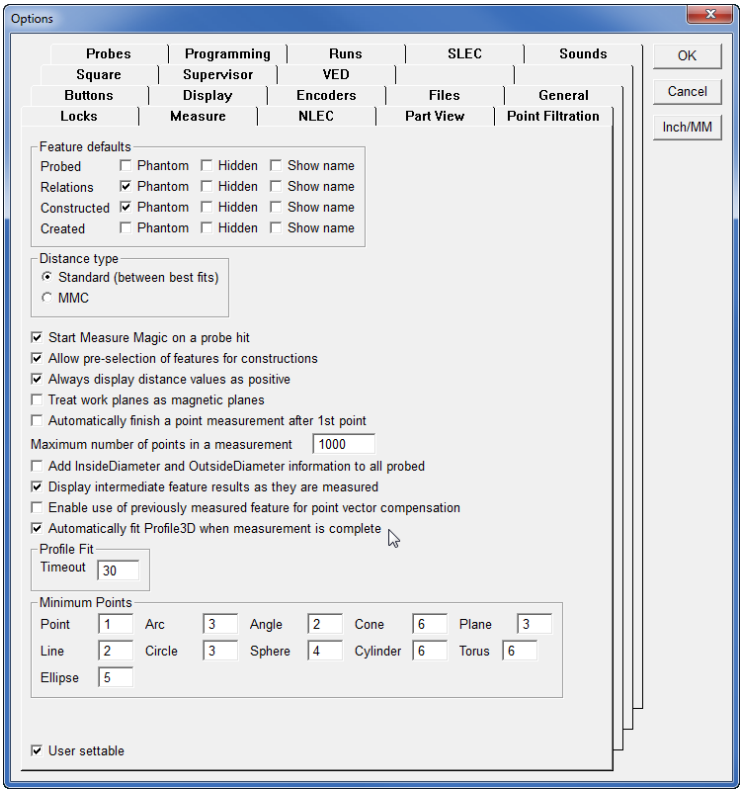
To decrease the fit analysis time, well distributed points all around the part under measure are recommended. In addition, part alignment and datum can be performed before commencing the Profile3D measurement or using the Six Point alignment function in the Profile3D measurement.

Profile3D Automatic Fit Analysis

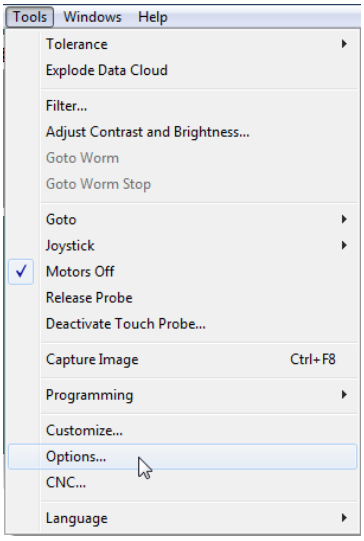
The fit analysis can be set to be performed automatically once the measurement is completed by clicking the **OK** button.

To select the Profile3D automatic fit analysis option:

- Select **Tools>Options...**
- Select the **Measure** tab
- Select the **Automatically fit Profile3D when measurement complete** checkbox



- Click **OK**



Select Tools>Options

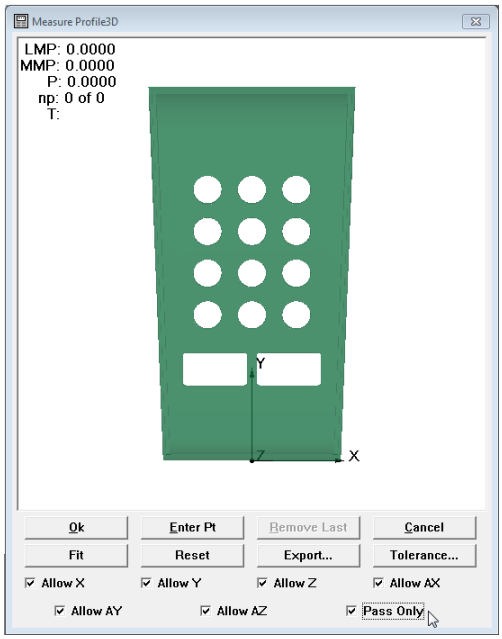
Profile3D Pass Only Option

The **Pass Only** option on the **Profile3D Measure** window specifies that the fitting algorithm terminates the fitting process when it finds a fit that is within all specified surface tolerances. If the **Pass Only** option is checked, and a tolerance has been specified for every surface on which points have been probed, then the fitting algorithm will terminate as soon as it finds a fit in which all probed points pass their tolerance. If no such fit can be found then the fitting algorithm will continue the process of fitting until it either finds the best fit it can or none is found.

The **Pass Only** option is used to improve performance in cases where time is a more important factor than accuracy. This is to be used when a tolerance pass/fail is all that is required and not the exact amount of the error. The amount of performance improvement varies depending on how loose the tolerance is, and how close the part is to its nominal form. The fitting of a part that fails its tolerance will not have any performance improvement by using the **Pass Only** option.

To select the **Pass Only** option:

- Select **Measure>Profile3D**
- Select the **Pass Only** checkbox
- Click **OK**



Select the Pass Only checkbox

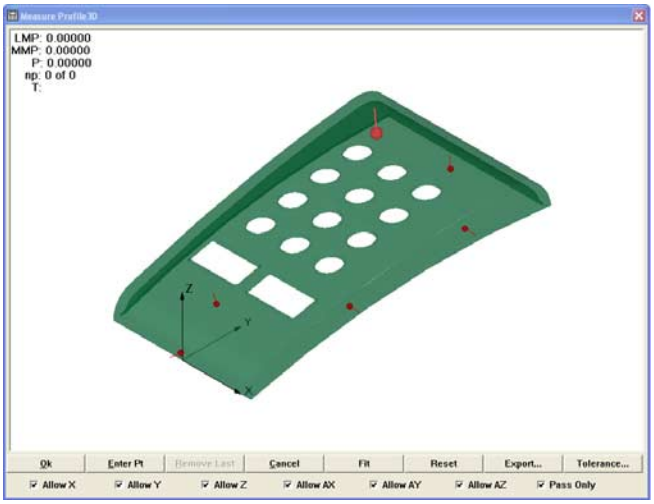
Six Point Alignment

A six point alignment allows the operator to align the model with the orientation of the part on the stage.

To perform a six point alignment:

- Right click in the **Measure Profile3D** window
- Select **Alignment>Six Point**. A prompt appears with directions for marking points on the model.
- Select **OK** in the prompt
- Mark three points on the model surface to define a primary plane
- Mark two points on the model on a surface that is close to perpendicular to the first surface to define a secondary line
- Mark one point on the model on a surface that is close to perpendicular to both the first and second surfaces to define a zero point

When six points are selected the first point is highlighted on the model



Six points selected and first point highlighted

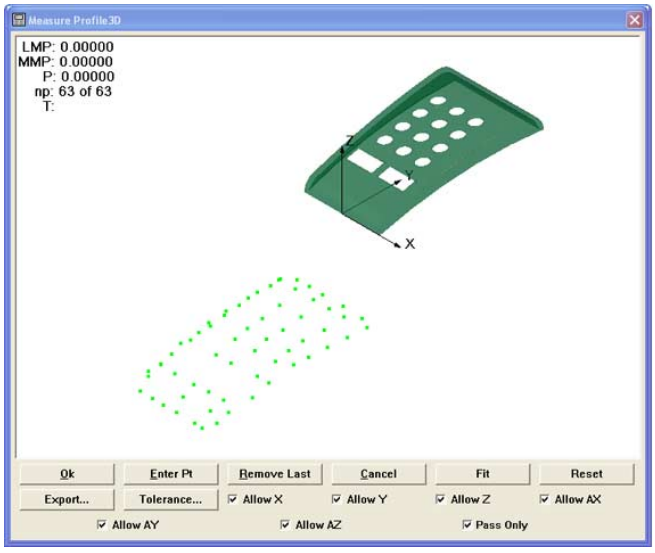
- Probe the first point on the part. The next point is highlighted on the model.
- Repeat probing the remaining five points
- Select **Fit** in the **Measure Profile3D** window. The six point alignment is completed and the initial fit is done.

Resetting the Fit Analysis

The results of a fit analysis can be reset and the point data refined in preparation for revised fit operations. The user can review the positions of data points, remove data points or change the degrees of freedom for the fit analysis.

To reset the fit analysis back to the original probed point locations:

- Click **Reset**



Points shown after resetting the fit analysis

Adjust the Display Magnification of Error Whiskers

During the fit analysis, probed points that do not fall precisely on the nominal profile generate form errors. When tolerances are specified, the errors are divided into two groups, those that pass (Passed) and those that fail (Failed) tolerance criteria.

The passed and failed form errors can be displayed as whiskers that extend from the nominal surface. The length of each whisker is proportional to its magnitude of error.

Typically, the magnitude of most form errors is so small in proportion to the part displayed that without magnification they would not be visible. The **Measure Profile3D** window menu and toolbar include fixed, automatic and user-definable levels of magnification for displaying error whiskers.

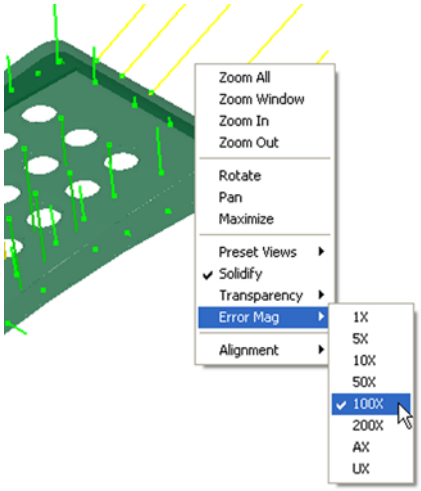
Fixed magnifications: display magnifications are selected directly by the user.

Automatic magnification (AX): the system optimizes the display of error whiskers based on the maximum form error value.

User-defined magnification (UX): A data field will be displayed for specifying the level of magnification.

To adjust the display magnification of form error whiskers:

- Right-click the **Measure Profile3D window** to display the **Measure Profile3D** window menu
- Click **Error Mag** to display a menu of magnification alternatives
- Click the desired magnification selection



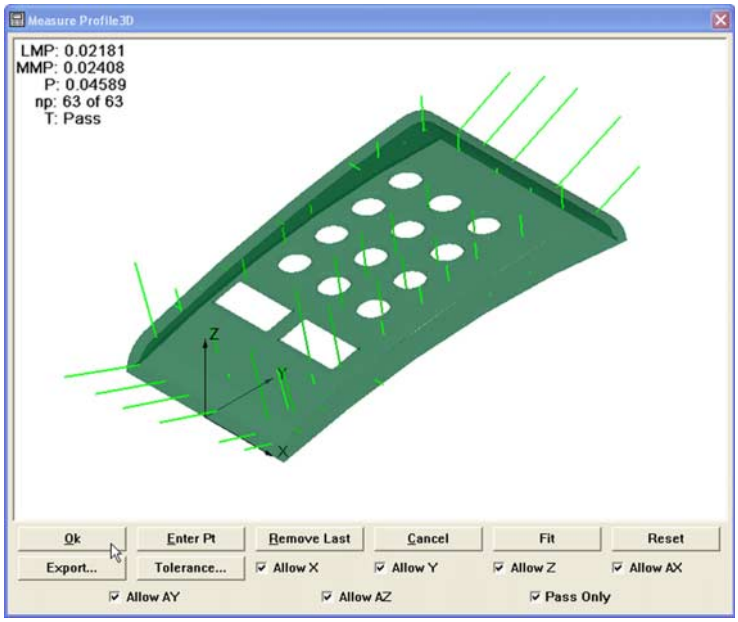
Select error magnification from Measure Profile3D window menu

Add Profile3D Feature to the Feature Template

When the final fit analysis is finished, the Profile3D measurement can be completed and the Profile3D feature can be added to the Feature template.

To complete the Profile3D measurement:

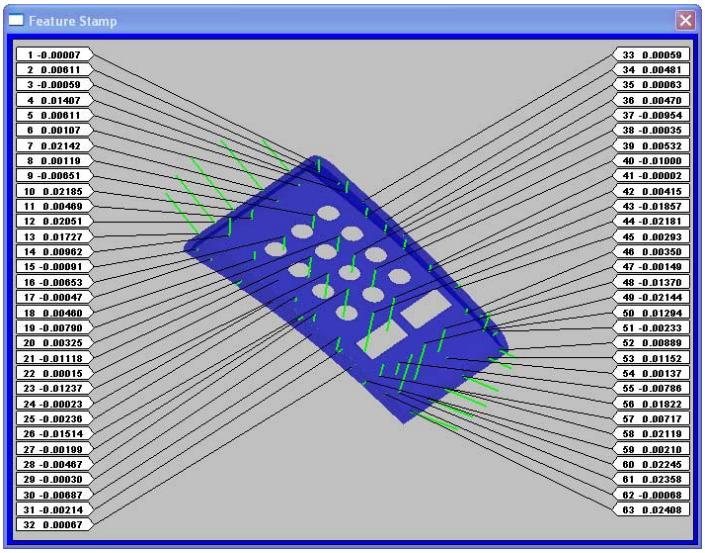
- Click **OK** in the **Measure Profile3D** window



Complete the Profile3D measurement

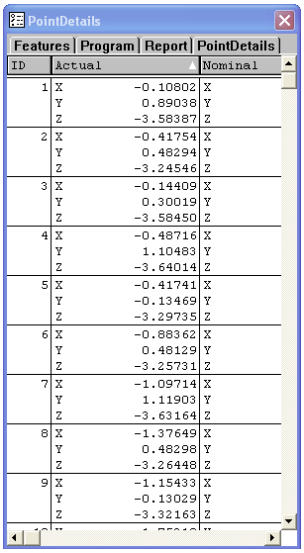
To view the Profile3D measurement results:

- Click on the **Feature Stamp Icon** in the results window



Profile3D feature stamp

Measurements results are also available in the **PointDetails** template.



PointDetails template

If the **PointDetails** template is not open it can be viewed by clicking **Windows>Open Template...** and selecting **PointDetails.5ft**.

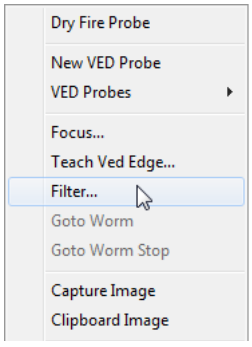
Auto Focus (option)

The Auto Focus teach function has been removed from the IK 5000 software. The focus search distance is now entered directly into the search distance field. The previous Auto Focus algorithm used the teach function to teach the optimum search distance and Z axis velocity, depending on the connected camera's frame rate. This could cause an intermittent Auto Focus failure when used on a significantly different surface from the surface used during the teach process. The algorithm now automatically adjusts the Z axis velocity depending on the camera frame rate and the user specified search distance.

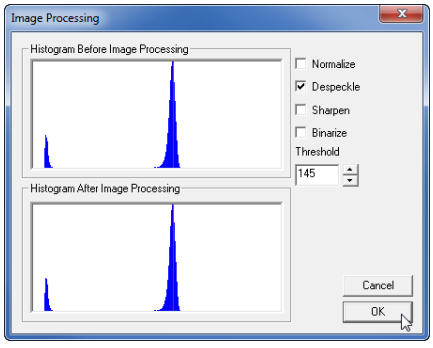
High resolution cameras may produce less than optimal results which can be improved by decreasing the search distance. Decreasing search distance also speeds up the focal point acquisition time but requires initial position to be closer to the targeted focal point. On some cameras turning on the despeckle filter prior to performing auto focus will improve results, despeckle filter can then be turned off after focus.

To turn on the despeckle filter:

- Right click on the **Video** window and select **Filter**...
- Check the **Despeckle** checkbox
- Click **OK**



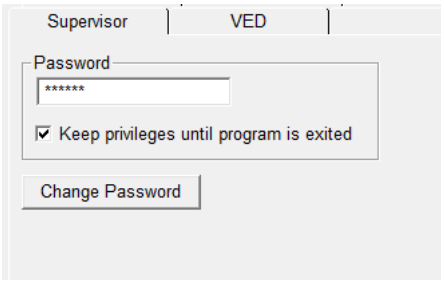
Right click the Video window and select Filter...



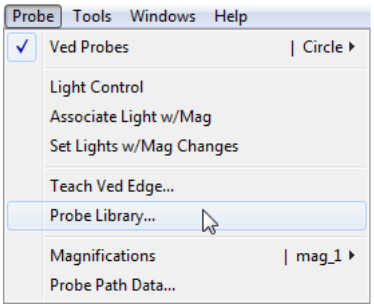
Check the Despeckle box and click OK

To set Auto Focus options:

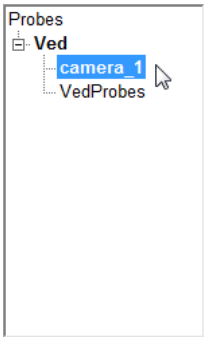
- Click **Tools>Options**
- Verify the Supervisor **Password** is entered and the **Keep privileges until program is exited** option is checked
- Click **Probe>Probe Library**
- Click on the target camera



Verify the Password is entered and Keep privileges until program is exited is checked

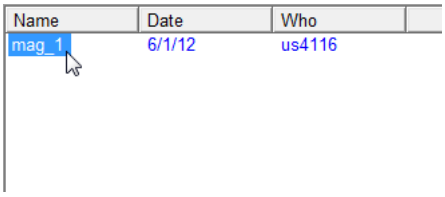


Click Probe>Probe Library

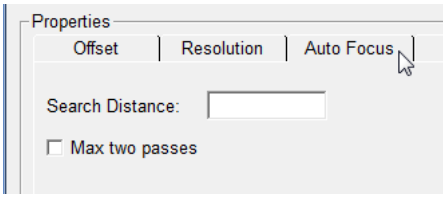


Click on the target camera

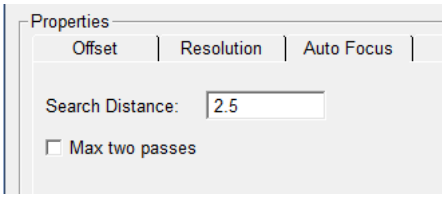
- Click on the target magnification
- Click the **Auto Focus** tab
- Enter the desired search distance in the **Search Distance** field
- Click **OK** to save the changes and exit the **Probe Library**



Click on the target magnification



Click the Auto Focus tab

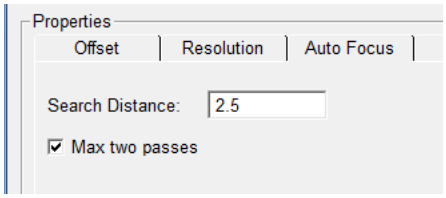


Enter the desired search distance and click OK

Auto focus will search and make as many passes as necessary until a optimum focus point is found. The **Max two passes** feature limits the number of auto focus passes to two. Selecting this feature can speed up the auto focus process but may result in less reliable repeatability of the auto focus height.

To enable Max two passes:

- Select the **Max two passes** check box



Select the Max two passes check box

Autofocus Final Check

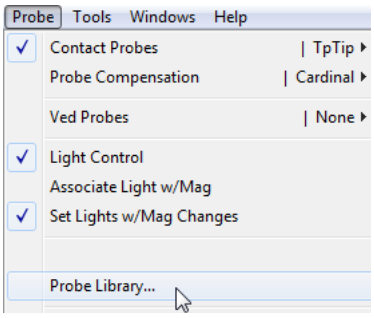
An option to perform a final autofocus check is available from the Auto Focus tab in the Probe Library.

Enabling a final check helps to avoid missing an optimum focus point when autofocusing with a search distance smaller than 1 mm and a magnification greater than 20 x. An optimum focus point can be missed due to skipping of the focus frames during the focus search, machine drive mechanics, changes in lighting, or the objective lens being used.

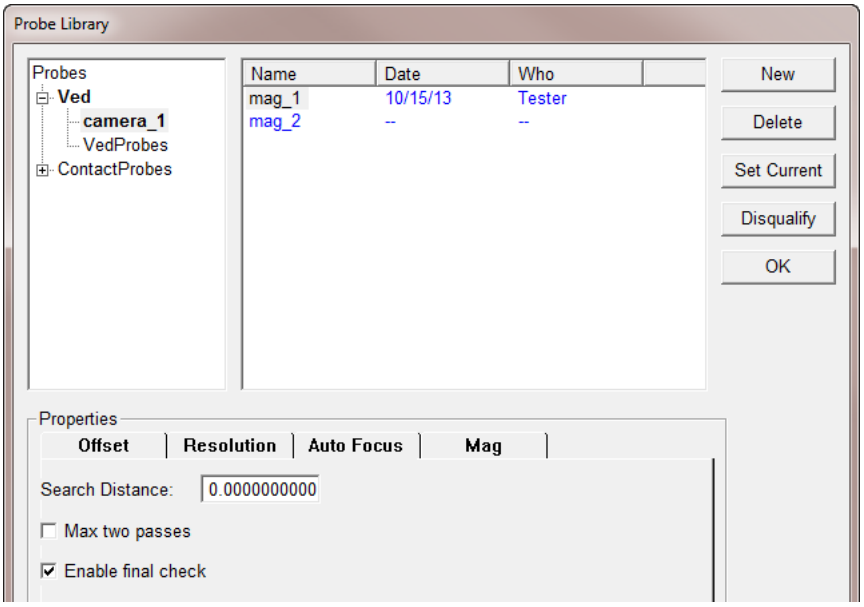
A final check compares the difference in contrast values of frames in the range of 5 µm above to 5 µm below the initial focus point. If there is a point with a larger contrast value found the system will focus on the point of greater contrast.

To enable the autofocus final check:

- Select **`Probe>Probe Library...`**
- Select the target camera and target magnification
- Select the **`Autofocus`** tab
- Check the check box next to **`Enable final check`**



Click Probe>Probe Library...



Check the check box next to Enable final check

Manual VED Height Point Measurement

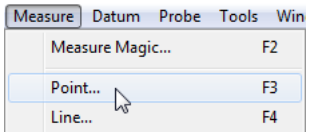
The VED height probe can be used to measure height points in a VED configured system. This is done automatically in a motorized stage system. In a manual system this function must be performed manually using the VED height probe.

To perform VED height point measurement manually:

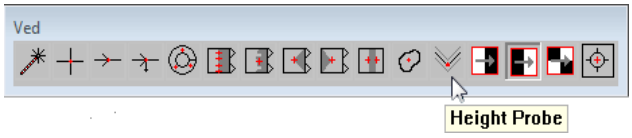
- Click **Measure>Point...**

The **Measure Point** window opens.

- Select the VED Height probe

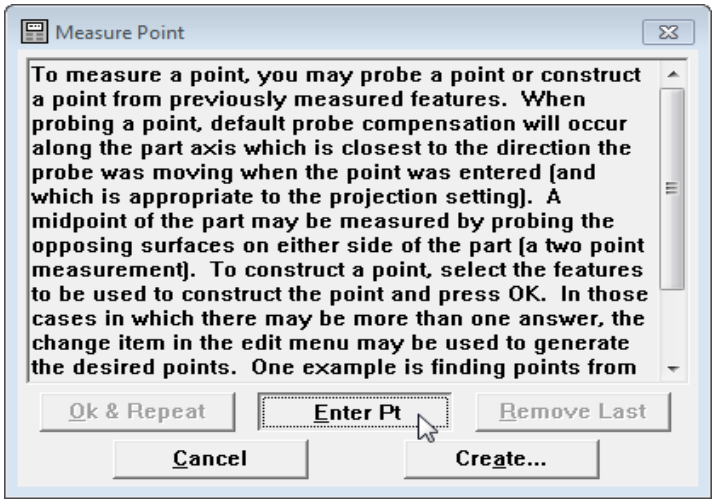


Click **Measure>Point...**



Select the VED Height probe

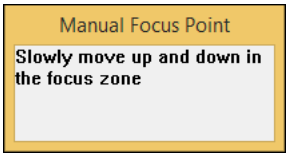
- Press **Enter Pt** in the Measure Point window



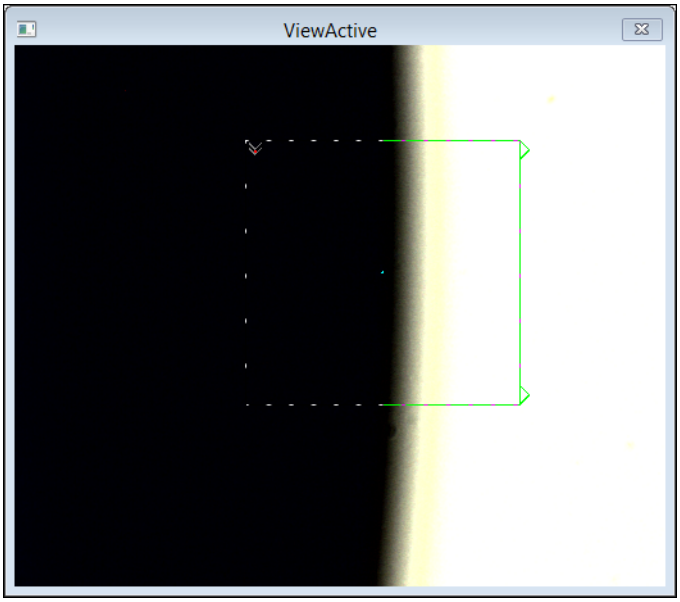
Press **ENTER PT**

- Follow the prompt on the screen to slowly move up and down in the focus zone

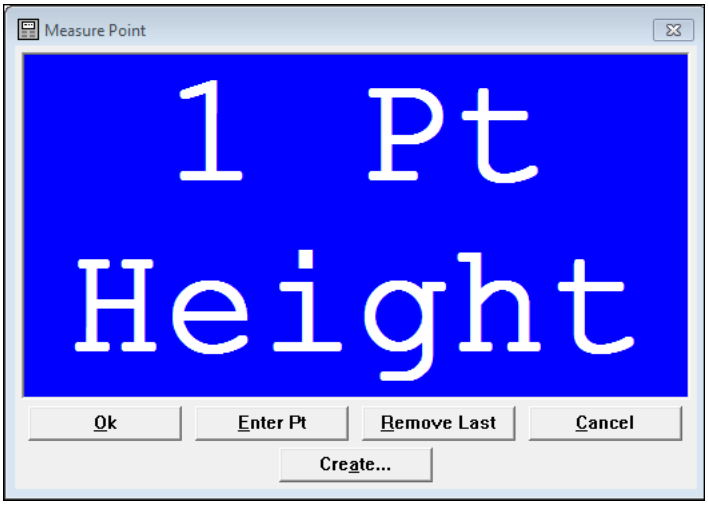
Once the system finds focus, it will fire the probe and take a point measurement which will be posted in the Measure Point window.



Follow the prompt



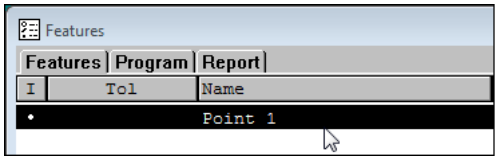
Slowly move up and down in the focus zone



The system will fire the probe and take a point measurement

- Press **OK** in the Measure Point window. The Point height measurement is complete.

Press **OK** in the Measure Point window



The Point height measurement is complete

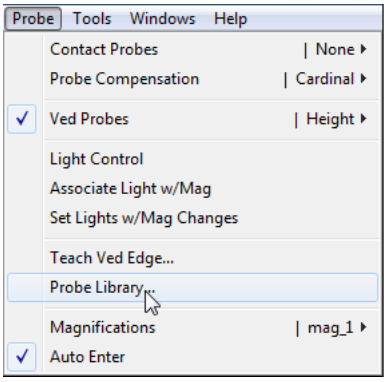
Optimet Laser Lens Selection

The Optimet laser is a laser measuring probe supported by IK 5000. Multiple laser lenses are available for probing different types of surfaces and materials. Each lens has an ID.

The **Lens** settings in the IK 5000 software must match the physical lens that is attached to the probe. These settings allow the IK 5000 to adjust the laser beam properties to match the physical lens. When attaching a physical lens to the probe, verify that the settings match the physical lens.

To verify the Optimet laser lens:

- Click **Probe>Probe Library...**



Click **Probe>Probe Library...**

- Select the **Optimet** probe
- Select the ID for the physical lens attached to the probe from the **Index** drop-down list
- Verify that the **Focal Distance** setting matches the focal distance of the physical lens

Verify that the **Focal Distance** setting matches the focal distance label on the physical lens

- Click **OK**.

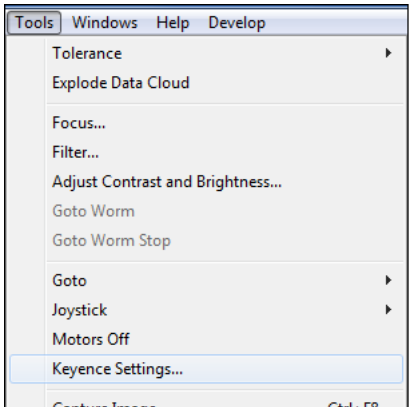
Keyence Laser Settings

The Keyence LK-G5000 Series laser is a laser measuring probe supported by IK 5000. The Keyence laser can be used to measure features that require a laser probe for acquiring a feature data cloud.

How the Keyence laser acquires data points can be controlled by changing settings in the IK 5000 software.

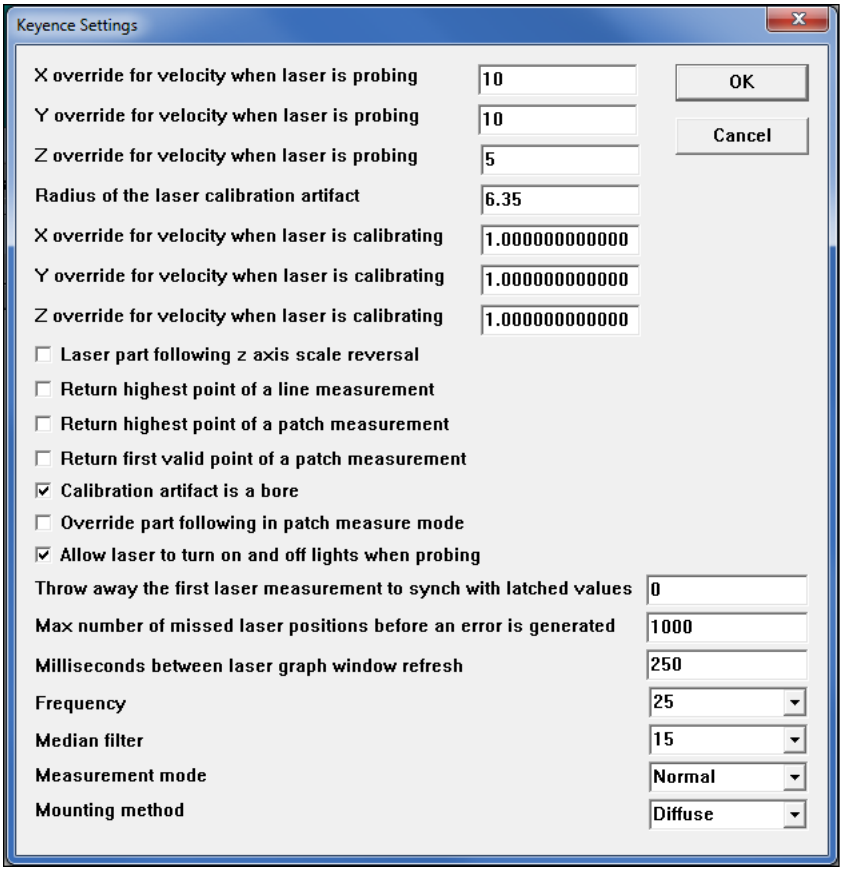
To edit Keyence laser settings:

- Click **`Tools>Keyence Settings...`**



Click **`Tools>Keyence Settings...`**

▶ Edit any settings that require changes



Edit any settings that require changes

▶ Click **OK**

The changes are saved and the **Keyence Settings** window closes.

Refer to the documentation provided with the Keyence LK-G5000 for additional information regarding the **Frequency**, **Median filter**, **Measurement mode** and **Mounting method** settings.

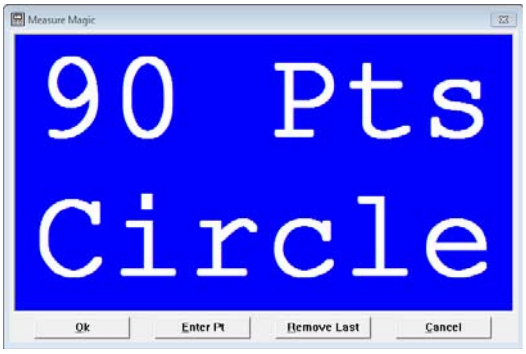
Exploding the Data Cloud

Explode Data Cloud is used to extract point features from the data cloud used in a probed feature.

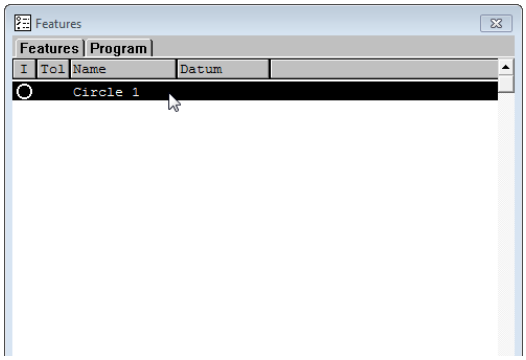
To extract point features from the data cloud:

- Verify the system is in **No Program** or **Editing** state of programming
- Probe a feature
- Select the target feature

Verify the system is in No Program or Editing state



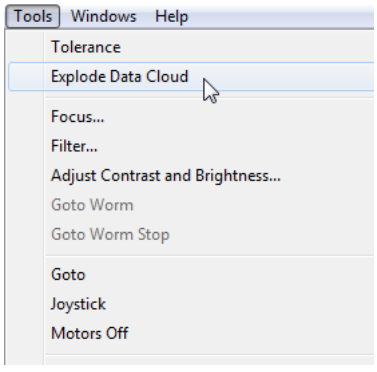
Probe a feature



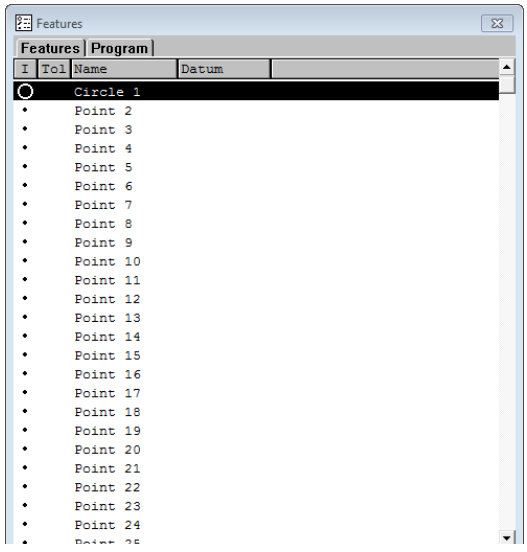
Select the target feature

- Click **Tools>Explode Data Cloud**.

The **Features** List is populated with point features created from the data cloud of the selected feature.



Click Tools>Explode Data Cloud



Point features are created from the data cloud

Rotating and Positioning of VED Tools using Part Reference Frame

If a part datum has been performed, the system will rotate and position the VED tools, including the Video Charts, to the part reference frame.

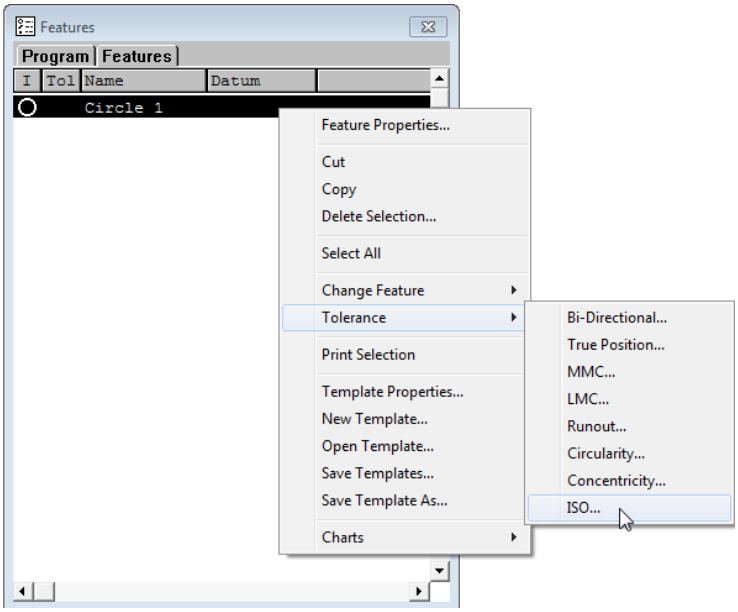
ISO Tolerance

The ISO tolerance enables the use of data stored in ISO 286-2 Standard tables.

The ISO tolerance can be added to the following features:

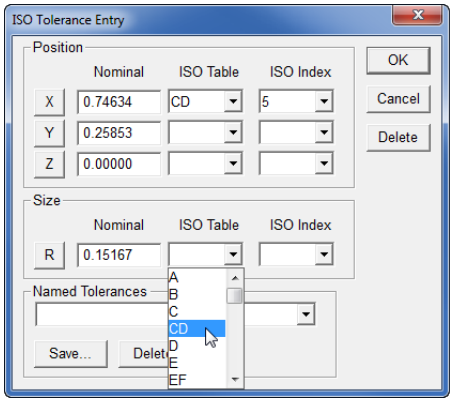
- Point
- Circle
- Arc
- Sphere
- Line
- Slot
- Torus
- Rectangle

The ISO tolerance is applied to a feature similar to other tolerances, for instance by right-clicking on a feature in the **Features** list template.



Right-click on a feature in the Feature list

Once the ISO tolerance is selected the **ISO Tolerance Entry** dialog box appears. The **ISO Tolerance Entry** dialog box is used to select which ISO tolerance table entry will be used to tolerance each applicable feature parameter.



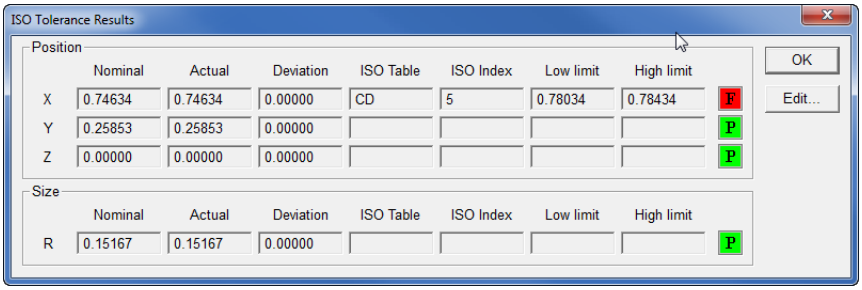
ISO tolerance entry table

Available parameter selections are:

- ISO table
- ISO Index

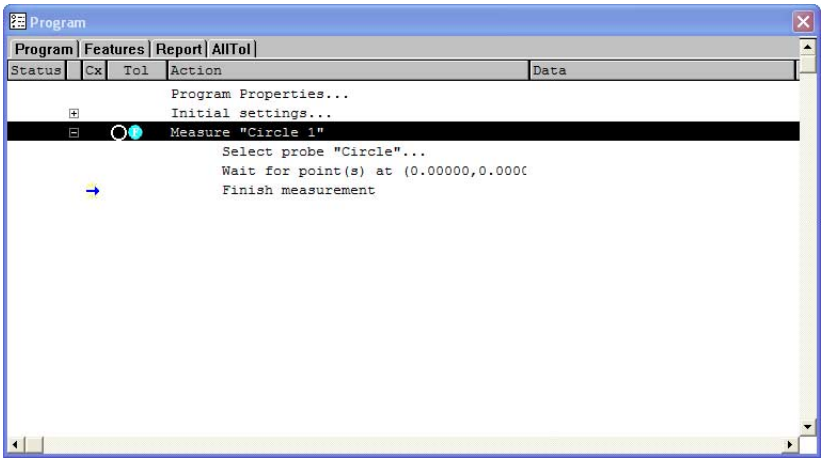
The ISO index entry can be used to select which column, 1 through 18, to use for the selected ISO tolerance table. For questions about the ISO tolerance table selection please refer to the ISO 286-2 standard.

The nominal value for the feature parameter along with the selected ISO tolerance table and index entries are used to determine the low and high limits to be used when tolerancing the feature. These limits are calculated through a look-up into the ISO tolerance table. If no ISO tolerance table entry or ISO tolerance index entry is specified then the tolerance is considered empty and the tolerance will pass. Also, if the nominal value for the feature parameter is outside of the range of nominal values found within the selected tolerance table, then the tolerance will also pass.



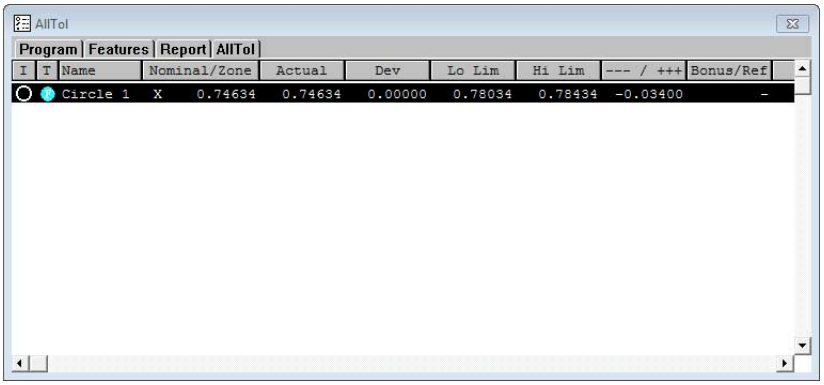
ISO tolerance results

Similar to all tolerances, the ISO tolerance can be added to features within a program.



ISO tolerance added to a feature in a program

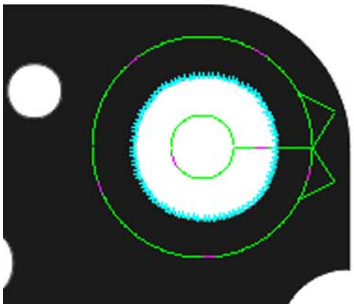
The ISO tolerance results can also be viewed in reports when using the **`AllTol`** template.



AllTol template

Continuous Probe Firing

Probes can be dry-fired continuously to show data points that would be acquired under the current light, focus, part alignment and other current measurement conditions if the probe were actually fired. The data points are shown in blue until the probe is fired to acquire points, then the acquired points are shown momentarily in red.



Continuous Fire enabled

Continuous probe firing is useful for setup and troubleshooting activities, but requires substantial system resources and should be avoided during routine use to maintain high throughput.

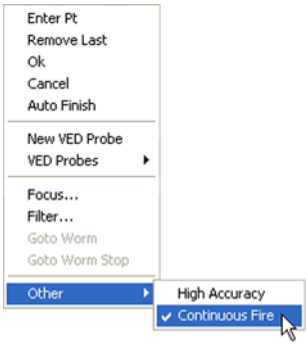
To enable Continuous Fire:

- Right-click the **Live Video** window to display the **Window** menu
- Select **Other>Continuous Fire**

or

- Click the **Continuous Fire** button on the **VED** toolbar

Enable Continuous Fire from VED toolbar



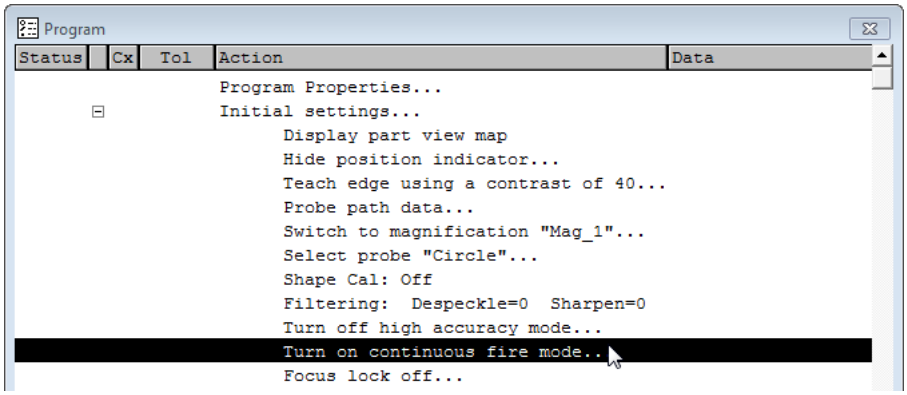
Enable Continuous Fire from menu

Programming Continuous Probe Firing

Continuous probe firing is programmable. A Continuous Fire mode step is recorded in the Initial settings program step when a new program is recorded. Continuous probe firing is turned off in the Initial settings step by default.

After a program is run the state of Continuous Fire returns to the same state that was in place before the program was executed.

Initial settings step



Enable Continuous Fire from Initial settings... step

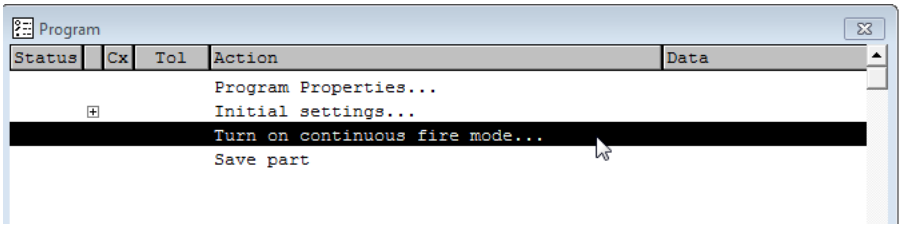
To turn Continuous Fire on in the Initial settings:

- Click the **+** sign next to **`Initial settings...`** to expand the Initial settings step in the program window
- Double-click **`Turn off continuous fire mode...`**

The text changes to **`Turn on continuous fire mode...`**. Continuous Fire mode will now be turned on when the program is run.

Program Step

When the state of Continuous Fire is changed from the Live Video window or the VED toolbar, a separate Continuous Fire step is recorded in the program. The state of this step can be changed.



Change the state of a Continuous Fire program step

To turn Continuous Fire off in an existing program step:

▶ Double-click on **`Turn on continuous fire mode...`**

`Turn off continuous fire mode...` is displayed in the Program window and the program will now turn Continuous Fire off at this step in the program.

To turn Continuous Fire on in an existing program step:

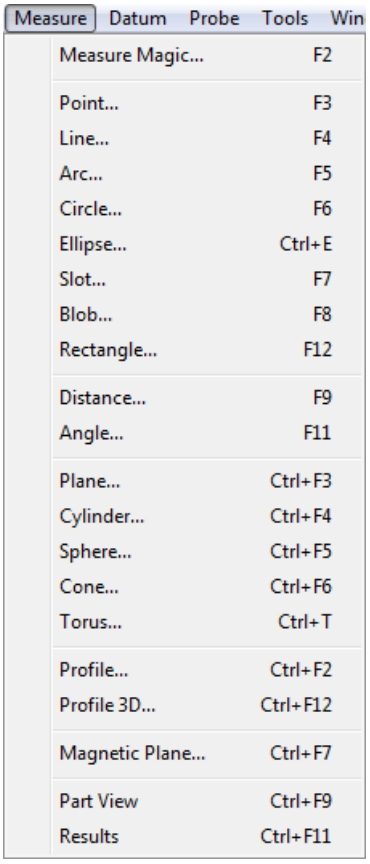
▶ Double-click on a **`Turn off continuous fire mode...`** step

`Turn on continuous fire mode...` is displayed in the Program window and the program will now turn Continuous Fire on at this step in the program.

Measure Shortcuts

Keyboard shortcuts for starting Ellipse, Torus, Profile, and Profile3D measurements are now available. The shortcuts are listed in the Measure menu.

In addition to the new measure shortcuts, the Rectangle menu item has been moved to a more appropriate location in the Measure menu.



Measure menu

Ellipse Shortcut

To start an Ellipse measurement:

- Press **Ctrl+E** on the keyboard

Torus Shortcut

To start a Torus measurement:

- Press **Ctrl+T** on the keyboard

Profile Shortcut

To start a Profile measurement:

- Press **Ctrl+F2** on the keyboard

Profile3D Shortcut

To start a Profile3D measurement:

- Press **Ctrl+F12** on the keyboard

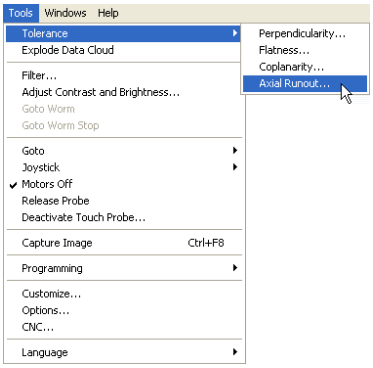
Axial Runout Tolerance Check for Plane Feature

Axial runout tolerance can be applied to a Plane feature that is perpendicular to a reference axis, e.g. plane to cylinder.

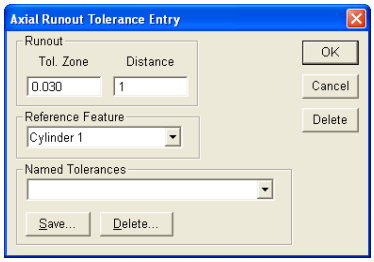
To apply axial runout tolerance:

- Measure a cylinder to be used as a reference in the tolerance
- Measure the plane to apply the axial runout tolerance to
- Select the **Plane** feature in the **Features** template
- Select **Tools>Tolerance>Axial Runout**
- Enter tolerance parameters
- Click **ok**

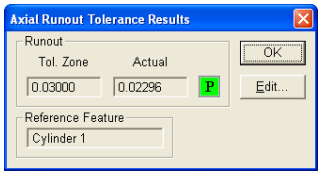
Tolerance results are displayed.



Select Tools>Tolerance>Axial Runout



Enter tolerance parameters and click OK



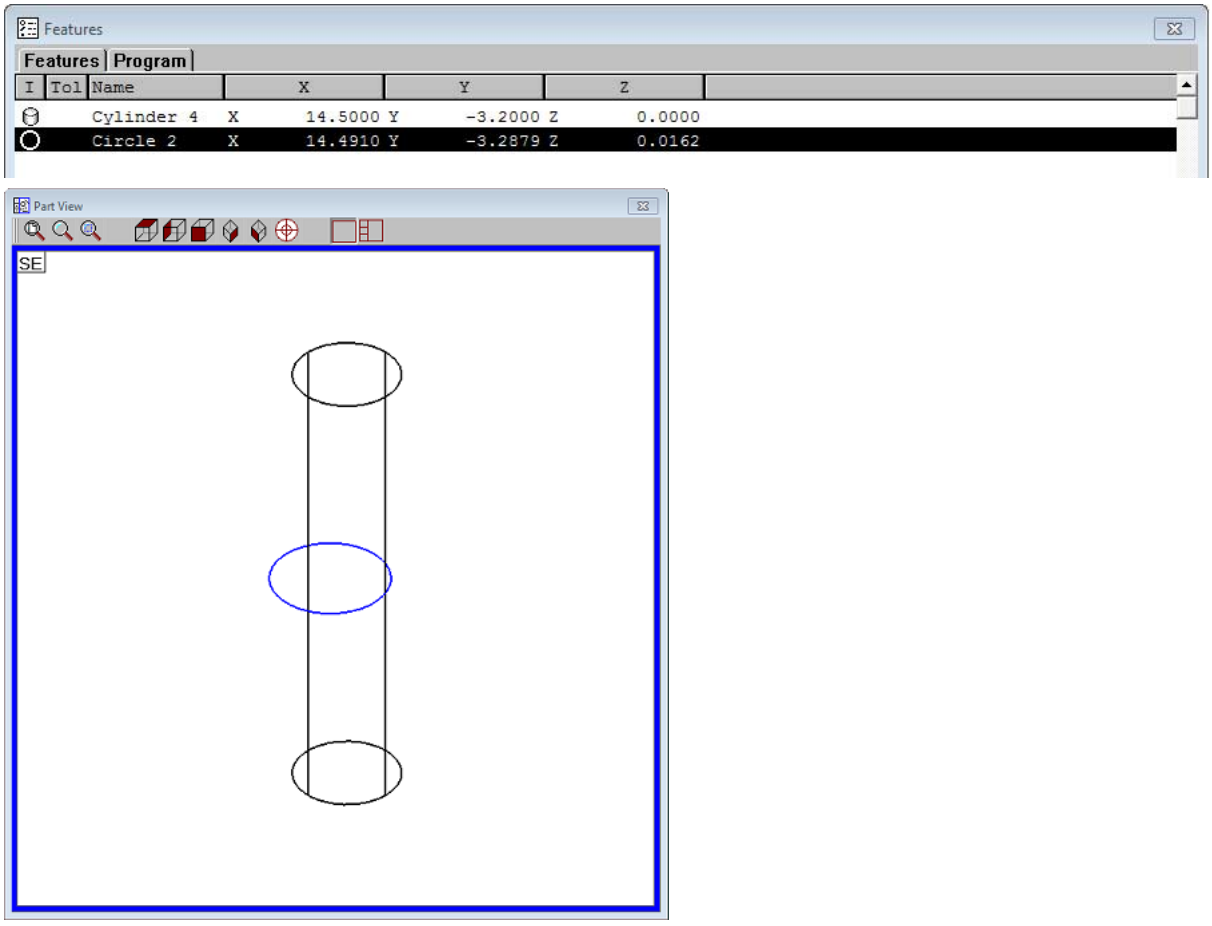
Tolerance results are displayed

Performing a runout tolerance on a circle using a cylinder as a reference

Performing a runout tolerance on a circle requires a reference feature such as another circle or cylinder.

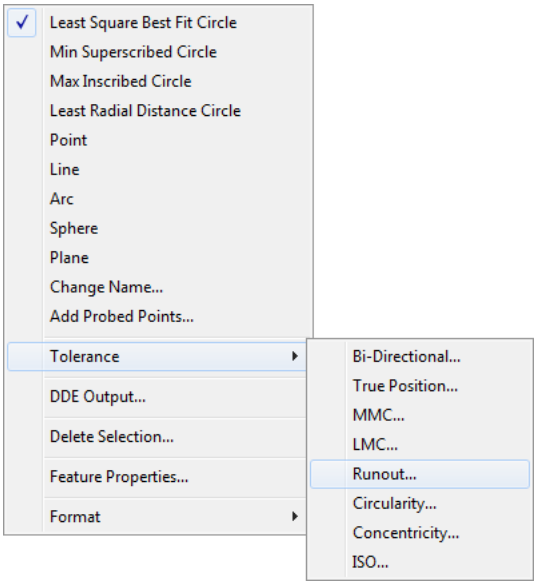
To use a cylinder as a reference feature:

- Measure a cylinder to be used as the reference feature for the runout tolerance and then measure the circle to be toleranced.



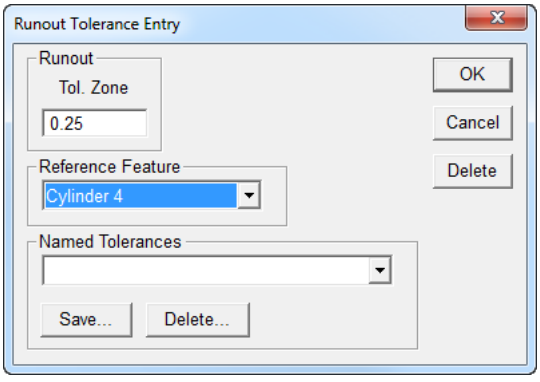
Measure a cylinder to be used as the reference feature and then measure the circle

- Right-click on the Results window
- Select **Tolerance>Runout…**



Right-click and select **Tolerance>Runout…**

- Enter the **Tol. Zone**
- Select the cylinder to use as the reference feature from the **Reference Feature** drop-down list
- Enter a name into the **Named Tolerances** field if desired
- Click **OK** in the Runout Tolerance Entry window to accept the values entered

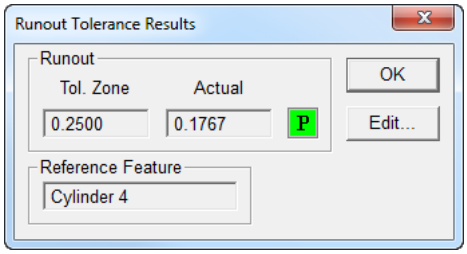


Enter Runout Tolerance Entry information and click **OK**

- Click **OK** in the Runout Tolerance Results window to save changes and close the window

Or

- Click **Edit...** to go back to the entry window

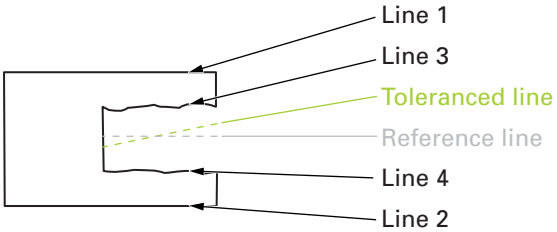


Click **OK** or **Edit...**

Symmetry Tolerance

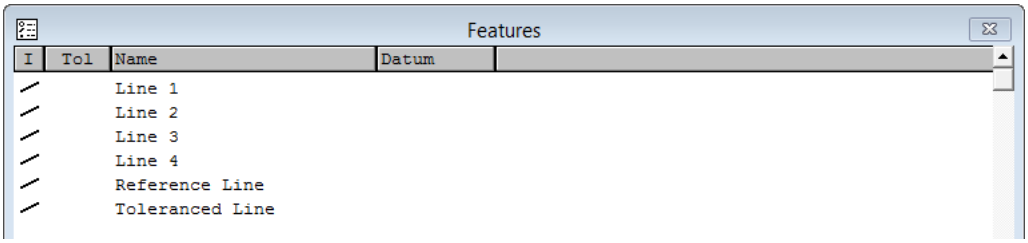
A symmetry tolerance can be applied to Line and Plane features that are symmetrical to a reference feature.

Symmetry Tolerance Applied to a Line

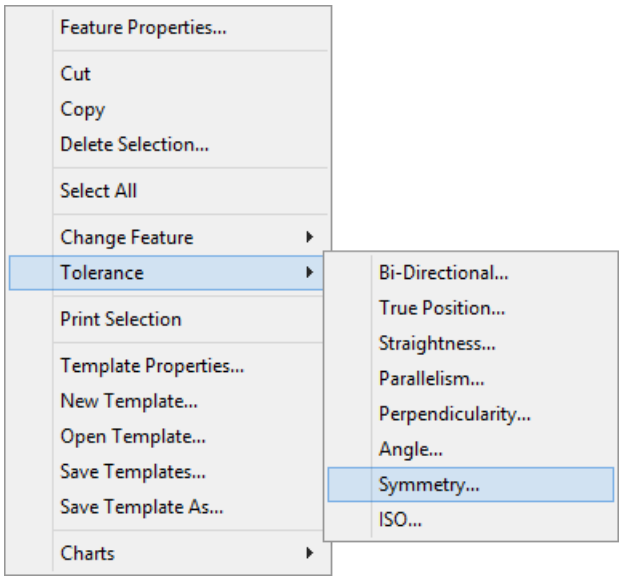


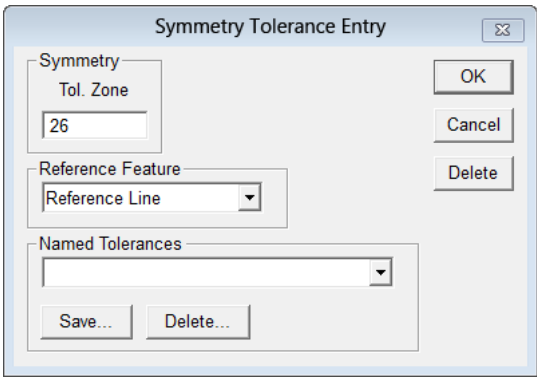
To apply the symmetry tolerance to a line:

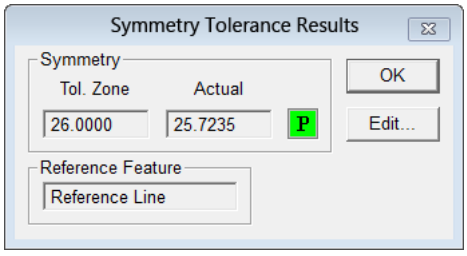
- Measure four lines, two lines on opposite sides that are considered to be master edges and two lines from edges that are to be symmetrically toleranced. Lines 1 and 2 are measured from master edges while lines 3 and 4 are measured from the edges to be symmetrically toleranced.
- Construct a reference line from lines 1 and 2
- Construct the line to be tolerance for symmetry from lines 3 and 4
- Select the resultant line from the construction using lines 3 and 4

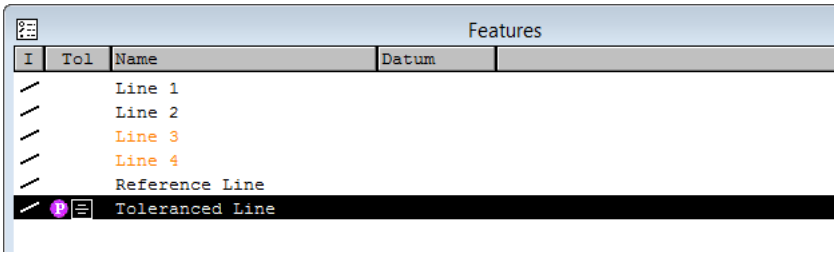


- Apply symmetry tolerance with the constructed line from lines 1 and 2 as the reference feature



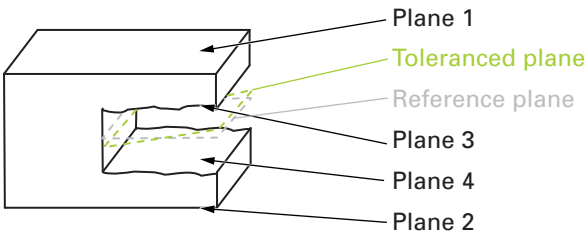






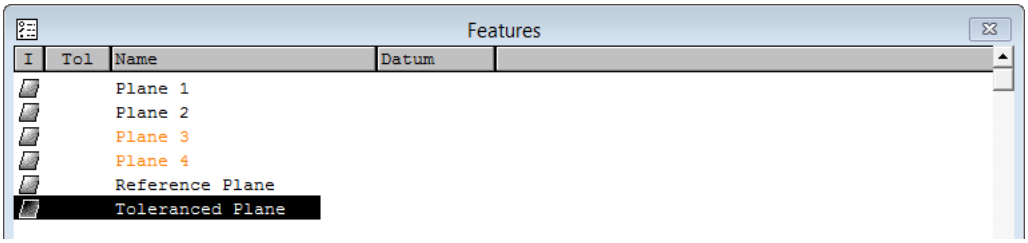
The actual tolerance is calculated by doubling the maximum distance between the endpoints of the constructed tolerance line from the endpoints of the constructed reference line.

Symmetry Tolerance Applied to a Plane

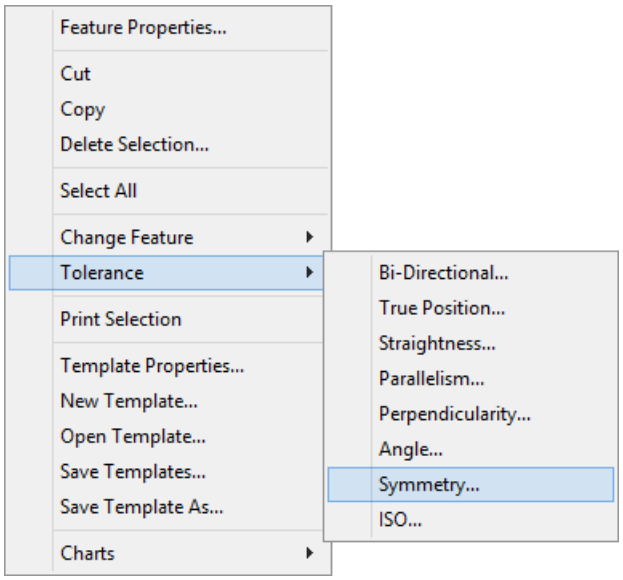


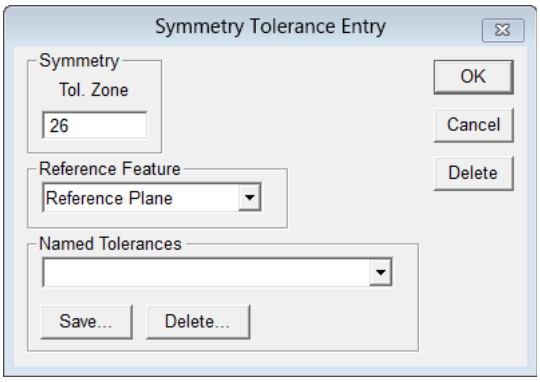
To apply the symmetry tolerance to a plane:

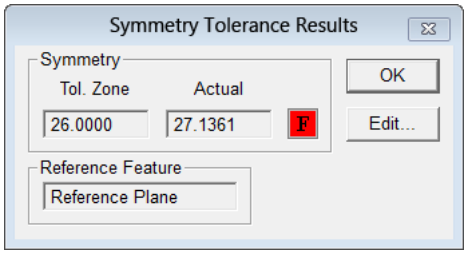
- Measure four planes, two planes on opposite sides that are considered to be master surfaces and two planes from surfaces that are to be symmetrically toleranced (see drawing above). Planes 1 and 2 are measured from master surfaces while planes 3 and 4 are measured from the surfaces to be symmetrically toleranced.
- Construct a reference plane from planes 1 and 2
- Construct the plane to be toleranced for symmetry from planes 3 and 4
- Select the resultant plane from the construction using planes 3 and 4

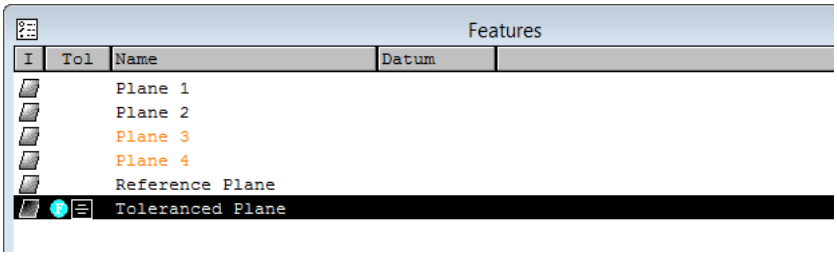


- Apply symmetry tolerance with the constructed plane from planes 1 and 2 as the reference feature





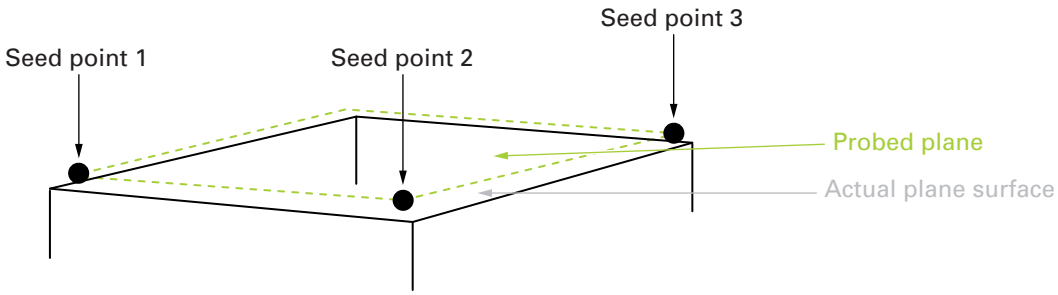




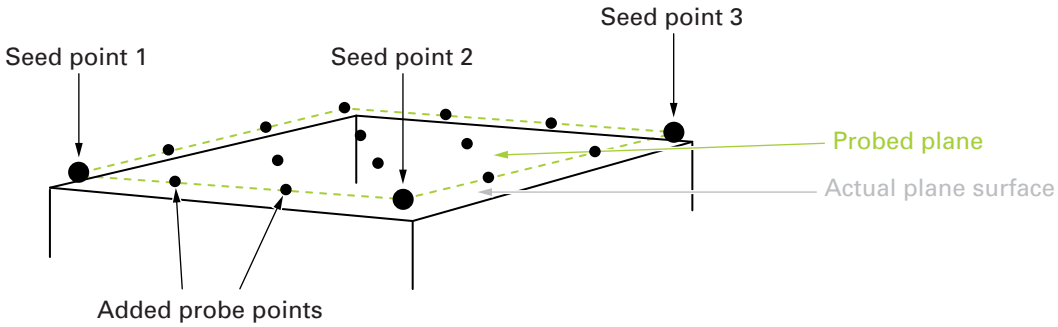
The actual tolerance is calculated by doubling the maximum distance between the endpoints of the constructed tolerance plane from the endpoints of the constructed reference plane.

Plane Probing Technique

A plane can be measured by probing three seed points for the data cloud. The IK 5000 fitting algorithm uses these three points as the first three corners of the plane.

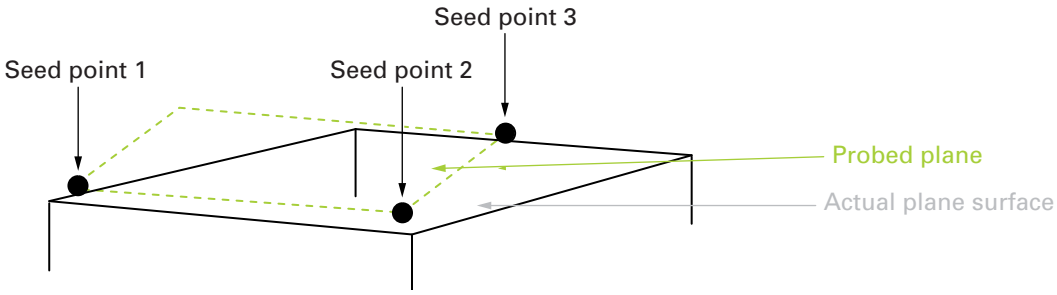


It is important to probe the three seed points on the corner surfaces of the actual plane if the **`Add Probe Points`** function is used to add more points to the data cloud of the plane measurement.

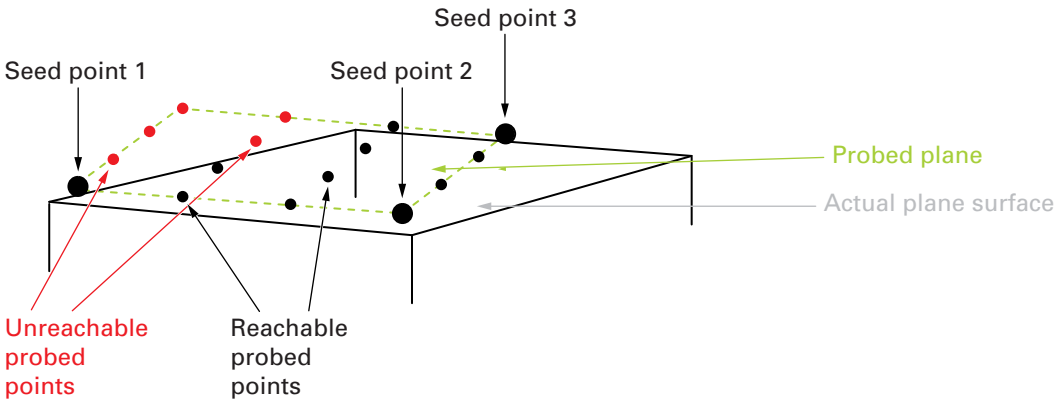


Correct probing technique

In some cases it may be necessary to probe the seed points in a different order and location. For example, probing the seed points where the first two are located at the immediate corners facing the operator while the third point is probed on the middle point of the opposite edge.



The resulting plane cannot be used with the **Add Probe Points** function because it will fail due to some points being located outside the actual surface. The system will miss target points as they will be located outside the actual plane and cannot be reached.



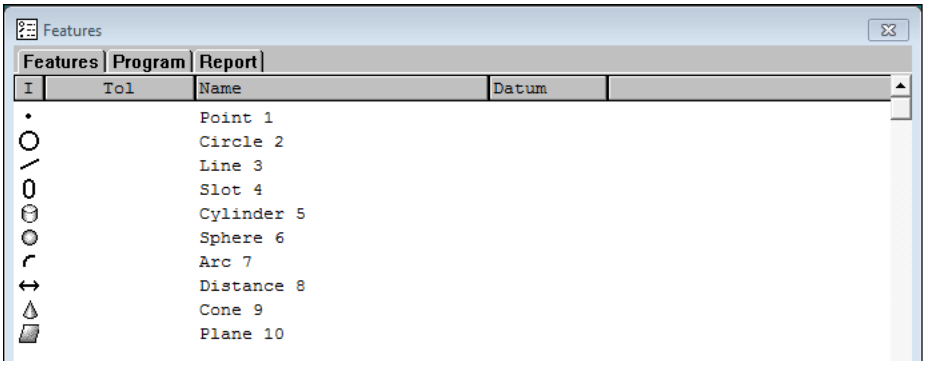
Incorrect probing technique

Exporting 3D CAD Model Features in STEP File Format

IK 5000 supports exporting features to the STEP file format.

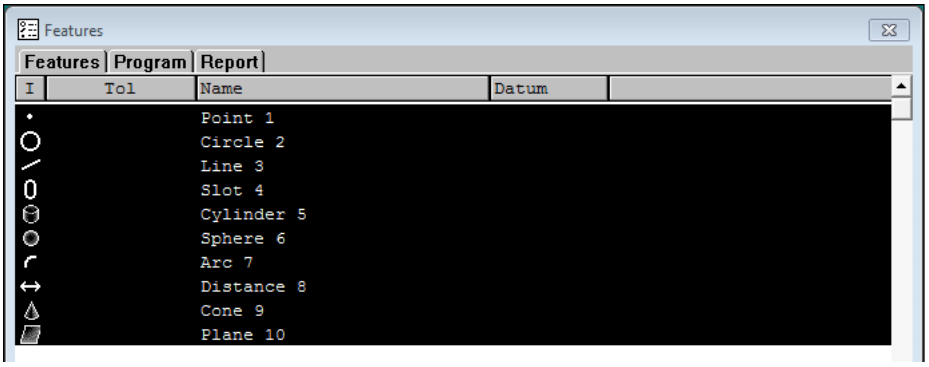
To export features to the STEP file format:

- Open a reporting template (e.g. Feature List template, Reports template, etc)
- Measure a feature or features to export



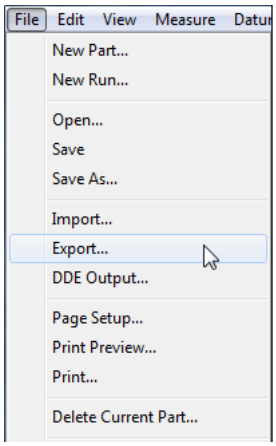
Open a reporting template

- Select the features to be exported

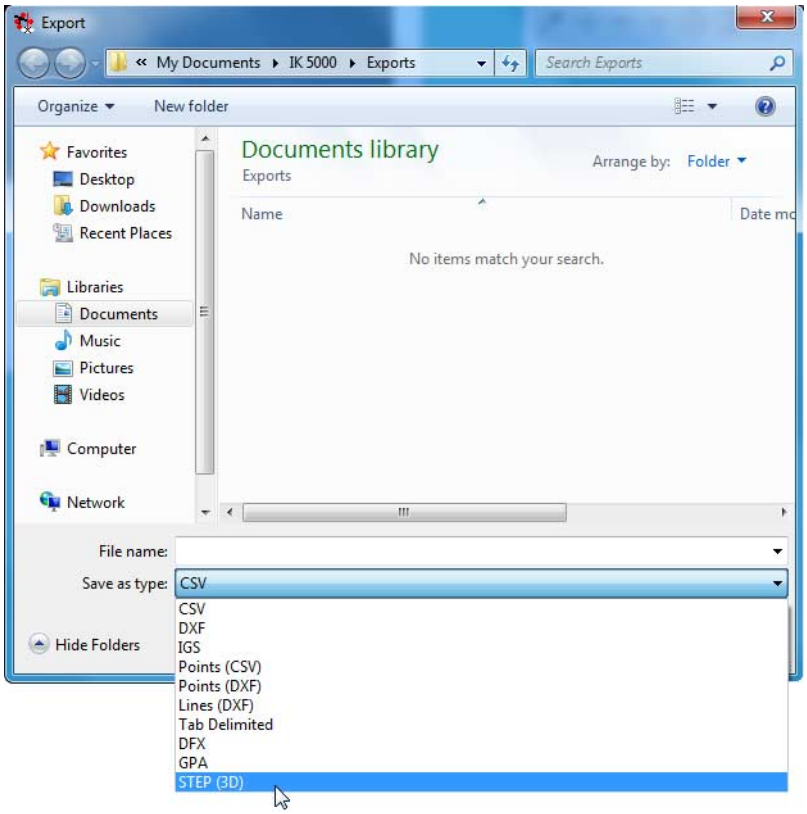


Select the features to be exported

- Click **File>Export...**
- Select **STEP (3D)** from the **Save as type** drop-down list

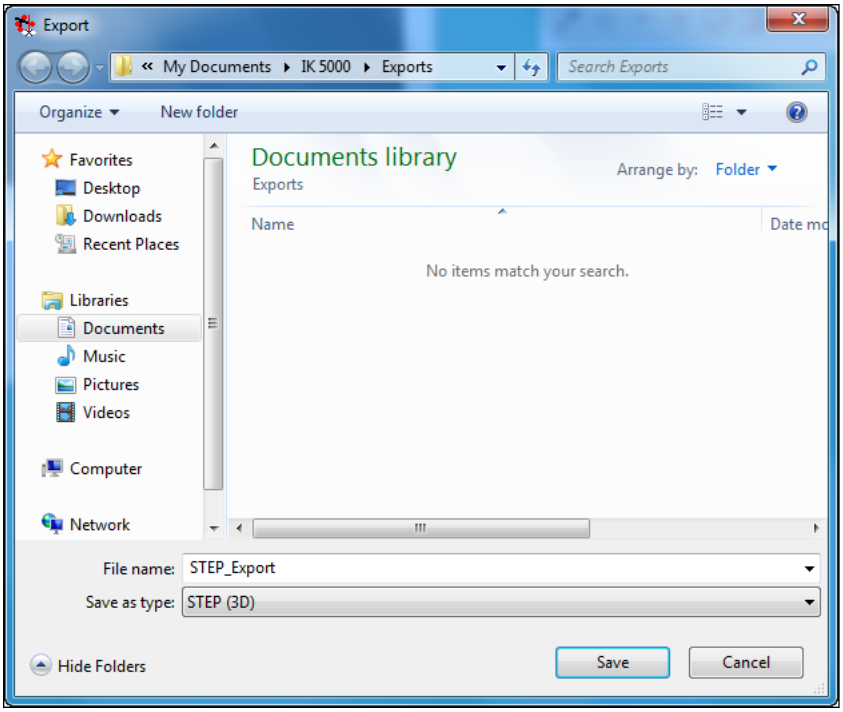


Click **File>Export...**



Select **STEP (3D)**

- Enter a name in the **File name** field
- Click **Save**



Enter a **File name** and click **Save**

The features will be saved to file in STEP file format.

Open Database Connectivity (ODBC) Data Export

The IK 5000 can export data to applications, like Microsoft Excel, using Open Database Connectivity (ODBC).

To send data to a target application:

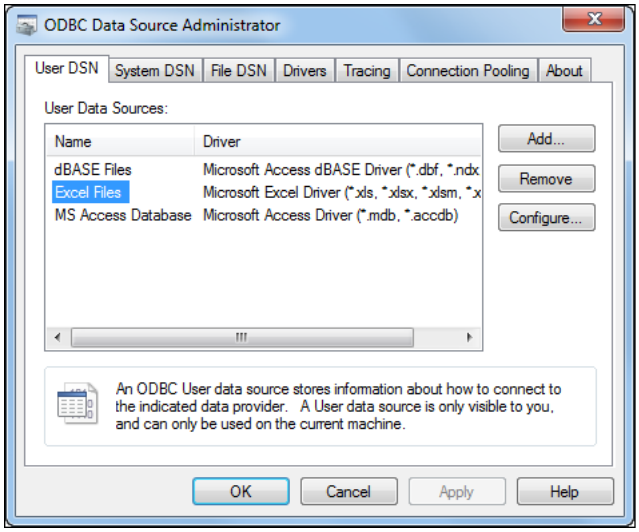
- Create a database in the target application (in the case of Excel, create a file)
- Configure the database as a data source using the operating system's ODBC Data Source administration

Refer to the PC's operating system instructions for how to configure an ODBC data source. An example is provided below.

Microsoft Windows 7 database configuration example

- Open the Windows **`Control Panel`** from the **`Start`** menu
- Select **`Administrative Tools`**
- Select the **`Data Sources`** shortcut

The ODBC Data Source Administrator window opens.



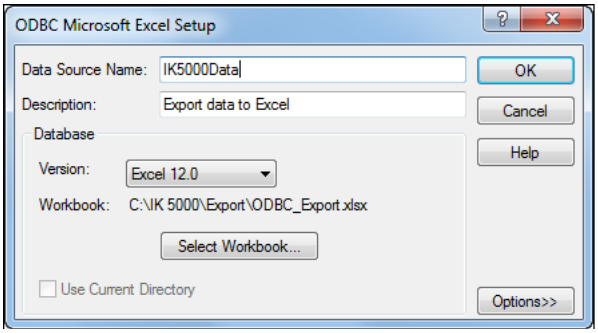
ODBC Data Source Administrator

- Select the **Excel Files** Data Source

If the Excel Files Data Source does not exist, click the **Add** button and add it from the installed ODBC drivers.

- Click **Configure**

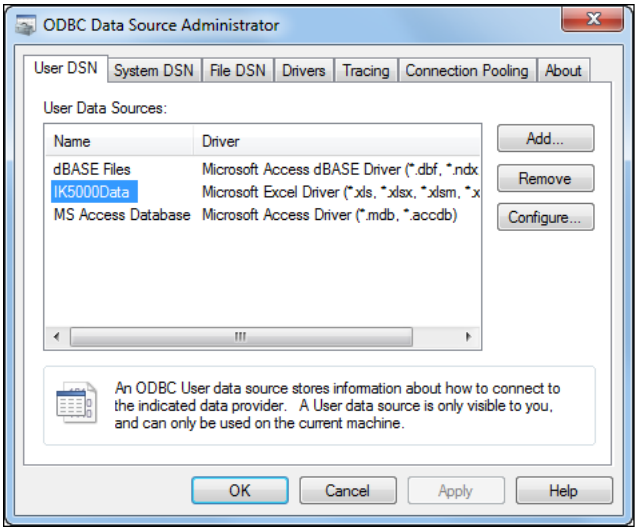
The ODBC Microsoft Excel Setup window opens.



ODBC Microsoft Excel Setup window

- Enter the **Data Source Name** and **Description** into the text fields
- Click the **Select Workbook** button and select the workbook to be used for the database
- Click **OK** to save the data source configuration

The Data Source is configured and the name is updated in the ODBC Data Source Administrator.



ODBC Data Source Administrator

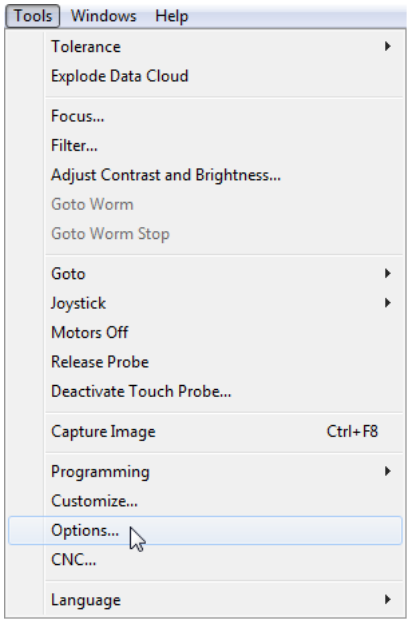
- Close the ODBC Data Source Administrator window
- Close the Excel file if it is open

To enable Prompt for ODBC Export in the IK 5000:

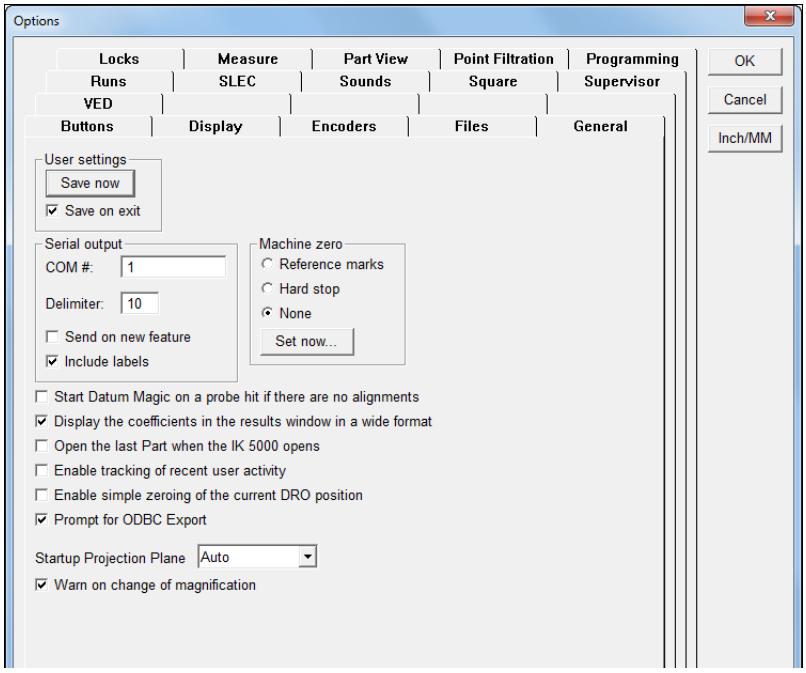
- Select **Tools>Options...** from the IK 5000 menu bar

The Options window opens.

- Select the **General** tab
- Select the **Prompt for ODBC Export** checkbox



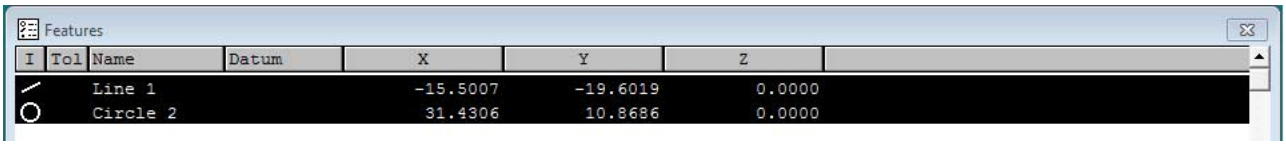
Select **Tools>Options...**



Select the **General** tab and the **Prompt for ODBC Export** checkbox

To send data to a target Microsoft Excel file:

- Select features or records from the available data templates

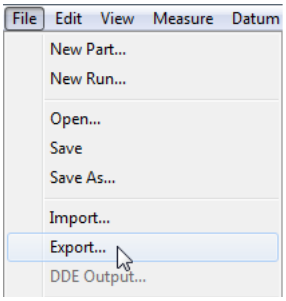


Select features or records

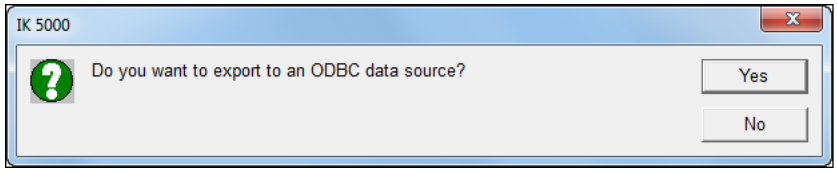
- Select **File>Export...**

A prompt will request confirmation to export data to an ODBC data source.

- Click **Yes**



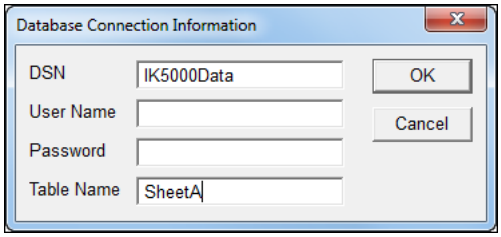
Select **File>Export...**



Click **Yes**

The Data Connection Information prompt appears.

- Enter the **DSN** (Data Source Name), **User Name** and **Password** for the data source
- Enter the **Table Name** that the data will be exported to
- Click **OK**



Enter the Database Connection Information and click **OK**

The target database file must be closed before performing an ODBC data export from the IK 5000.

If using DDE OUTPUT, the target database file has to be open before performing a DDE output from the IK 5000.

2

Setup

2.1 Setup

Updating the IK 5000 Software

The update should be performed according to the following procedure:

- Make a backup of the IK 5000 system settings
- Uninstall the installed IK 5000 software
- Install the new IK 5000 software version
- Restore the IK 5000 system settings

Backup System Settings

- Make a backup of the IK 5000 system settings (via menu: **Help>Backup Settings...**)

Uninstall Old Software Version

- Uninstall the installed IK 5000 software with the tools provided by the operating system (control panel)

Install New Software Version

- Install the new software version 3.0.x

Restore System Settings

- Restore the IK 5000 system settings (via menu: **Help>Restore Settings...**)

If an option that requires a license key is enabled the current license key remains valid. A new license key is not required.

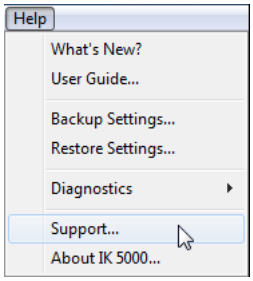
Support Information

IK 5000 technical support may be obtained by contacting HEIDENHAIN or the Original Equipment Manufacturer (OEM).

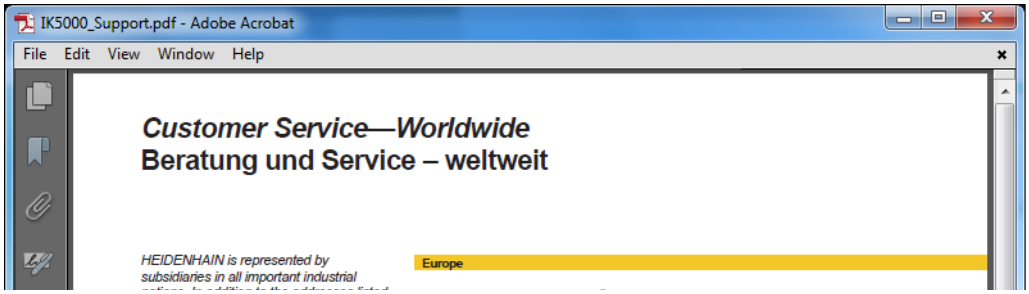
To access support contact information:

▶ Click **Help>Support**

A customer support information PDF file will open.



Click **Help>Support**



Customer support information is displayed

Files and Folders

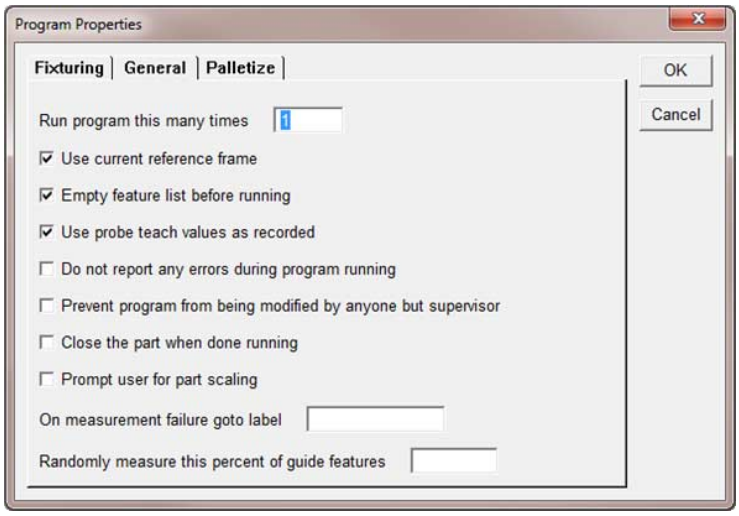
IK 5000 files and folders locations are distributed throughout Windows operating system folder locations in accordance with the operating system's files and folders location guidelines.

The files and folders are distributed as follows:

- IK 5000 files are located in the HEIDENHAIN\IK 5000 folder which is in the **Program Files** OS folder
- IK 5000 application is located in HEIDENHAIN folder in the **Start** menu
- Configuration files are stored in IK 5000 folder in **Program Data** OS folder
- User settings files are saved to the IK 5000 folder in **Local User** OS folder
- User document files are saved to the IK 5000 folder in the **Documents** OS folder

Some file types that were supported in previous versions are no longer used. File type .met is no longer supported. Files of this type have been removed from the system.

Use Current Reference Frame



Program Properties window

The **Use machine reference** option in the **Program Properties** dialog has been renamed to **Use current reference frame**. When an existing program is loaded into IK 5000 SW v3.0.0 or higher, the **Use current reference frame** option will take on the opposite state of the **Use machine reference** option.

The reference frame that is in place when a program is first recorded is now stored within the program. If **Use current reference frame** is unchecked, the program will use this stored reference frame as its initial reference frame when the program is executed.

If **Use current reference frame** is checked, the program will use the current part reference frame when the program is executed.

To change the current part reference frame:

- Select a different part reference frame from the status bar

The default state of **Use current reference frame** is unchecked. If a part reference frame has been created before the program is first recorded **Use current reference frame** will be automatically checked.

Programs written prior to SW v2.96.0 do not include the reference frame that was in place when the program was first recorded. If **Use current reference frame** is unchecked the program will use the machine reference frame as the initial reference frame when the program is executed.

If **Use machine reference** was previously used the same behavior can be obtained with **Use current reference frame**.

To obtain the same behavior:

- Select the machine reference frame prior to recording the program
- Uncheck **Use current reference frame** in the **Program Properties** dialog prior to executing the program

or

- Check **Use current reference frame**
- Select the machine reference frame prior to executing the program

Camera Parameters

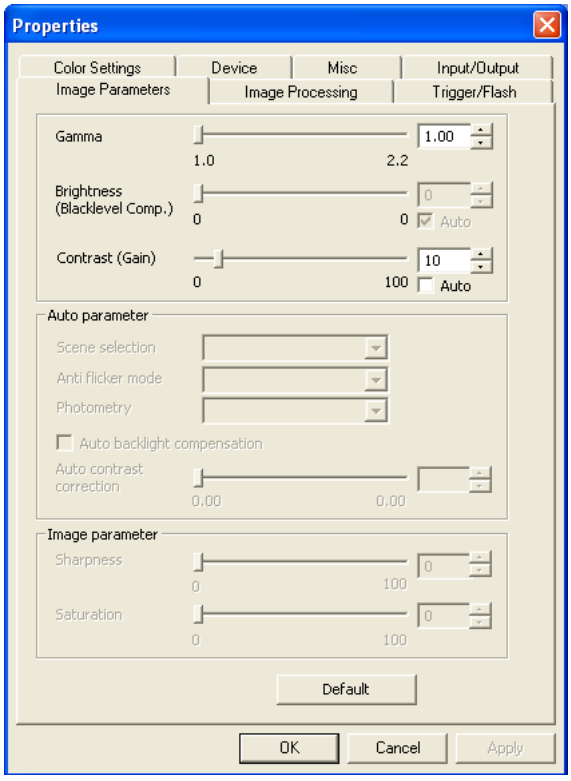
The IK 5000 supports the configuration of camera and image quality parameters through DirectShow interfaces. The configurable parameters are limited to those provided by the camera driver support for DirectShow. The IK 5000 cannot modify a parameter that the camera driver does not allow to be modified.

Example configurable parameters are gamma, brightness, contrast and color saturation. Brightness and contrast settings are saved when the IK 5000 is shut down and restored on the next start up.

To access digital camera parameters:

- Right-click in the **Video** window
- Select **Video Properties**

A video properties window will open displaying the available camera parameters that can be edited.



Example video properties window

Right click in the Video window and select Video Properties

Specify Video Probe Parameters

Use the **VED** screen in video edge detection systems to specify video probe measurements parameters.

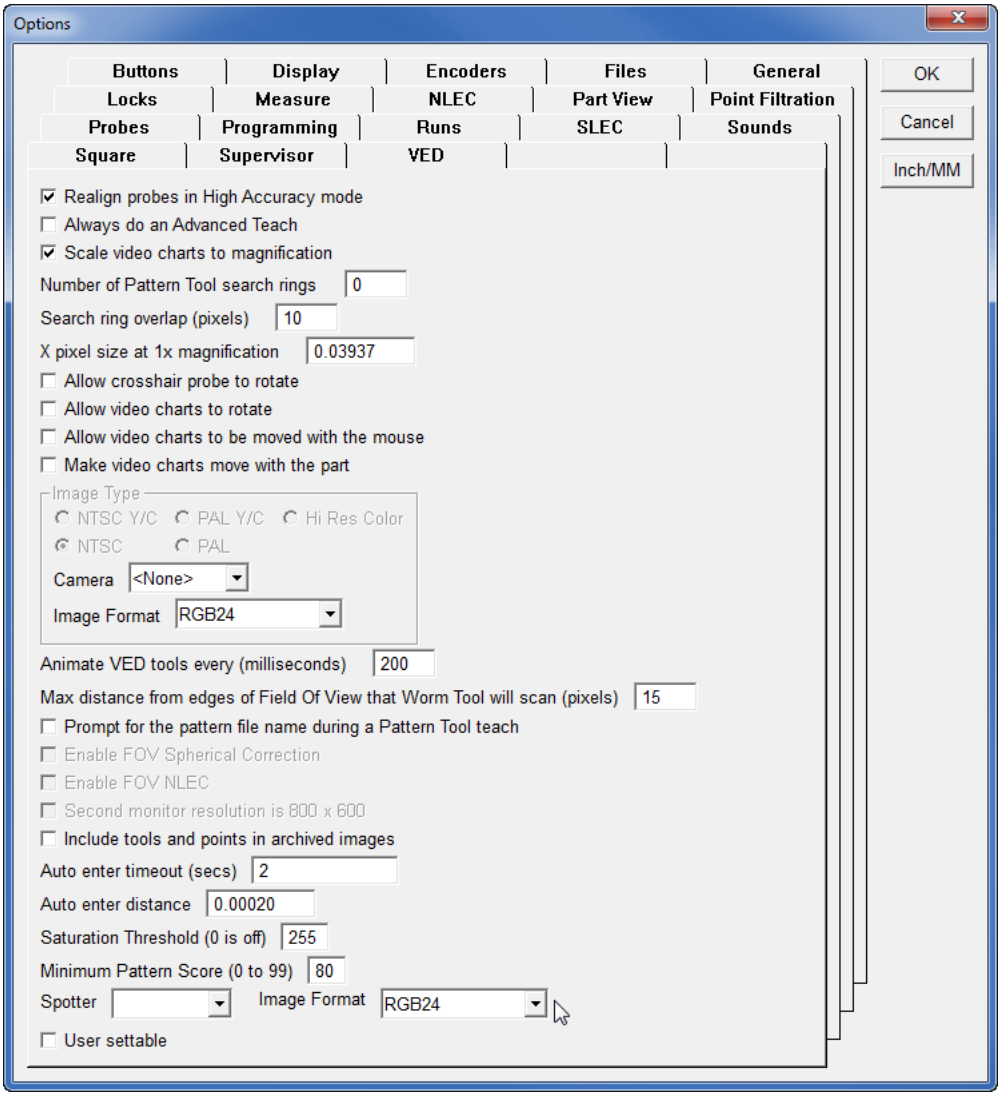
To display the **VED** screen:

▶ Click **Tools>Options...>VED**

Digital video probe and measurement parameters will be displayed for your system.

New options:

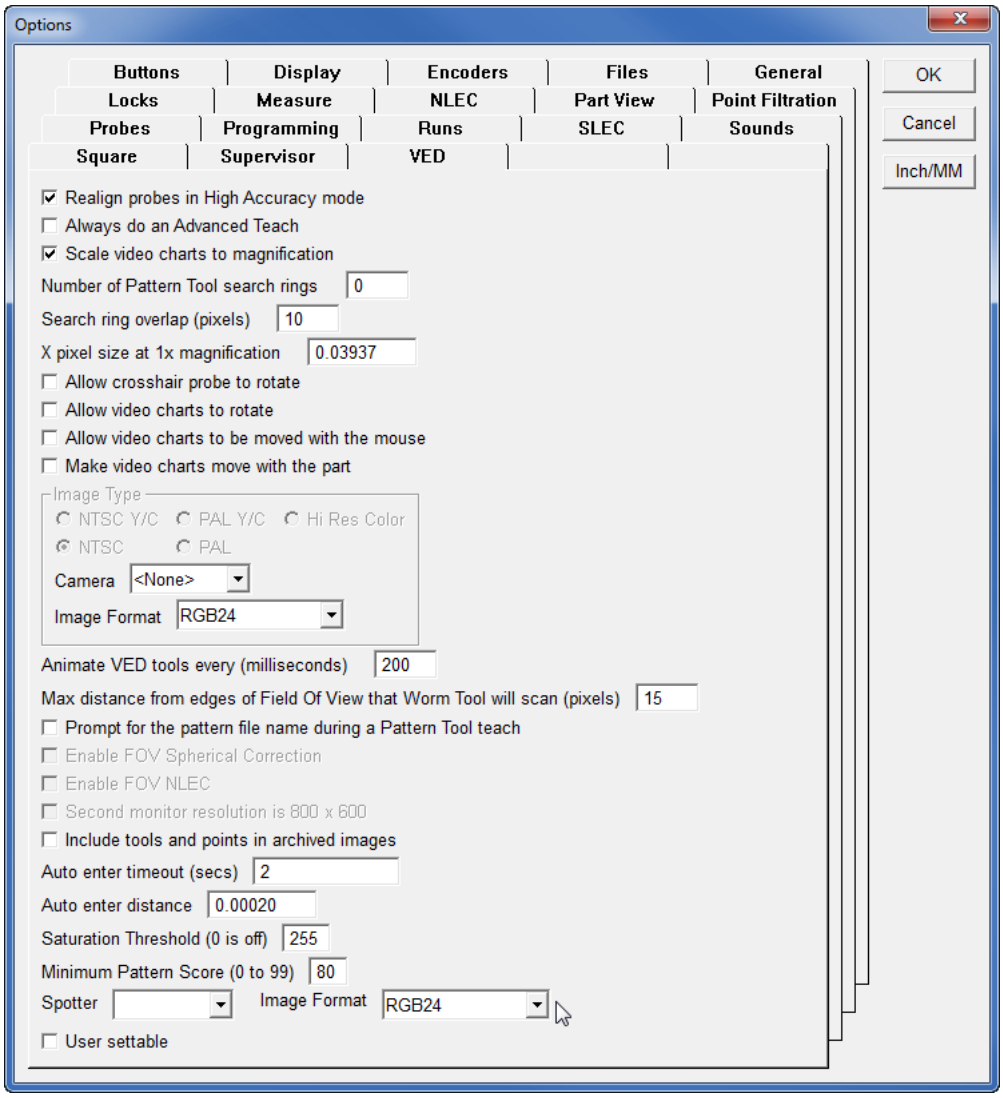
- **Camera**: Select the camera to use for VED (previously unnamed)
- **Image format**: Select the image color format



VED screen example

Spotter Camera Options

The digital camera selected as the spotter camera and the format for the spotter camera can be set from the **VED** setup tab.



Spotter camera options

Supported image formats:

- UYVY
- RGB24
- RGB32

To access Spotter camera options:

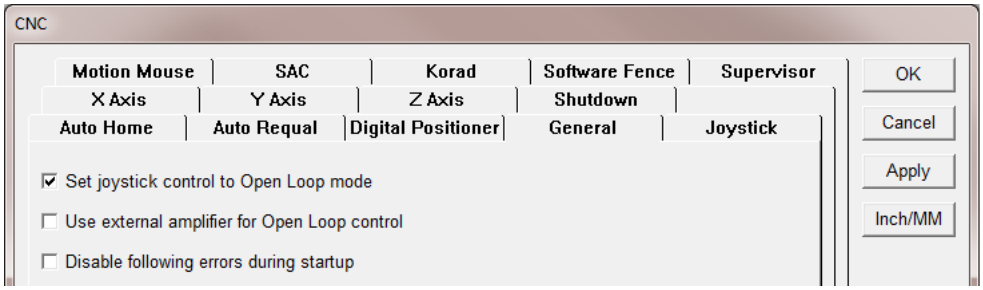
- Select **Tools>Options...**
- Select the **VED** tab
- Select a digital camera from the **Spotter** drop down menu
- Select an image format from the **Image Format** drop down menu
- Click **OK**

Scale Factor for Open Loop Joystick Control (Optional)

A CNC system can be operated in either **Closed Loop** or **Open Loop** joystick control mode. This can cause a system to move with different velocities when a user switches from one mode to the other. To make the velocity comparable, a user can apply a scaling factor to the joystick parameters when in open loop control mode. The Scale factor ranges from 0.01 to 1.00. This is a unit-less value. The default value is 0.5.

Verify the system is set to Open Loop joystick control mode:

- Click **Tools>CNC...**
- Enter the **Password**
- Click the **General** tab
- Select the **Set joystick control to Open Loop mode** checkbox if not already selected

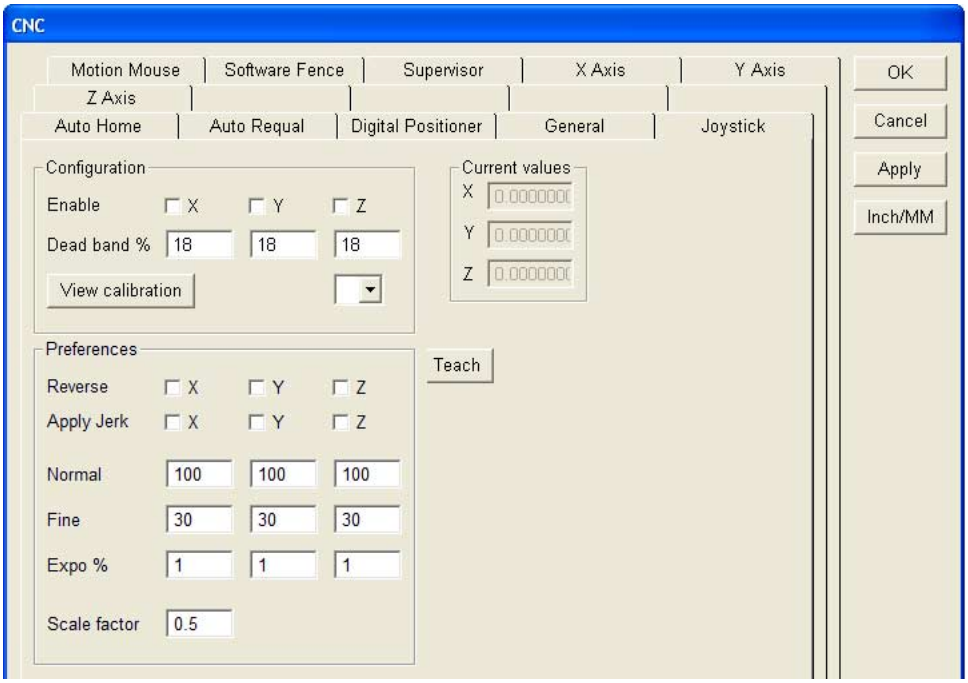


Set joystick control to Open Loop mode

- Click **OK**

Set Scale factor:

- Click **TOOLS>CNC...**
- Enter the **Password**
- Click the **Joystick** tab
- Enter the desired scaling factor into the **Scale factor** field

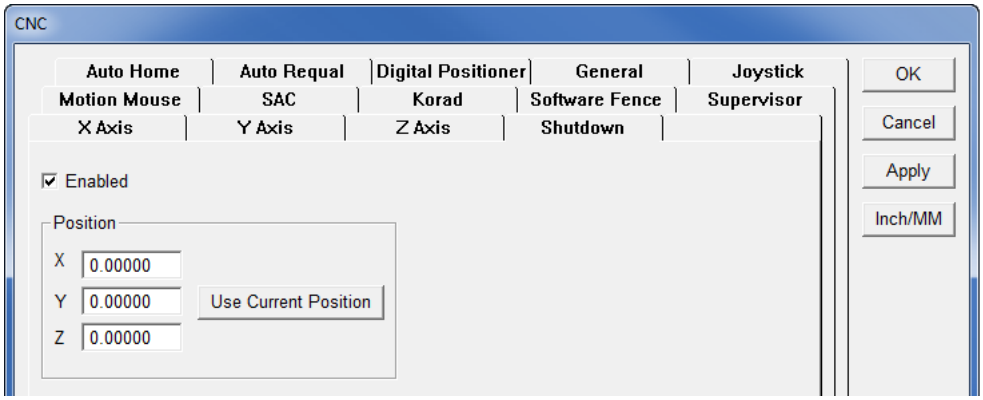


Enter the desired scaling factor

- Click **Apply**
- Click **OK**

Shutdown

The Shutdown function applies to CNC enabled systems only. During system shutdown the probe returns to a position defined in machine coordinates regardless of probe type. The position may be specified using X, Y and Z coordinates or by choosing the current probe location.



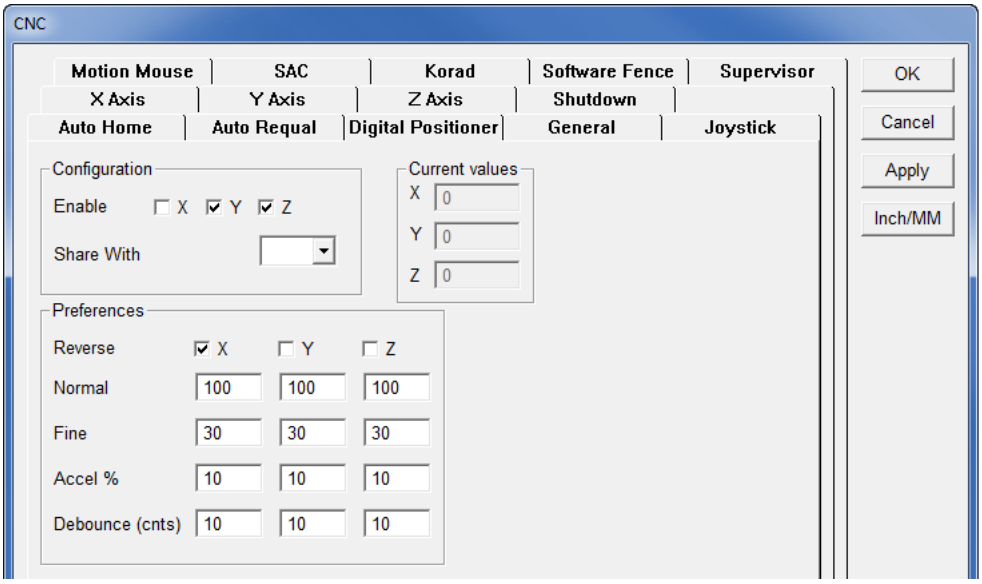
Shutdown settings window

To specify the probe position at shutdown:

- Select **Tools>CNC...**
- Enter the Supervisor password into the **Password** field
- Select the **Shutdown** tab
- Select the **Enabled** checkbox
- Enter the X, Y and Z coordinates or click **Use Current Location**
- Click **OK**

Debounce Parameters

Y and Z axis debounce count values for the digital positioner use their own axis value. Previously they used the X axis value.



Digital Positioner settings window

To specify debounce count values:

- Select **Tools>CNC**...
- Enter the Supervisor password into the **Password** field
- Select the **Digital Positioner** tab
- Enter values into the **Debounce (cnts)** fields
- Click **OK**

Software Fences

A software fence is used to limit axial movement in a CNC system.

The IK 5000 has three software fence options:

- Machine Fence
- User Fence 1
- User Fence 2

A machine fence is a software fence established to prevent collisions at the end of an axis.

A user fence is a software fence established to limit axial travel, usually in relationship to part geometry.

If a user fence and a machine fence are active at the same time the more restrictive fence defines the limits on an axis

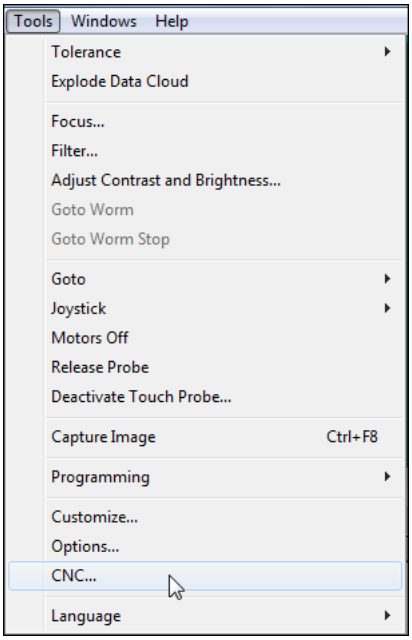
A software fence may be established in two ways:

- Manually
- Using the Teach function

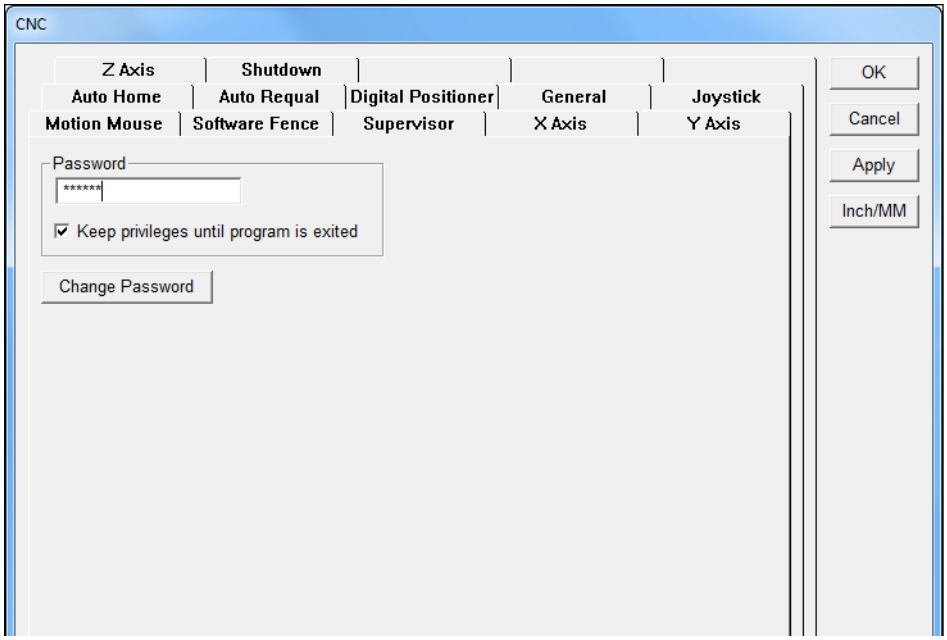
Manually Creating a Software Fence

To manually create a software fence:

- Select **Tools>CNC...**
- Select the **Supervisor** tab
- Enter the Supervisor **Password**
- Press the **Enter** key on the keyboard

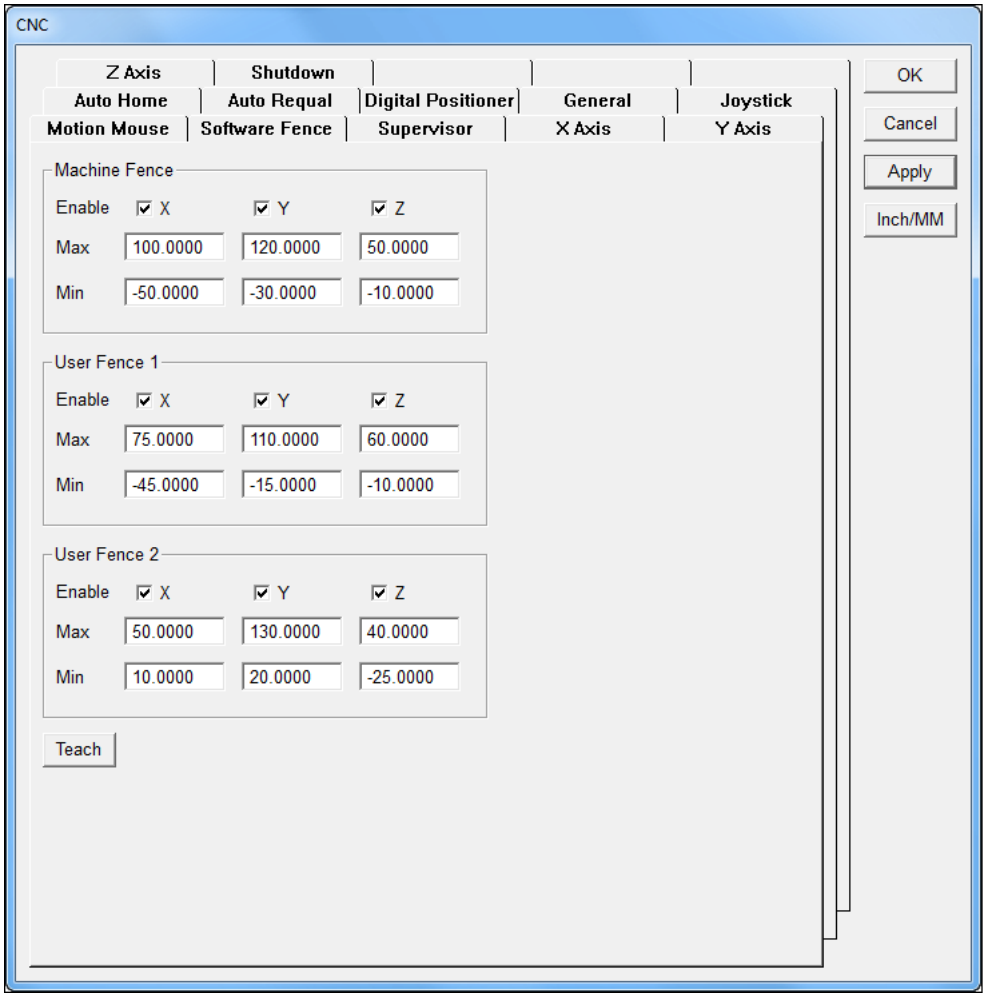


Select **Tools>CNC...**



Select the **Supervisor** tab, enter the Supervisor **Password** and press **Enter** on the keyboard

- Select the **Software Fence** tab
- Enter maximum coordinates in the **Max** field for each desired axis
- Enter minimum coordinates in the **Min** field for each desired axis
- Select the **Enable** checkbox next to an axis to enable the software fence for that axis. If the **Enable** checkbox for an axis is not checked, the software fence is not applied to that axis.



Select the **Software Fence** tab, enter coordinates and enable axes

- Click **Apply** to apply the changes

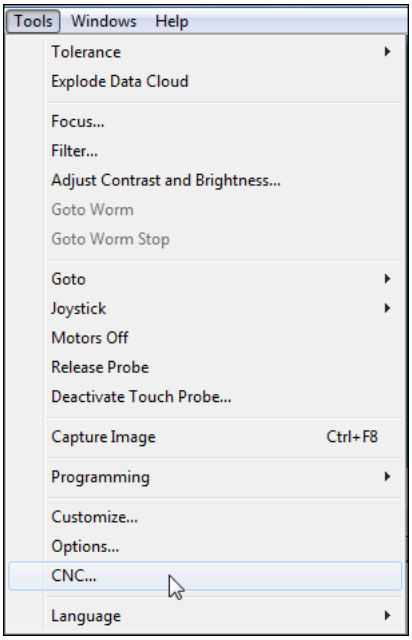
or

- Click **OK** to apply the changes and exit the **CNC** window

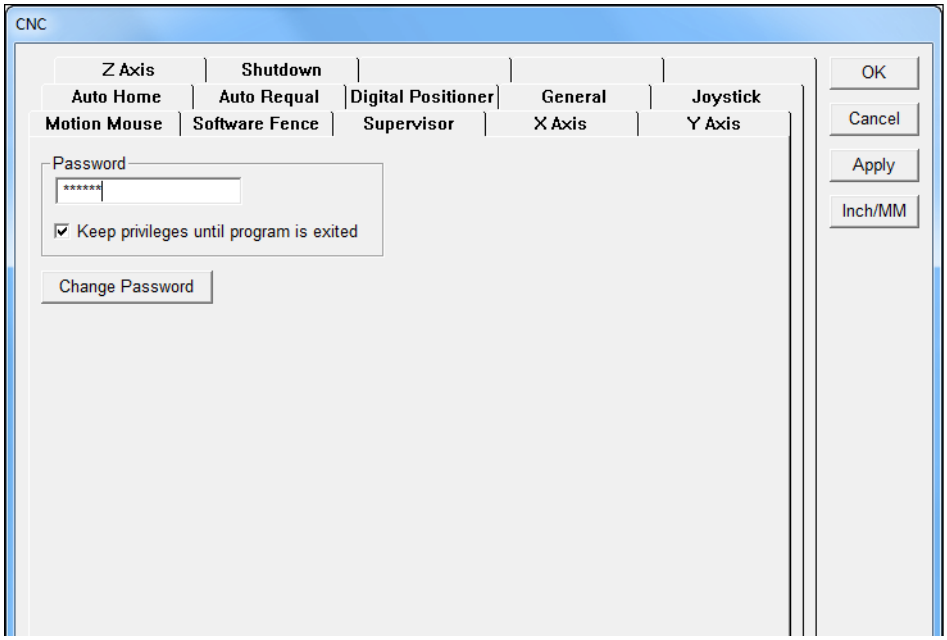
Using the Teach Function to Create a Software Fence

To create a software fence using the Teach function:

- Select **Tools>CNC...**
- Select the **Supervisor** tab
- Enter the Supervisor **Password**
- Press the **Enter** key on the keyboard

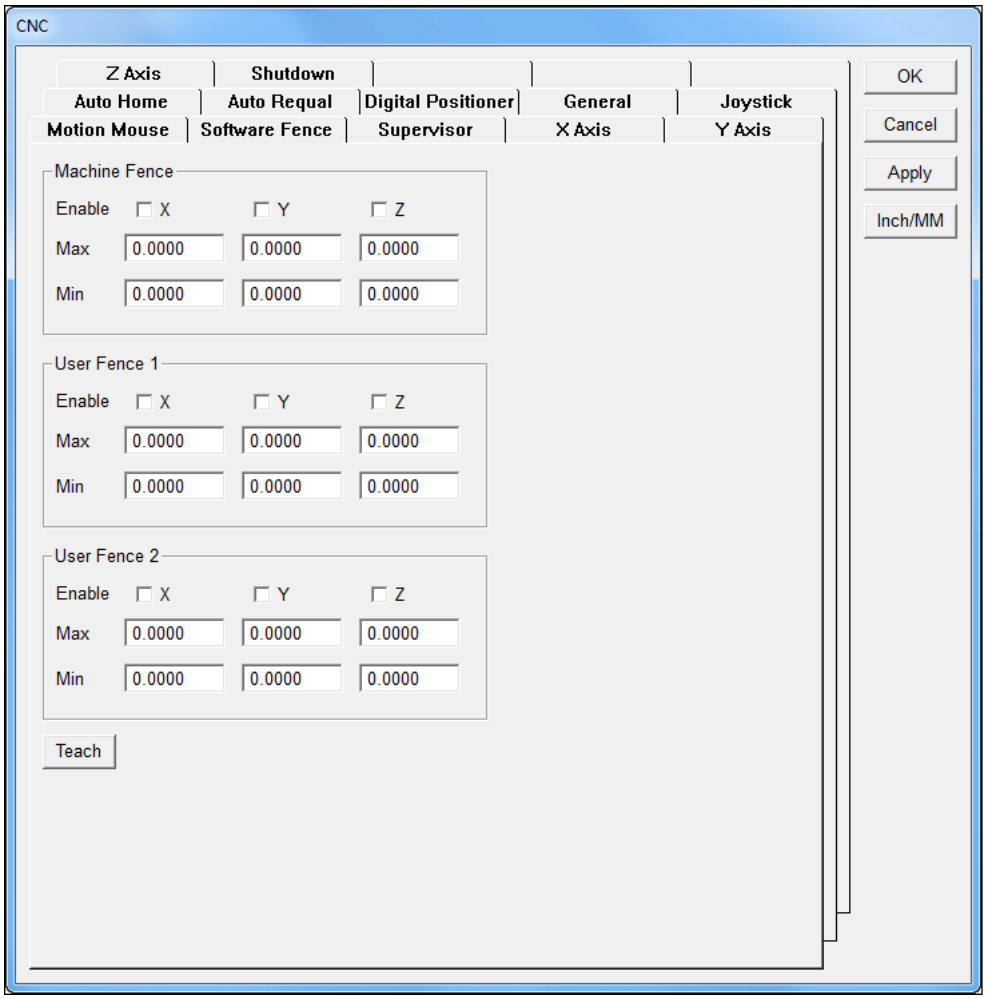


Select **Tools>CNC...**



Select the **Supervisor** tab, enter the Supervisor **Password** and press **Enter** on the keyboard

- Select the **Software Fence** tab
- Click the **Teach** button



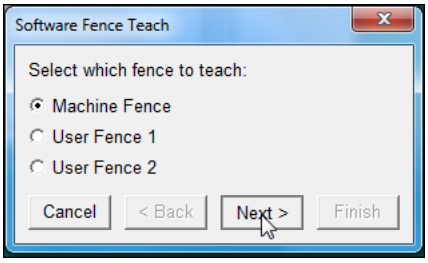
Select the **Software Fence** tab and click the **Teach** button

The **Software Fence Teach** dialog window opens.

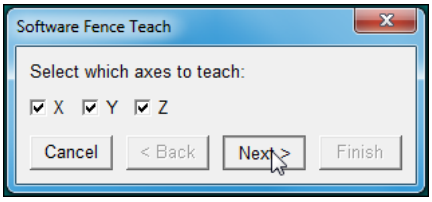
- Click a radio button to select the desired software fence to teach
- Click **Next**
- Check the box next to the axis or axes to teach

Any single axis or combination of axes may be selected.

- Click **Next**

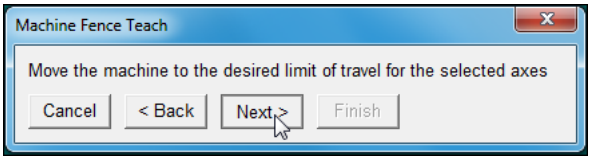


Click a radio button to select the desired software fence to teach and click **Next**

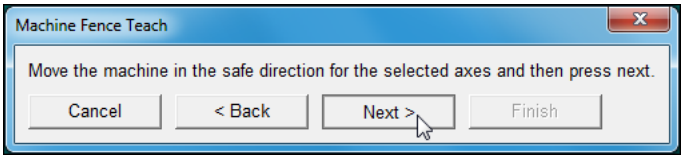


Check the box next to the axis or axes to teach and click **Next**

- Move each axis to the desired limit of travel for the selected axes. In general, position the axes at the point that is to mark the beginning of positive travel.
- Click **Next**
- Move each axis in the safe direction
- Click **Next**



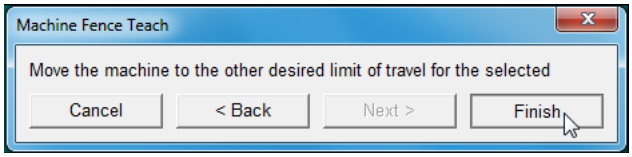
Move each axis to the desired limit of travel and click **Next**



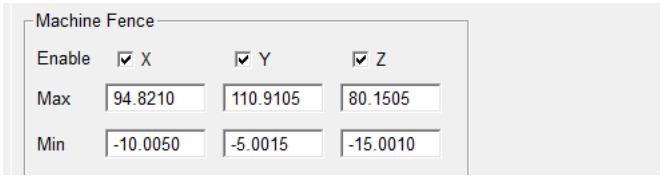
Move each axis in the safe direction and click **Next**

- Move each axis to the desired limit of travel in the opposite direction. This should be the furthest distance away for each axis to travel from the point taught in the previous two steps.
- Click **Finish**

The system fills in the minimum and maximum values.

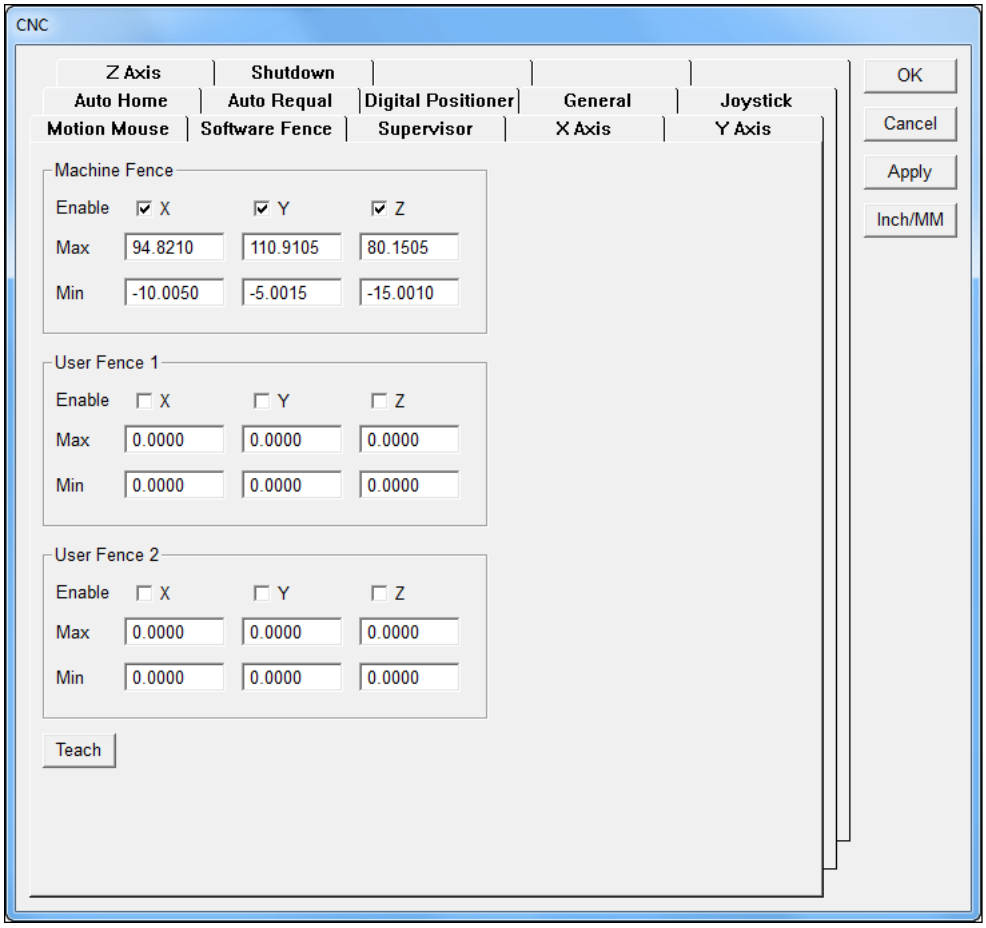


Move each axis to the desired limit of travel in the opposite direction and click **Finish**



The system fills in the minimum and maximum values.

- Select the **Enable** checkbox next to an axis to enable the software fence for that axis. If the **Enable** checkbox for an axis is not checked, the software fence is not applied to that axis.



Enable axes

- Click **Apply** to apply the changes

or

- Click **OK** to apply the changes and exit the **CNC** window

If a user fence and the machine fence are active at the same time the more restrictive fence defines the limits on an axis

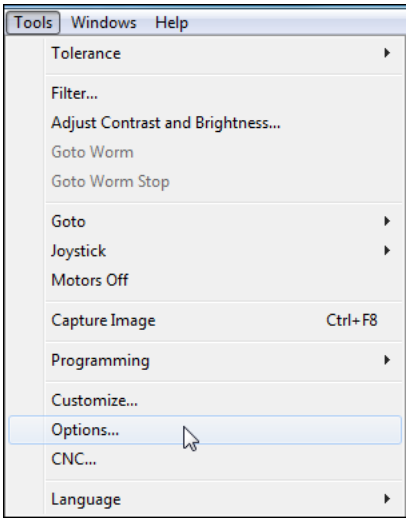
Joystick Button Assignment

Joystick buttons can be assigned to toggle the user fence state or to activate a user fence.

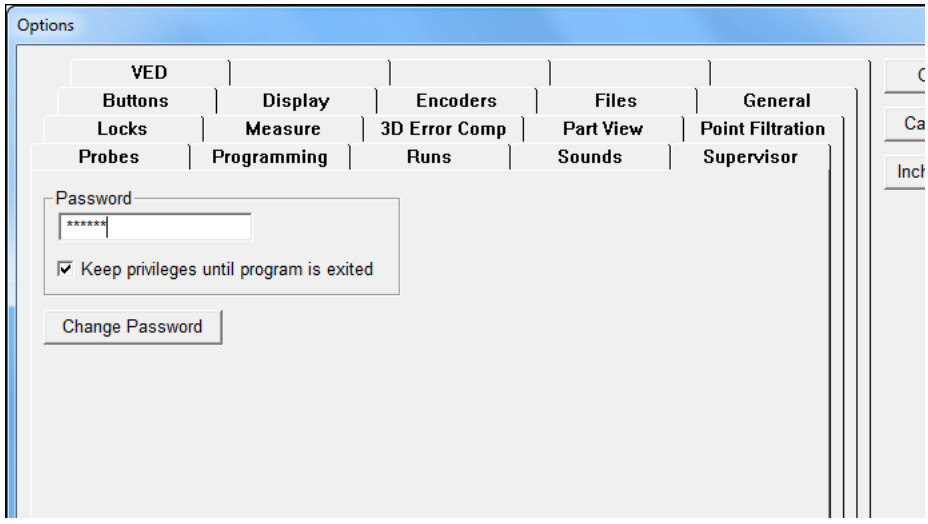
Option	Description
Toggle User Fence	Toggles the user fence state between **Fence Off**, **User Fence 1** and **User Fence 2** when the assigned button is pressed.
Activate User Fence 1	Activates User Fence 1 settings when the assigned button is pressed.
Activate User Fence 2	Activates User Fence 2 settings when the assigned button is pressed.

To assign a user fence option to a button:

- Click **Tools>Options...**
- Select the **Supervisor** tab
- Enter the supervisor **Password**

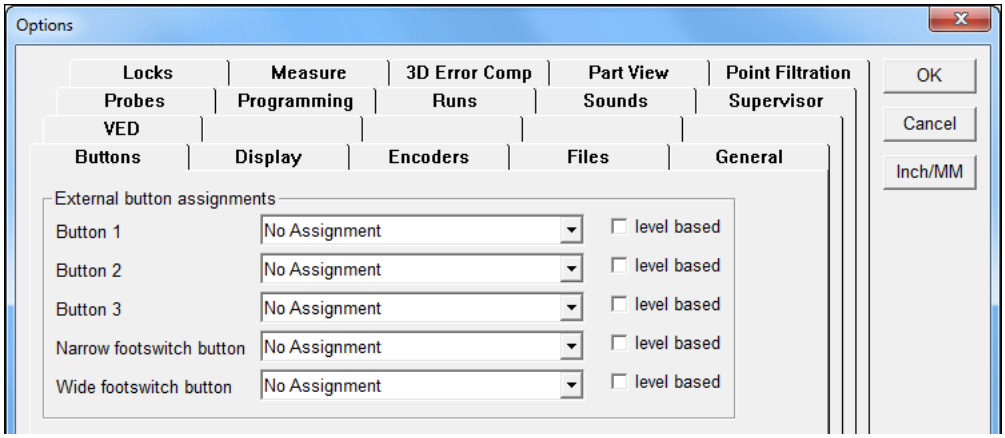


Click **Tools>Options...**



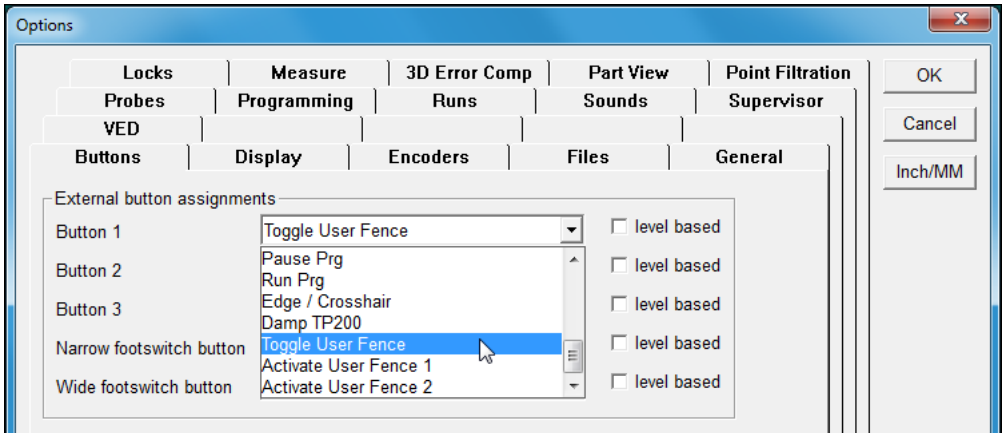
Select the **Supervisor** tab and enter the supervisor **Password**

▶ Select the **Buttons** tab



Select the **Buttons** tab

▶ Select **Toggle User Fence**, **Activate User Fence 1** or **Activate User Fence 2** from the drop-down list of the desired button



Select **Toggle User Fence**

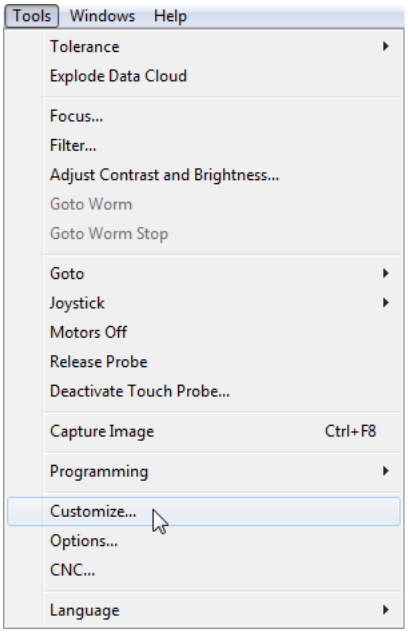
▶ Click **OK** to apply the changes and exit the **Options** window

User Fence Status Display

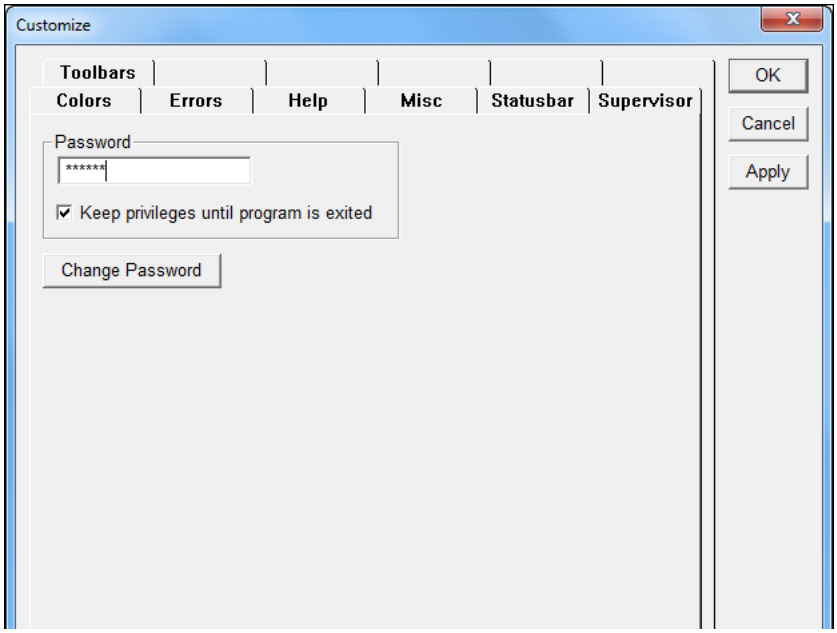
The user fence state can be displayed and toggled in the status bar.

To show or hide the user fence status bar item:

- Click **Tools>Customize...**
- Select the **Supervisor** tab
- Enter the supervisor **Password**

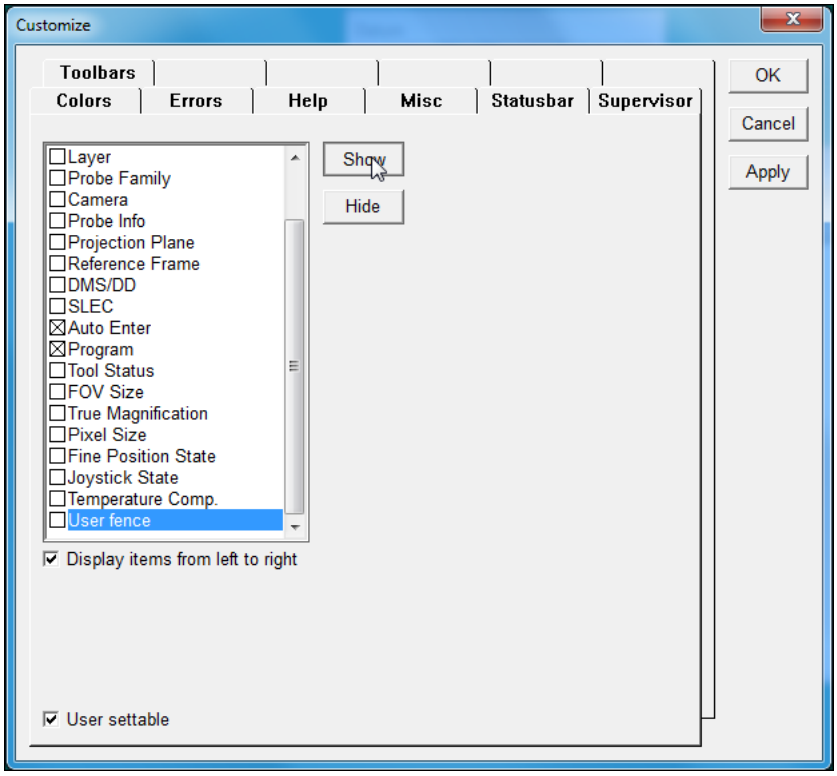


Click **Tools>Customize...**



Select the **Supervisor** tab and enter the supervisor **Password**

- Select the **Statusbar** tab
- Scroll to the **User fence** option and select it
- Click **Show** to display the User fence status or **Hide** to remove the User fence status item from the status bar



Select the **Statusbar** tab, select **User fence** and click **Show** or **Hide**

- Click **Apply** to apply the changes

or

- Click **OK** to apply the changes and exit the **CNC** window

The status bar will be updated.

Toggling the User Fence State

The user fence can be toggled between **Fence Off**, **User Fence 1** and **User Fence 2** when displayed in the status bar.

To toggle the user fence state:

- Click the user fence state in the status bar

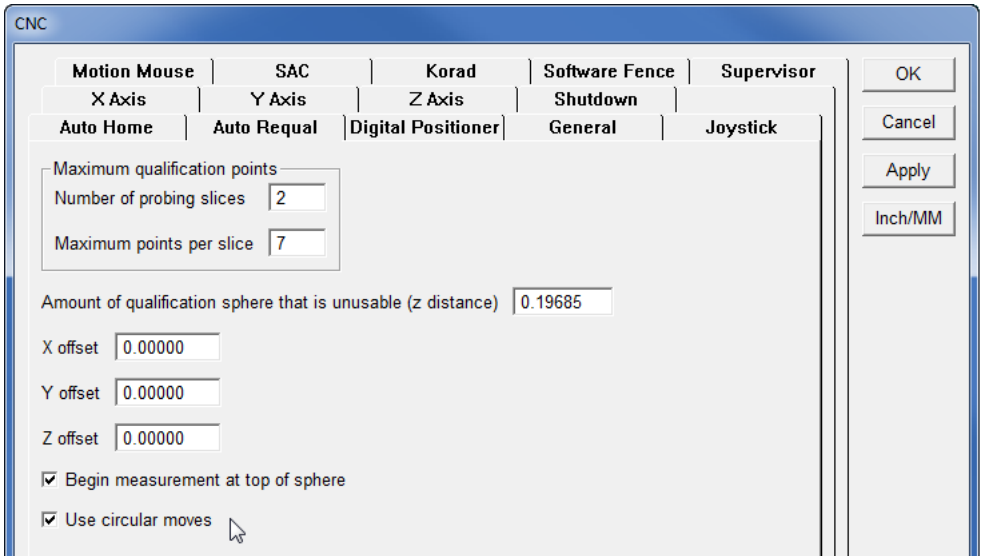
The state will toggle to the next user fence option.

Click the user fence state in the status bar to toggle the status.

Touch Probe Requalification and Starting Point

Touch probe auto requalification can be set to **Use circular moves** and to begin the requalification measurements at the top of the sphere and not along the diameter.

Deselecting this option allows the automatic requalification to be executed by linear movements, minimizing stage vibrations for systems with large measuring stages.



Auto Requal settings window

To select 'Auto Requal' parameters:

- Select **Tools>CNC**...
- Enter the Supervisor password into the **Password** field
- Select the **Auto Requal** tab
- Select the **Begin measurement at top of sphere** and **Use circular moves** checkboxes
- Click **OK**

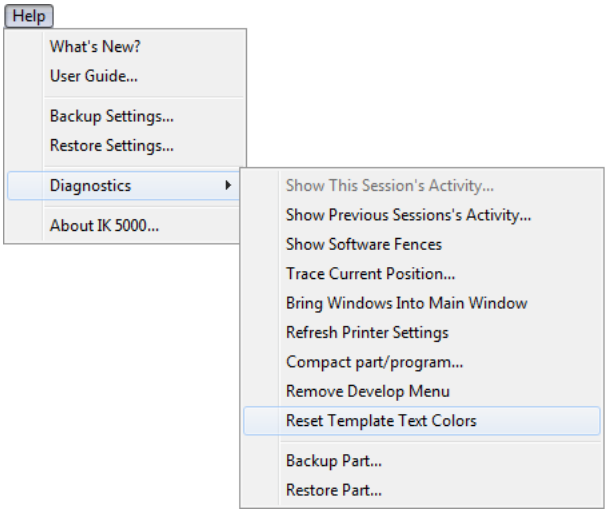
Reset Template Text Colors

It is possible to set the text color of a template to the same color as the background. When this happens the text is not readable.

The **Reset Template Text Colors** option makes it simple to revert the text colors for all open templates back to the default color. The default text color for all templates is black.

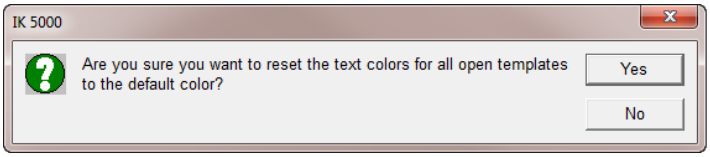
To reset template text colors:

- Select **Help>Diagnostics>Reset Template Text Colors**. A prompt appears requesting verification to reset the text colors.



Select **Help>Diagnostics>Reset Template Text Colors**

- Click **Yes**



Click **Yes**

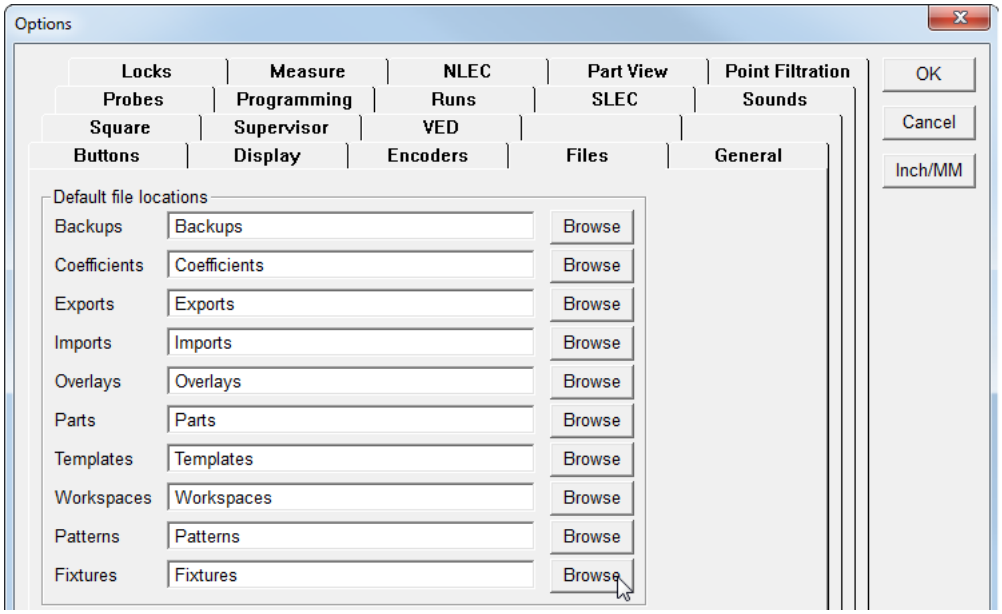
Default Fixtures Files Location

A default location for Fixtures files can be selected. The IK 5000 uses this location to save and load reference frames.

The default location is the Fixtures subfolder of the IK 5000 folder located in the user's system documents folder.

To change the Fixtures file location:

- Select **`Tools>Options...`**
- Select the **`Files`** tab
- Browse or enter a new **`Fixtures`** file location



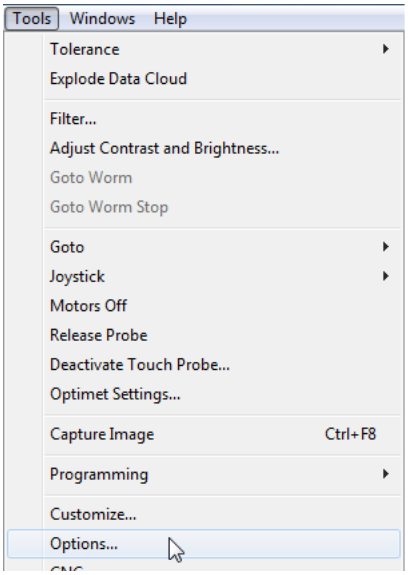
Browse or enter a new **`Fixtures`** file location

Disabling measurements when homing is not completed

Measurements can be disabled when machine zero homing is not completed.

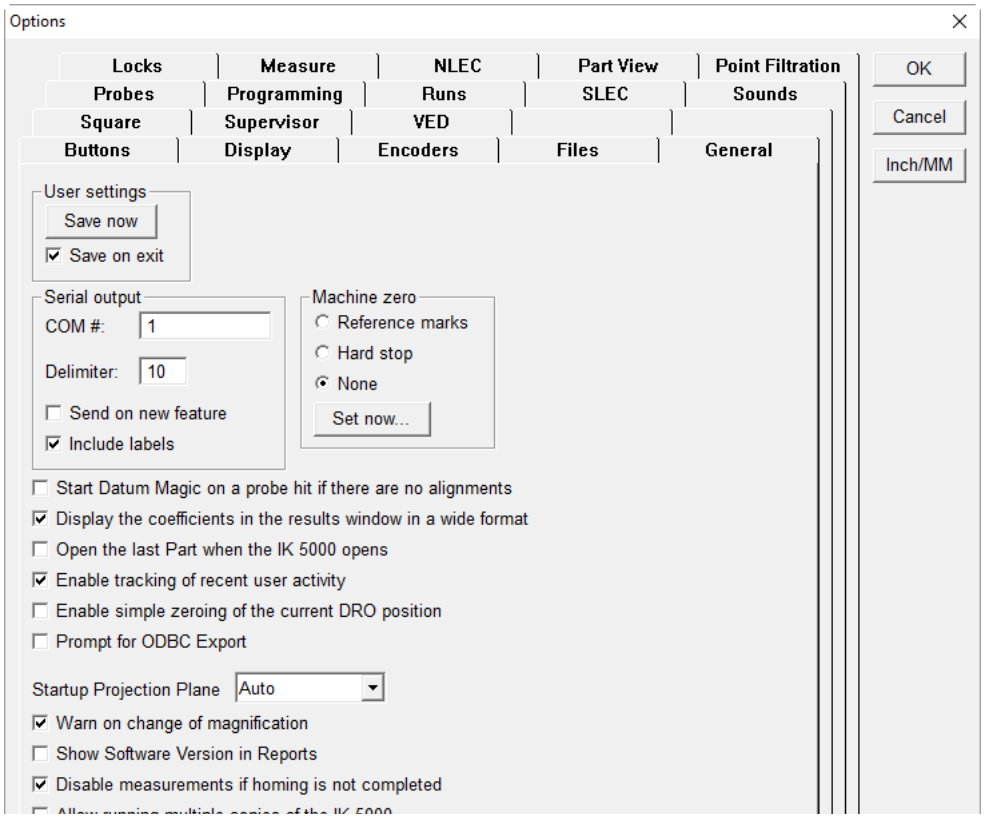
To disable measurements when homing is not complete:

- Click **Tools>Options…**



Click **Tools>Options…**

- Select the **General** tab
- Check **Disable measurements if homing is not completed**



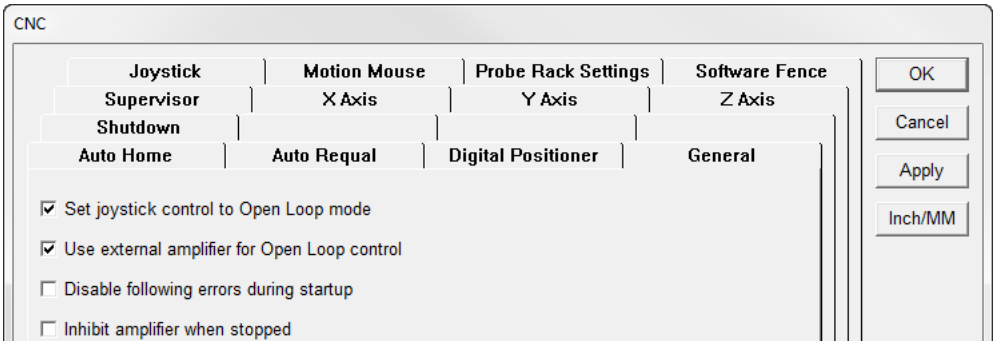
Click **Tools>Options…** and check **Disable measurements if homing is not completed**

- Click **OK** to save changes and exit the Options window

External Amplifier Open Loop Control (Optional)

IK 5000 Open Loop joystick control provides the ability to toggle between two joystick speeds when Speed Toggle is assigned to a joystick button. External amplifier Open Loop joystick control provides the ability to toggle between three speeds. The ability to toggle between three speeds provides increased control over the velocity of movements.

When **Set joystick to Open Loop mode** and **Use external amplifier for Open Loop control** are enabled, Open Loop functionality is controlled by the HEIDENHAIN amplifier connected to the IK 5000 system. **Use external amplifier for Open Loop control** is disabled by default.



CNC window General settings tab

To enable external amplifier Open Loop control:

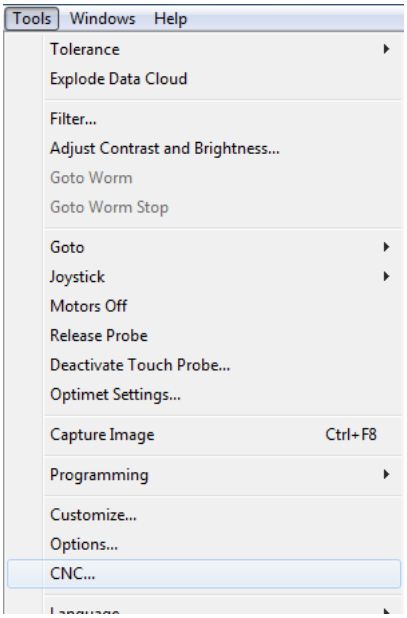
- Select **Tools>CNC...**
- Select the **General** tab
- Select the **Set joystick to Open Loop mode** and **Use external amplifier for Open Loop control** checkboxes
- Click **OK**

Warning on CNC movement

A pop-up warning can be enabled that warns you when the system is about to make a CNC movement.

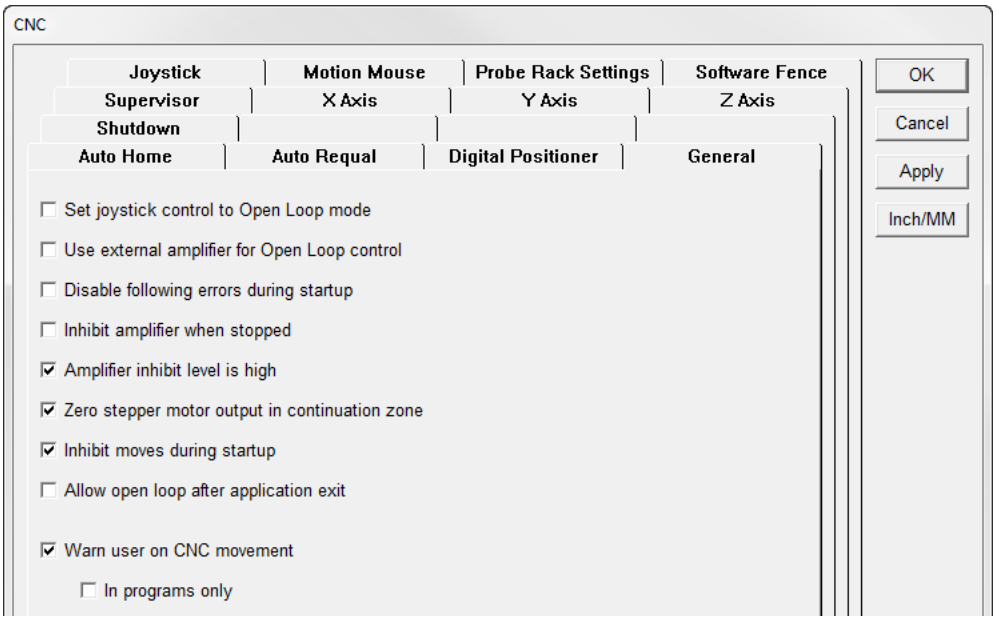
To enable the CNC movement warning:

- Click **Tools>CNC…**



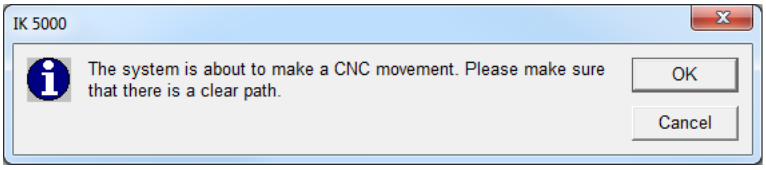
Click **Tools>CNC…**

- Select the **General** tab
- Check **Warn user on CNC movement**



Check **`Warn user on CNC movement`**

A pop-up warning will now appear when the CNC is going to make a movement.

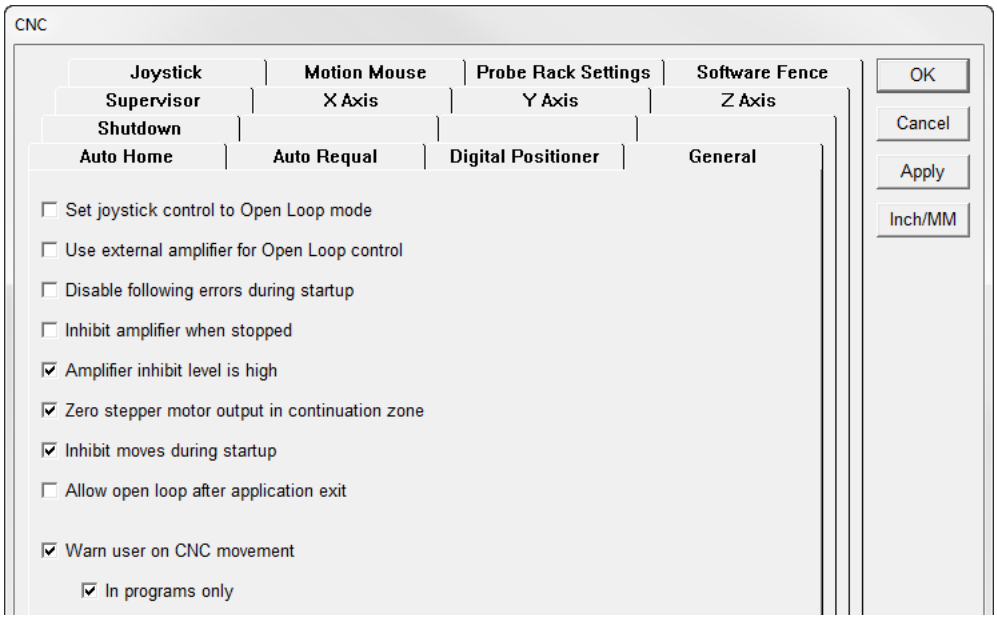


CNC movement pop-up warning

The warning can be restricted to the first CNC movement in a program.

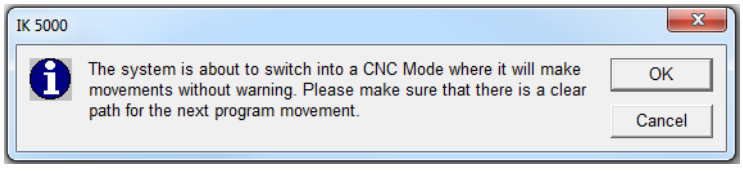
To restrict the warning to the first movement:

- Check **In programs only**



Check **In programs only**

The warning will now only appear on the first movement of a program.



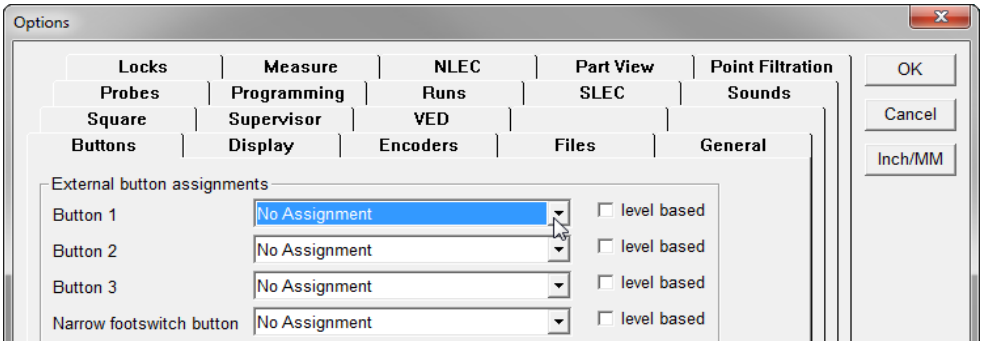
Pop-up warning on first CNC movement

- Click **OK** button to save the changes and exit the Options window

Joystick Button Assignment

Speed Toggle must be assigned to a joystick button in order use joystick speed control. The HEIDENHAIN QUADRA-STEP setup software is used to assign joystick button functions in the HEIDENHAIN amplifier. QUADRA-STEP setup software and the QUADRA-STEP setup guide can be downloaded from *www.heidenhain.de*.

The IK 5000 and external amplifier both sense joystick button presses. When Speed Toggle is assigned to a joystick button in the external amplifier, the same button must be set to **`No Assignment`** in the IK 5000 Options settings.



Options window Buttons tab

To set a IK 5000 joystick button assignment:

- Click **`Tools>Options`**
- Select the **`Buttons`** tab
- Click the drop-down arrow for the joystick button Speed Toggle was assigned to in the external amplifier
- Select **`No Assignment`**
- Click **`OK`**

RS-232 COM port allocations

The IK 5000 uses RS-232 for data transfer and to communicate with custom devices. Serial communication ports (COM ports) are configured to enable this communication.

Several COM ports can be available in a computer system and can be used for data transfer and communication. The ports are typically numbered COM 1, COM 2, etc...

The IK 5000 can use any COM port to communicate with an attached serial device and the port to use with each device can be configured.

In some cases it is necessary to assign a specific serial port to be used for a certain device. In these cases the COM port is set to a specific number and cannot be changed.

The COM port can be changed for the following functions and devices:

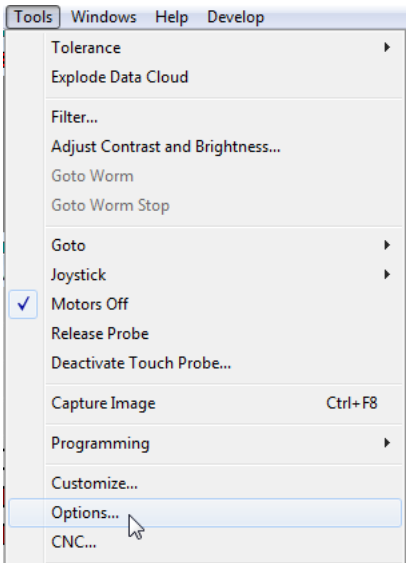
- Sending feature data
- Sending general RS-232 strings

Sending Feature Data

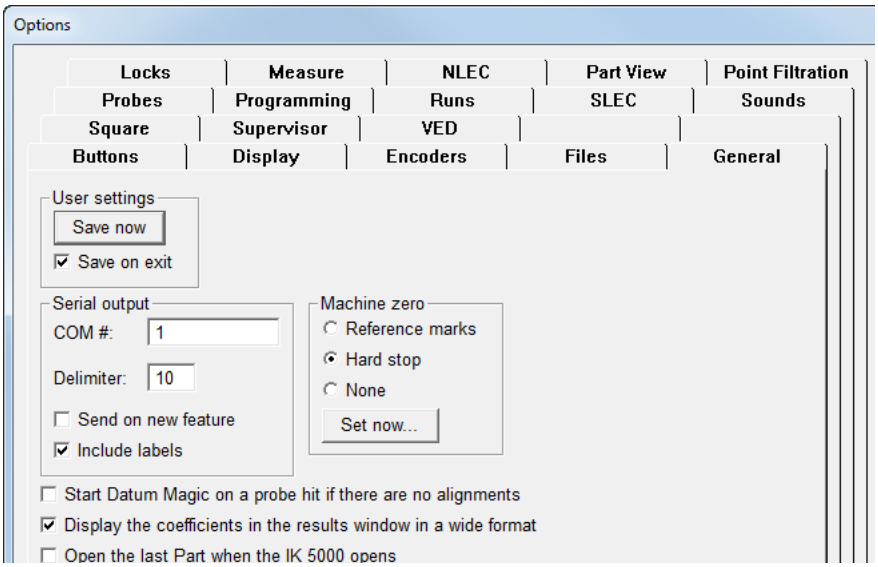
To configure the COM port for sending feature data:

- Select **Tools>Options...**
- Select the **General** tab
- Enter the desired COM port in the **COM#** field

COM 1 is the default setting.



Select **Tools>Options...**



Enter the **COM #**

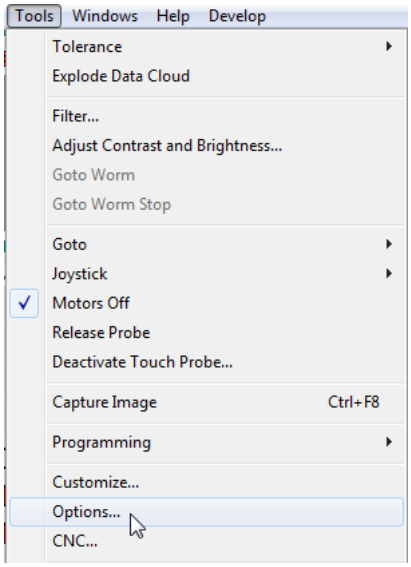
- Click **OK**

Other serial port parameters are set to 9600 baud, 8 data bits, 1 stop bits and 0 parity.

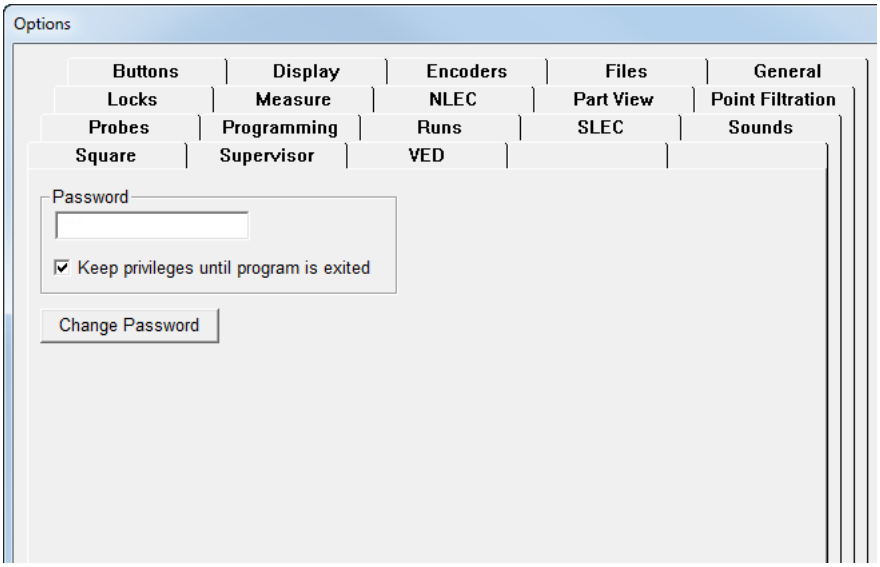
Sending General RS-232 Strings

To configure the settings for sending general RS-232 strings:

- Select **Tools>Options...**
- Select the **Supervisor** tab
- Enter “RS232” in the Password field



Select **Tools>Options...**

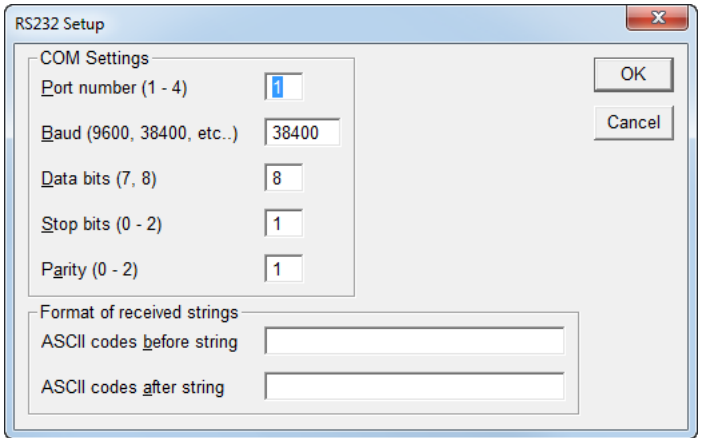


Select the **Supervisor** tab and enter “RS232” in the Password field

The RS232 Setup window appears.

- Enter the desired COM port in the **Port number** field

COM 1 is the default setting.



Enter the COM port in the **Port number** field

- Click **OK**

RS-232 Connections

The IK 5000 can communicate with certain RS-232 capable devices. Sending and receiving data between the IK 5000 and an RS-232 compatible device requires the baud, data bits, stop bits, and parity communications parameters match on both the IK 5000 and the device. The IK 5000 COM port must be set to the port where the device is connected.

The RS-232 cable type used to connect the IK 5000 to the device may vary depending on the device's serial port design. Some devices use a straight-through cable (no cross-overs) while others require a cable with cross-overs. Connecting computer-as-a-device requires a cable with cross-overs. The Serial Port setting on the computer set as the sending/receiving device must be set to match the settings on the IK 5000 serial port in the Windows OS Control Panel.

Printer Selection

The IK 5000 uses the Windows operating system default printer. If the Windows default printer is changed after the IK 5000 has been run, the IK 5000 printer settings need to be updated.

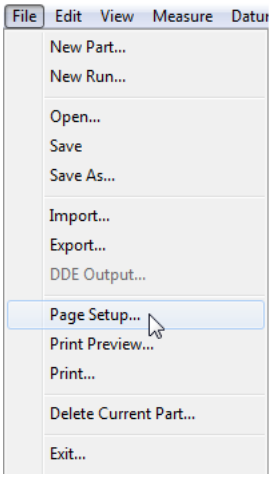
To update the printer settings:

- Click **`File>Page Setup...`**

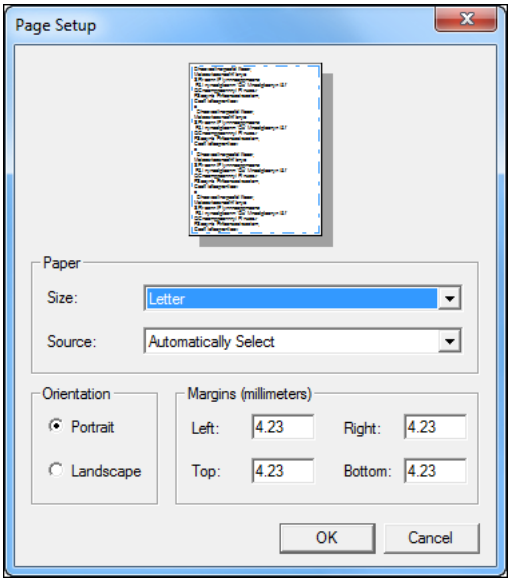
The Page Setup window opens.

- Make any desired changes in the Page Setup window
- Click **`OK`**

The changes are updated and the Page Setup window closes



Click **`File>Page Setup...`**



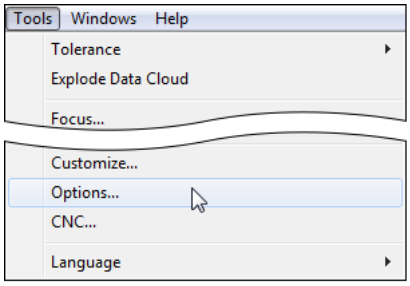
Make any desired changes in the Page Setup window

Displaying the Software Version on Reports

The IK 5000 software version can be displayed in a report header.

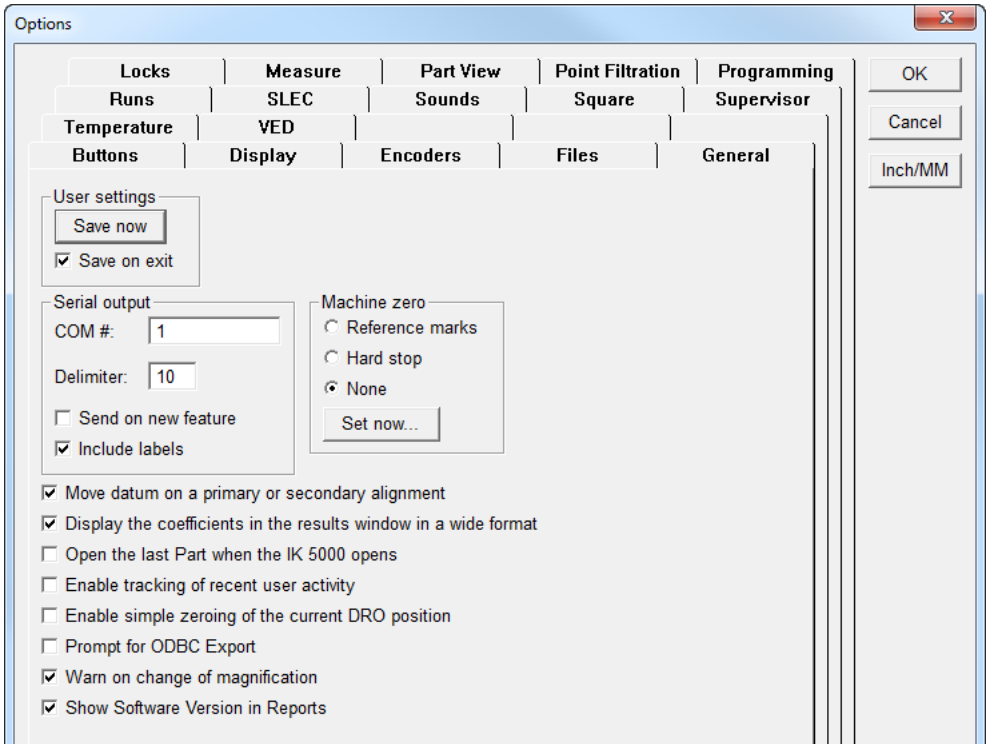
To display the software version:

- Click **Tools>Options…**



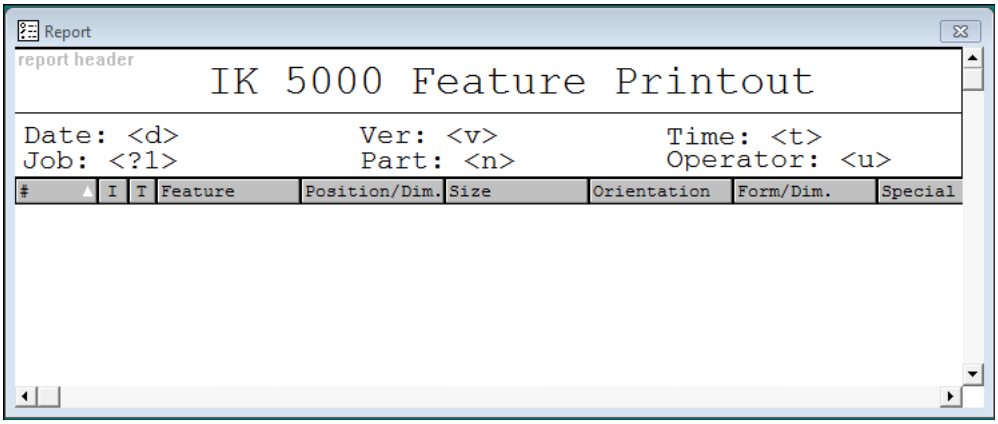
Click **Tools>Options…**

- Select the **General** tab
- Check **Show Software Version in Reports**



Select the **General** tab and check **Show Software Version in Reports**

- Click **OK**
  Ver: <v> will be displayed when the report template header is opened.



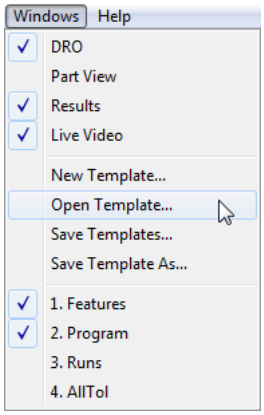
Ver: <v> will be displayed in the report template header.

Displaying System Units On Reports

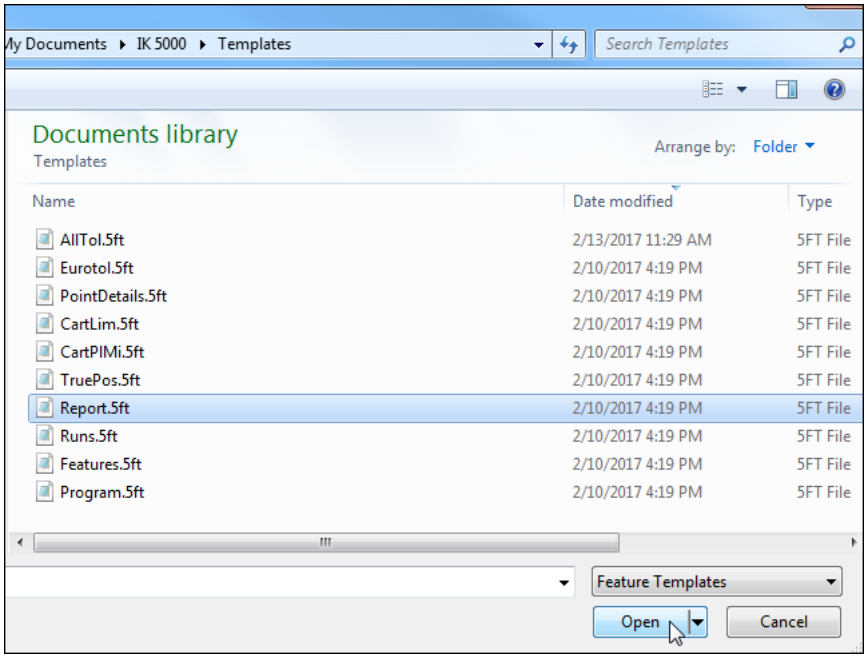
Report headers can display the current system linear units. This option can be activated by enabling **Show report header** for new report templates. For existing templates, the option must be activated manually.

To enable **Show report header** manually in a report template:

- Click **Windows>Open Template…**
- Select a report template from the list and click the **Open** button

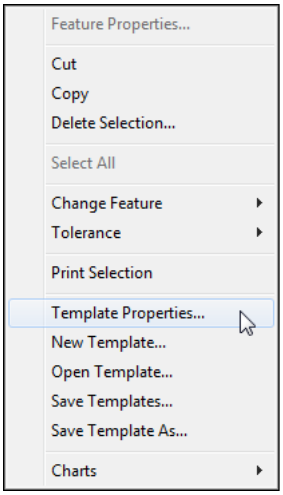


Click **Windows>Open Template…**

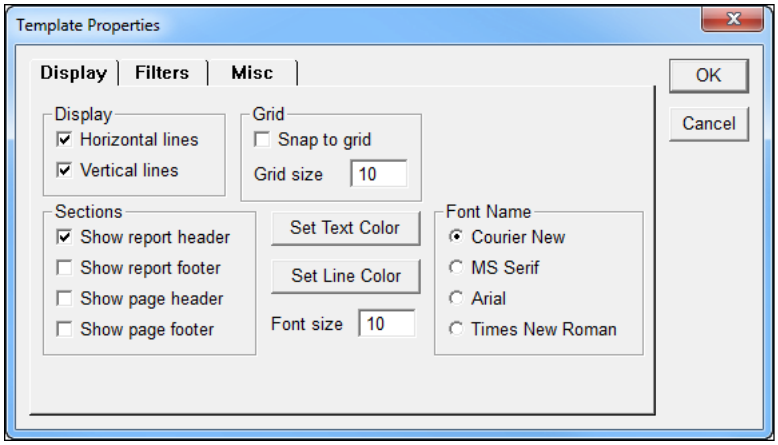


Select a report template from the list and click the **Open** button

- Right-click on the report template and select **Template Properties**
- Select **Show report header** in the **Sections** group and click **OK**

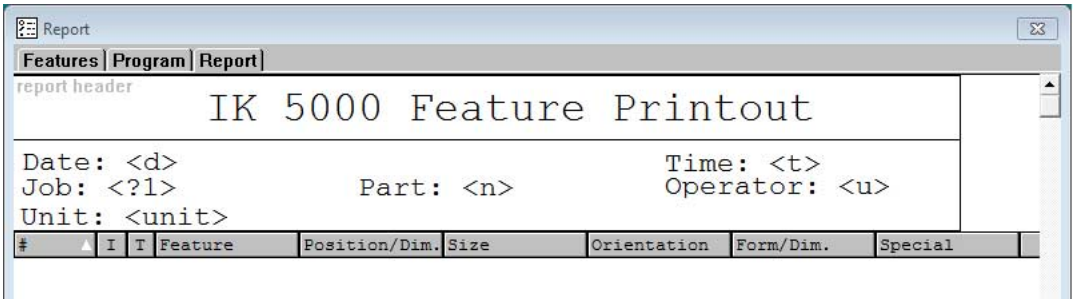


Right-click on the report template and select **Template Properties**



Select **Show report header** in the **Sections** group

The report template now shows the header with **Unit: <unit>** displayed.



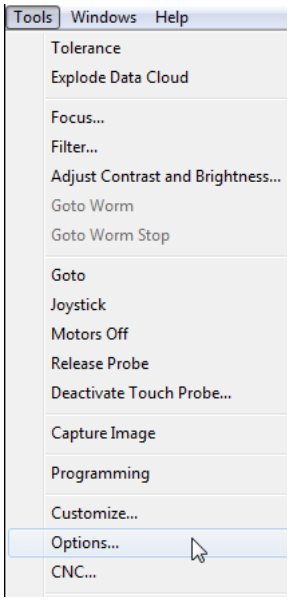
The report template now shows the header with **Unit: <unit>** displayed.

Configuring an Indexible Probe System

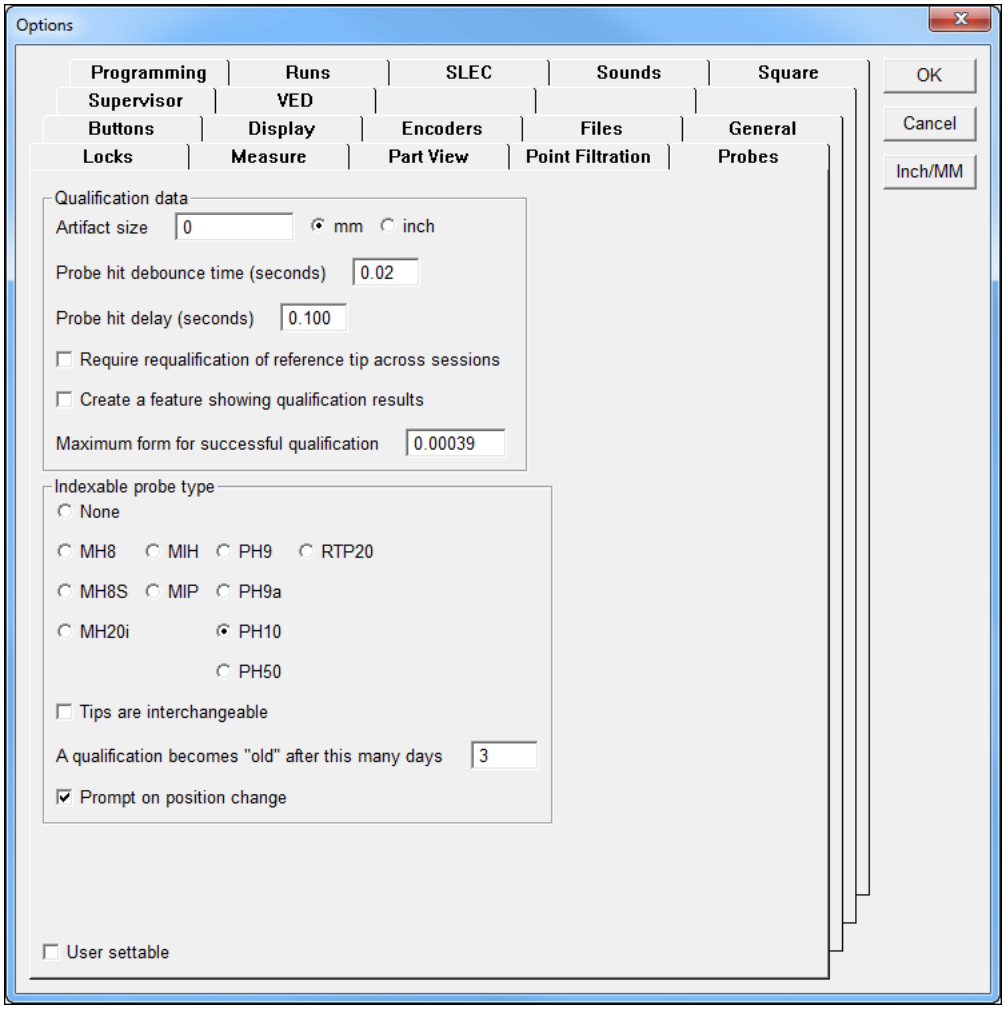
Configuring a system with an indexible contact probe head is done by turning on the **Indexible Probe Head** option and enabling the **Probe Selector**.

To configure the indexible system:

- Click **Tools>Options…**
- Select the **Probes** tab and enable the installed **Indexible probe type**



Click **Tools>Options…**



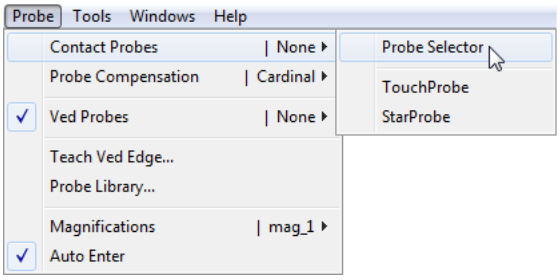
Select the **Probes** tab and enable the installed **Indexible probe type**

- Click **OK**

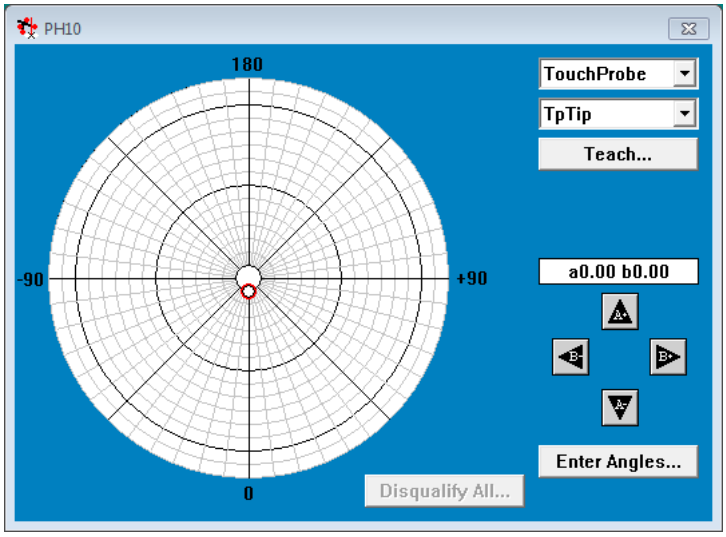
The **Probe Selector** window allows you to teach different probe angles.

To enable the probe selector window:

- Click **Probe>Contact Probes>Probe Selector**
- Stretch the probe selector window in order to access all of the controls if neccessary



Click **Probe>Contact Probes>Probe Selector**



Stretch the probe selector window in order to access all of the controls if neccessary

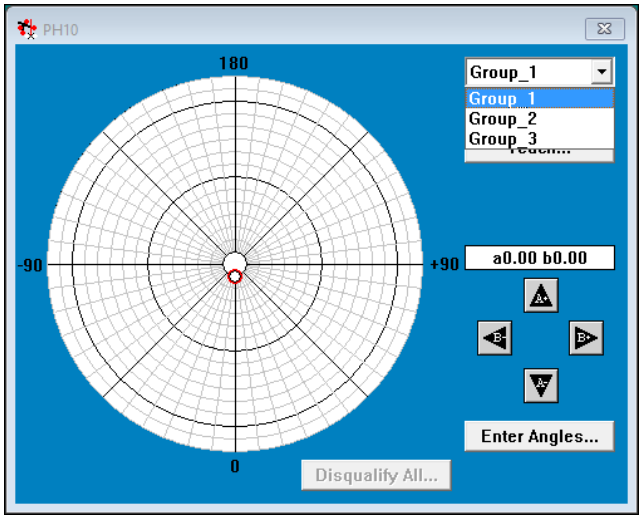
Adding probe groups and their probe tips is done in the probe library. Refer to the operating instructions for adding probe groups and tips.

Teaching (Calibrating) a Probe Tip Angle Position for CNC Systems with an Indexible Probe

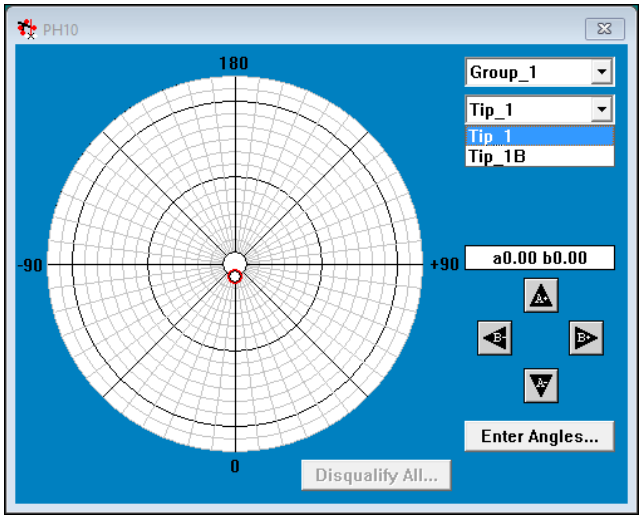
The following example assumes that valid data has been entered into the **Tools>CNC… Auto-Requal** tab and that groups and tips have been added in the **Probe Library**.

To teach an angle position for an indexible position:

- Select a group from the group drop-down list
- Select a probe tip from the probe tip drop-down list



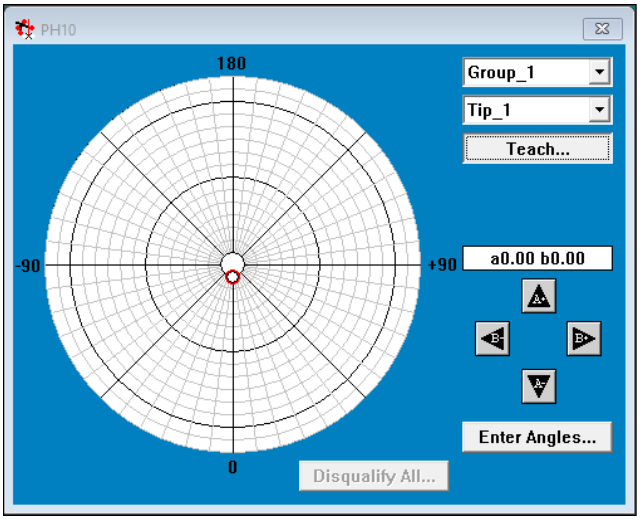
Select a group



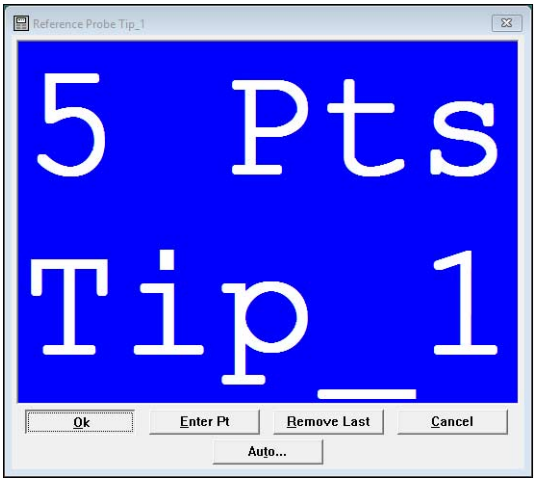
Select a probe tip

- Set the tip to the straight-down position (A0.00-B0.00, this is the reference position) by clicking on that position on the cobweb of the **Probe Selector** window (example A0.00-B0.00) or by repeatedly clicking on the **A+-** and **B+-** buttons
  The system will automatically switch to this probe indexible position if the indexible probe head is motorized. Otherwise the system will prompt you to manually switch to this probe indexible position.

- Manually teach this indexible position by probing a qualification sphere
- Once done probing the qualification sphere, click the **OK** button to complete the teach process

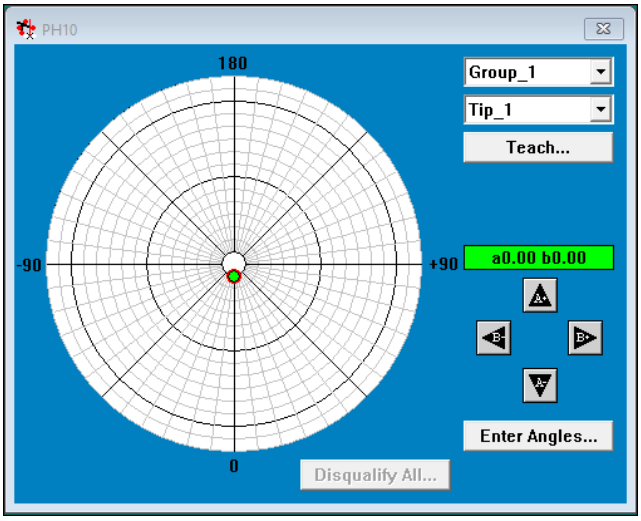


Click the position on the cobweb (example A0.00-B0.00) or repeatedly click on the **A+-** and **B+-** buttons



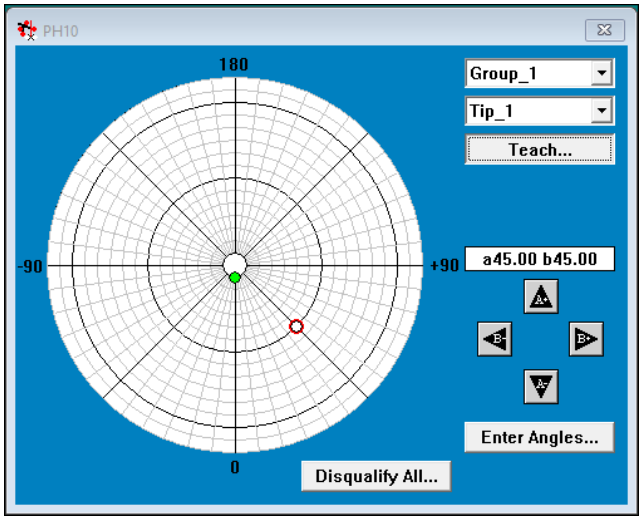
Click the **OK** button to complete the teach process

Once qualified, the position indicator will turn green.

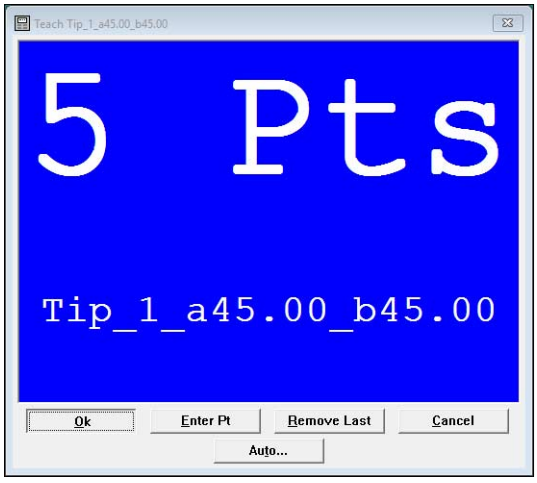


If the Stack length for the selected group is equal to zero, the system will calculate it and automatically add it. If you manually add a value that is greater than zero, the system will use that value.

- Select an indexible position by clicking on that position on the cobweb of the **Probe Selector** window (example A45.00-B45.00), or by repeatedly clicking on the **A+-** and **B+-** buttons
  If the indexible probe head is motorized, the system will automatically switch to this probe indexible position (Otherwise, the system will prompt you to manually switch to this probe indexible position).
- Teach this position by probing the qualification sphere that was used to qualify the reference position
  You can probe it manually or automatically. Automatic probing is done by clicking the **Auto…** button in the qualification window.
- Once done probing the qualification sphere manually, click the **OK** button to complete the teach process
  If done automatically, the system will complete the teach process automatically.

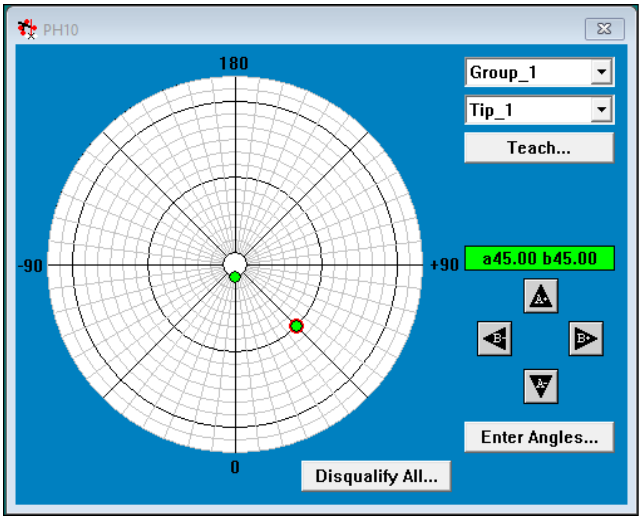


Click the position on the cobweb (example A45.00-B45.00) or repeatedly click on the **A+-** and **B+-** buttons



Click the **OK** button to complete the teach process

Once qualified, the position indicator will turn green.

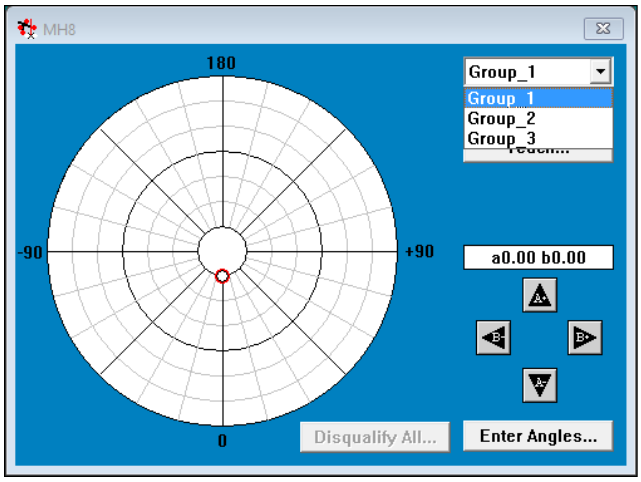


- The group Stack length can be reset by going to the Probe Library and entering a value of zero for that group. The next time the reference position for the reference tip of that group is qualified manually, a new Stack length will be calculated and automatically added.
- The system will not automatically change a group's Stack length if you manually enter a value that is greater than zero.
- The qualification sphere position for automatically probing can be set to a new location by moving the qualification sphere to the new location and then manually re-teaching the reference position of the reference tip of the reference group.
- The reference position can be re-qualified using automatic qualification sphere probing if the group Stack length is non-zero.
- Non-reference group’s Stack length will be calculated and added in automatically once a group has been cross-calibrated with the reference group.

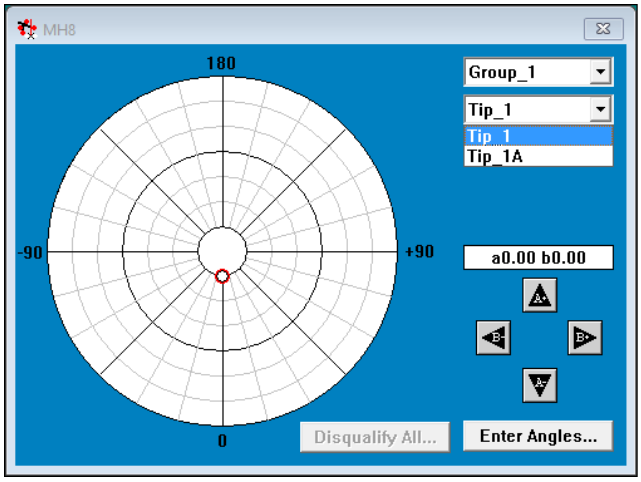
Teaching (Calibrating) a Probe Tip Angle Position for a Non-CNC System With an Indexible Probe

To teach an angle position for an indexible position:

- Select a group from the group drop-down list
- Select a probe tip from the probe tip drop-down list



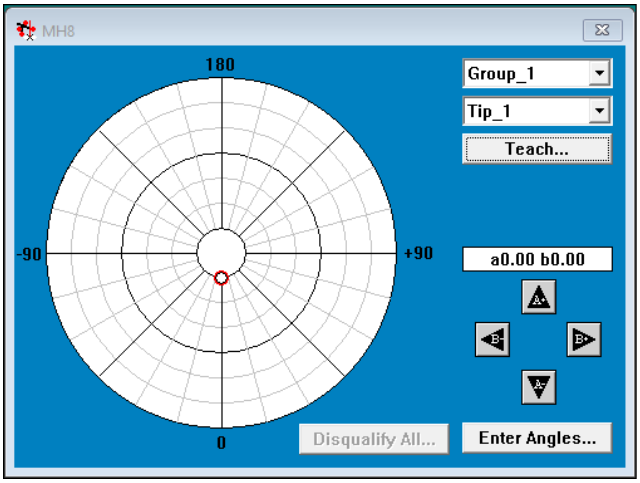
Select a group



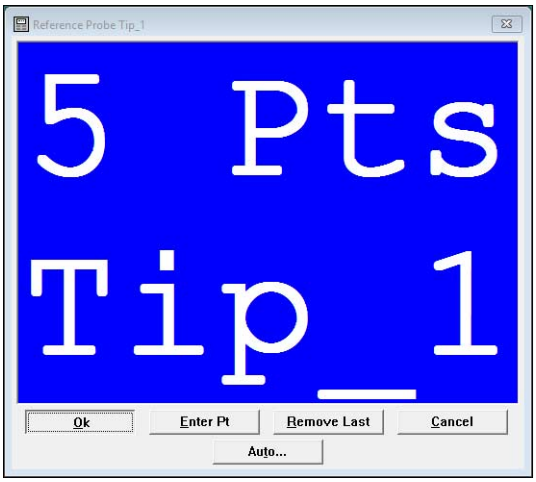
Select a tip

- Set the tip to the straight-down position (A0.00-B0.00, this is the reference position) by clicking on that position on the cobweb of the **`Probe Selector`** window (example A0.00-B0.00), or by repeatedly clicking on the **`A+-`** and **`B+-`** buttons.
  The system will prompt you to manually switch to this probe indexible position.
- Teach this indexible position by probing a qualification sphere

- Once done probing the qualification sphere, click the **OK** button to complete the teach process

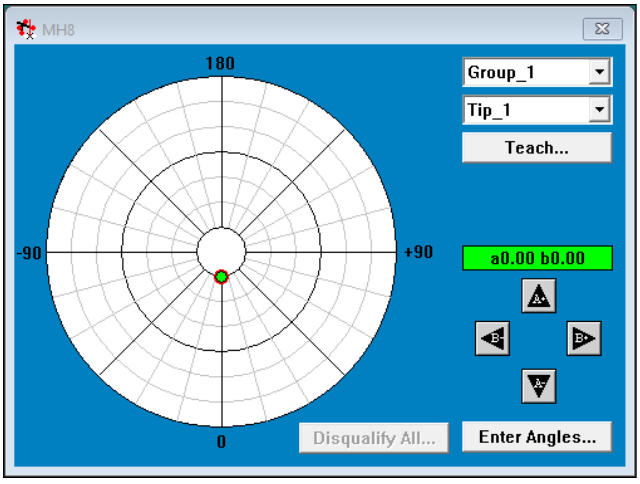


Click the position on the cobweb (example A0.00-B0.00) or repeatedly click on the **A+-** and **B+-** buttons

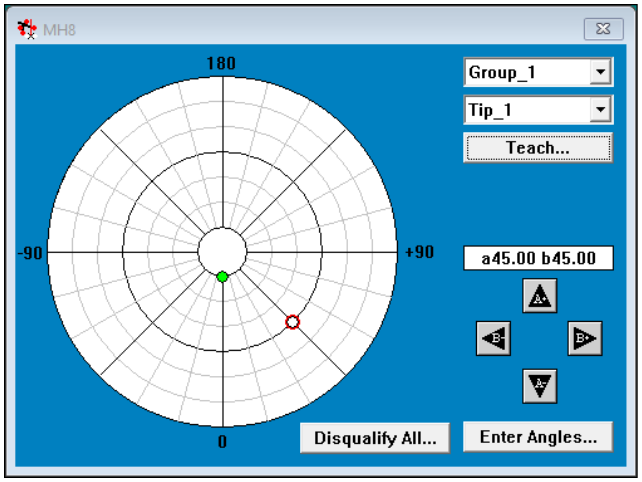


Click the **OK** button to complete the teach process

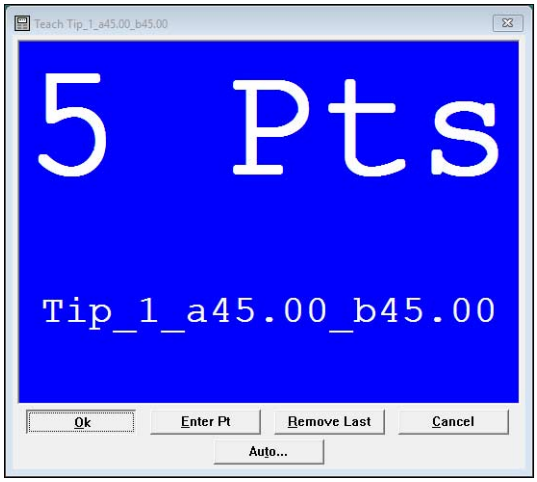
Once qualified, the position indicator will turn green.



- Select an indexible position by clicking on that position on the cobweb of the **Probe Selector** window (example A45.00-B45.00), or by repeatedly clicking on the **A+-** and **B+-** buttons
  The system will prompt you to manually switch to this probe indexible position.
- Teach this indexible position by probing the qualification sphere that was used to qualify the reference position
- Once done probing the qualification sphere, click the **OK** button to complete the teach process

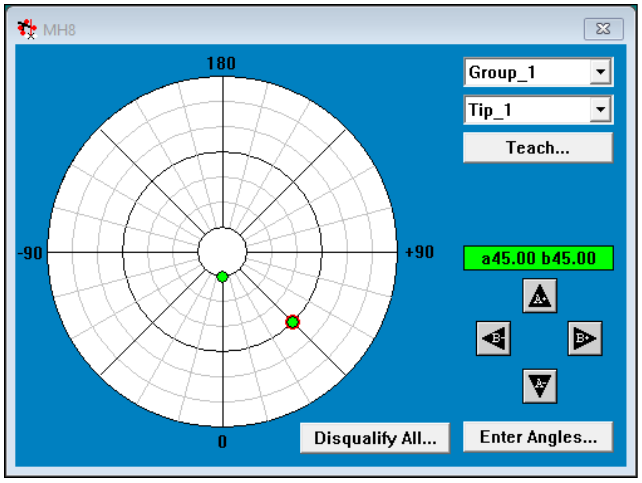


Click the position on the cobweb (example A45.00-B45.00) or repeatedly click on the **A+-** and **B+-** buttons



Click the **OK** button to complete the teach process

Once qualified, the position indicator will turn green.

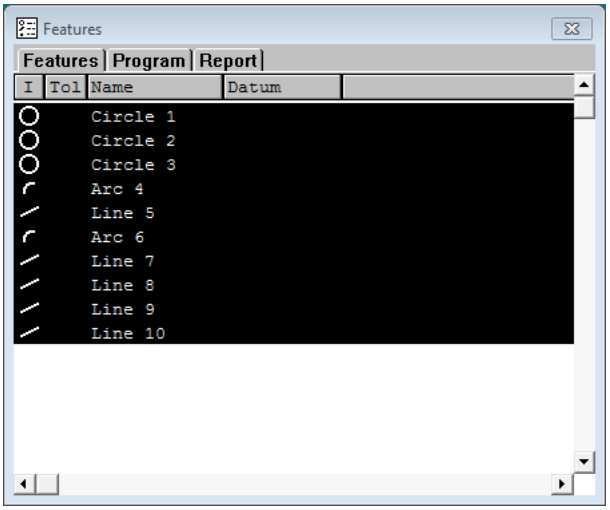


Setting the Circle Fit Algorithm during Auto Programming

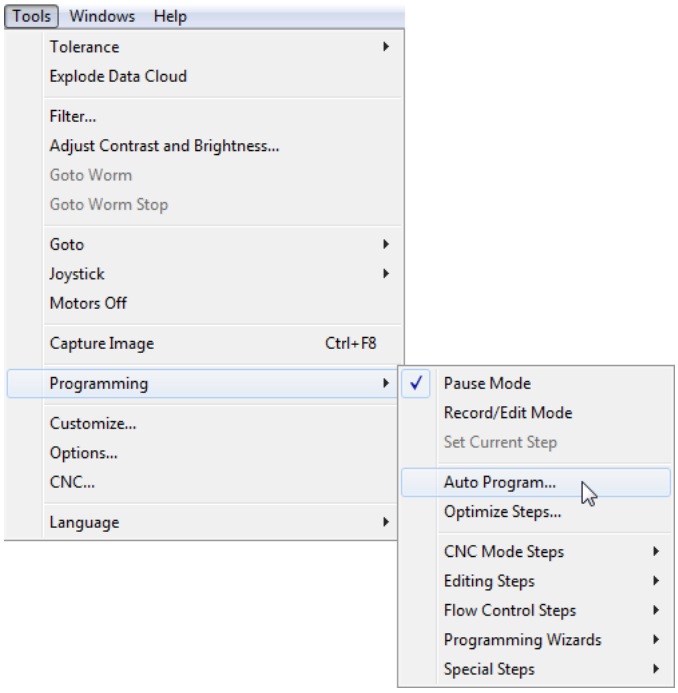
A circle fit algorithm can be set during auto programming to be used when a program is played back.

To set the fit algorithm:

- Select the features to be auto programmed from the feature list
- Click **`Tools >Programming>Auto Program…`**

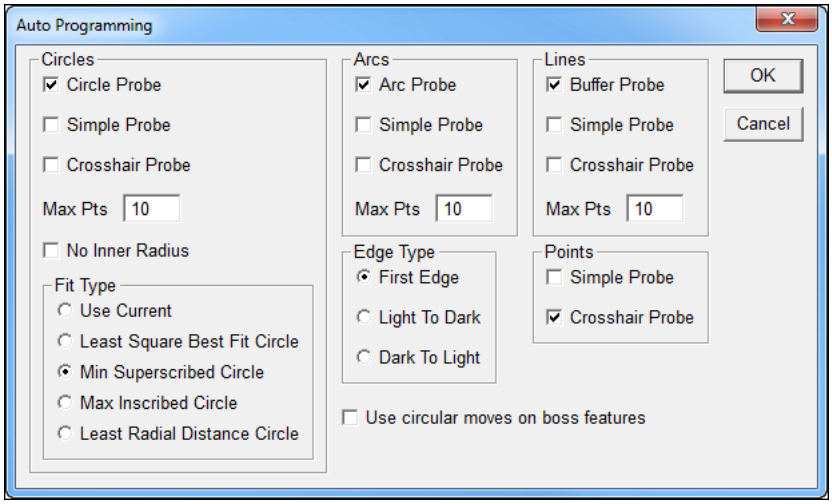


Select the features to be auto programmed from the feature list



Click **`Tools >Programming>Auto Program…`**

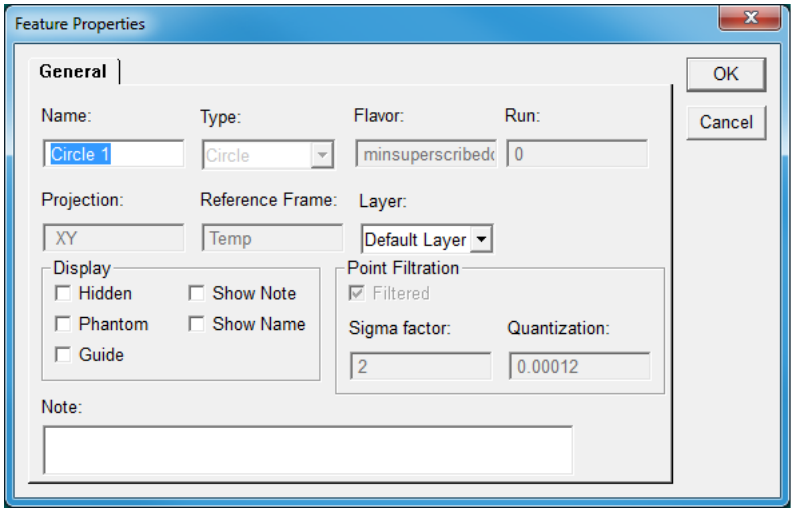
- Select the desired fit type under the **Circles** group, **Fit Type** subgroup



Select the desired fit type under the **Circles** group, **Fit Type** subgroup

- Click **OK**

Notice that the circles are measured using the selected fit type after playing back the program.

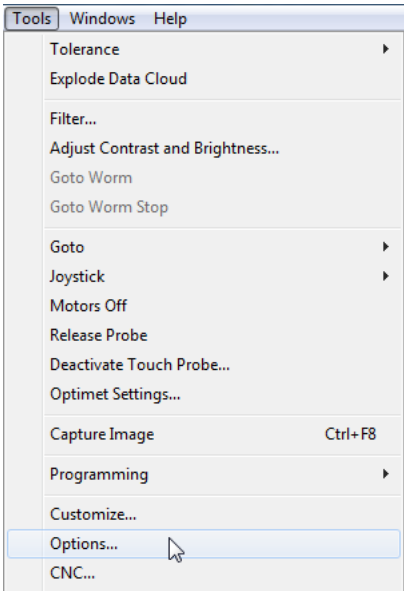


Configuring a Laser Pointer

A laser pointer can be connected to the IK 5000 to be used as an input.

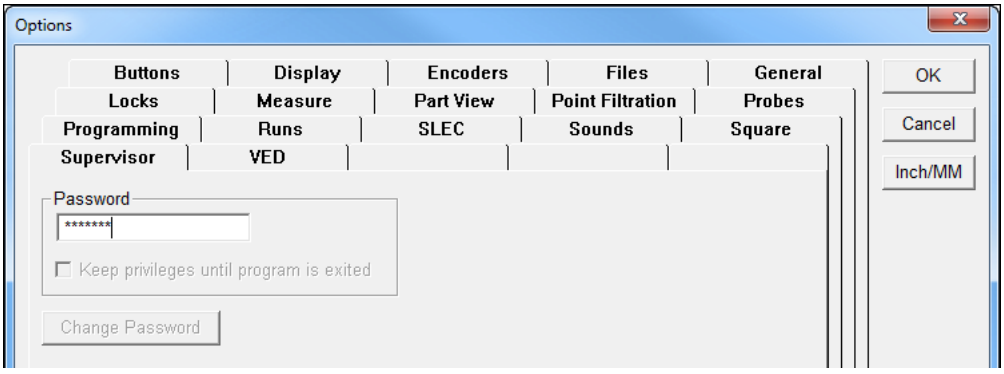
To configure the laser pointer:

▶ Click **Tools>Options…**



Click **Tools>Options…**

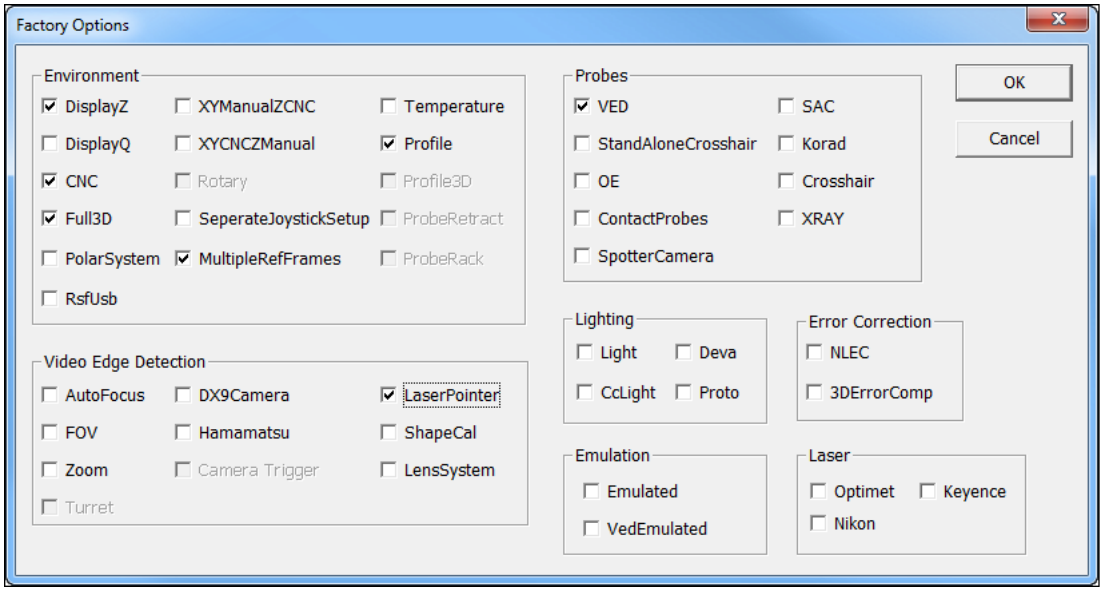
▶ Select the **Supervisor** tab and enter "Options" in the password field



Select the **Supervisor** tab and enter "Options" in the password field

The **CNC** option or the **CcLight** option must be selected for the **LaserPointer** option to be selectable.

▶ Enable the **LaserPointer** option



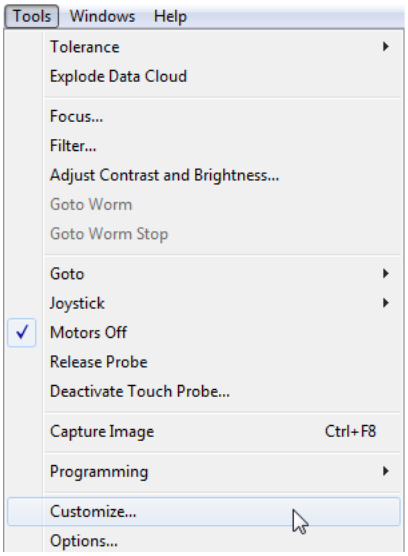
Enable the **LaserPointer** option

▶ Click **OK**

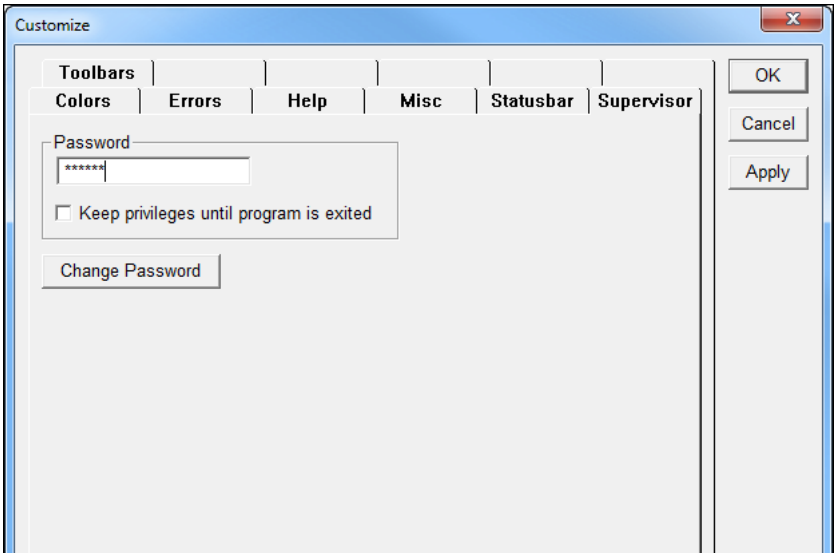
The laser pointer can be enabled or disabled with a button mapped to a toolbar.

To add the enable/disable **LaserPointer** button to a toolbar:

- Click **Tools>Customize...**
- Select the **Supervisor** tab and enter the supervisor password in the **Password** field

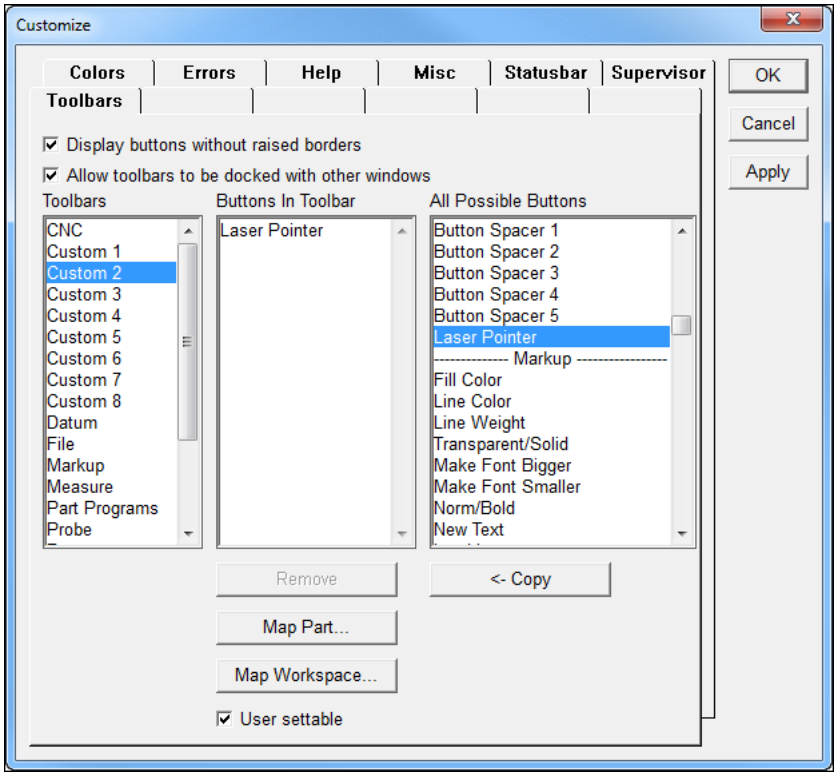


Click **Tools>Customize...**



Select the **Supervisor** tab and enter the supervisor password in the **Password** field

- Select the **Toolbars** tab
- Select a toolbar in the **Toolbars** column
- Select **LaserPointer** in the **All Possible Buttons** column
- Click **<- Copy** to copy the button to the selected toolbar

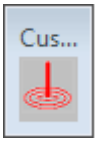


Click **<- Copy** to copy the button to the selected toolbar

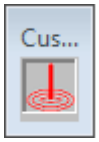
- Click **OK**

- Display the toolbar that the **LaserPointer** was mapped to, if it is not already displayed

Notice the button in the Off state and On state when the button is toggled.



LaserPointer
Off



LaserPointer
On

The Laser Pointer uses the first lamp enable line output; connector N, pin 3, **Lamp en 1**.

The first lamp enable line is not available for use with light control while the **LaserPointer** option is active.

Refer to the installation instructions for the lamp enable connector pin assignments.

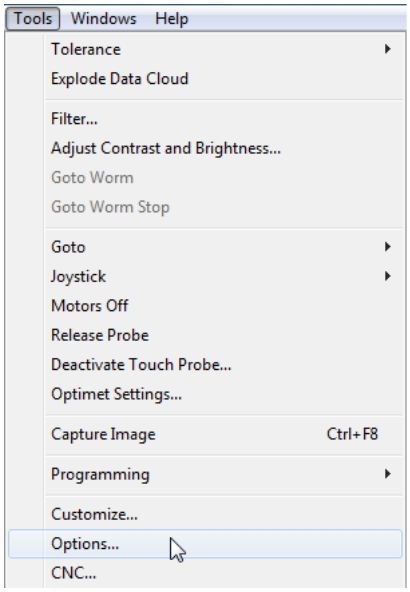
Waiting for an External Input Line Program Step

A program can be recorded with a step to wait for an external input line. The input line can be high (1) or low (0).

The **CNC** software option is required in order to use the **Wait For Input Line** feature.

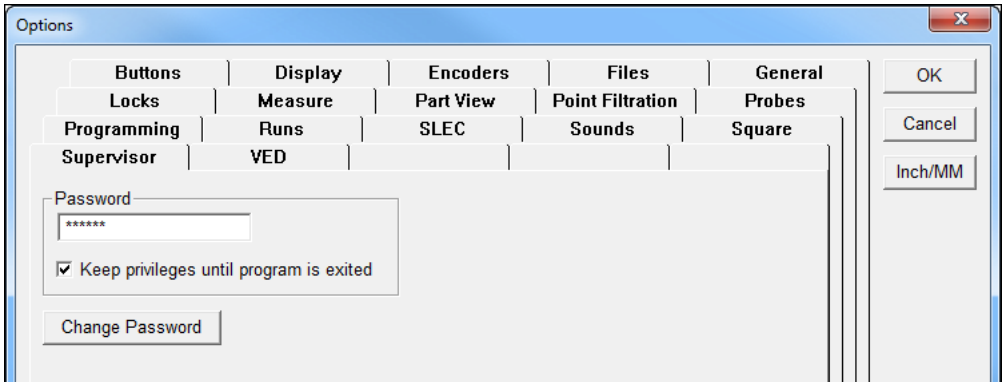
To record this step into a program:

- Click **Tools>Options...**



Click **Tools>Options...**

- Enter the supervisor password in the **Password** field



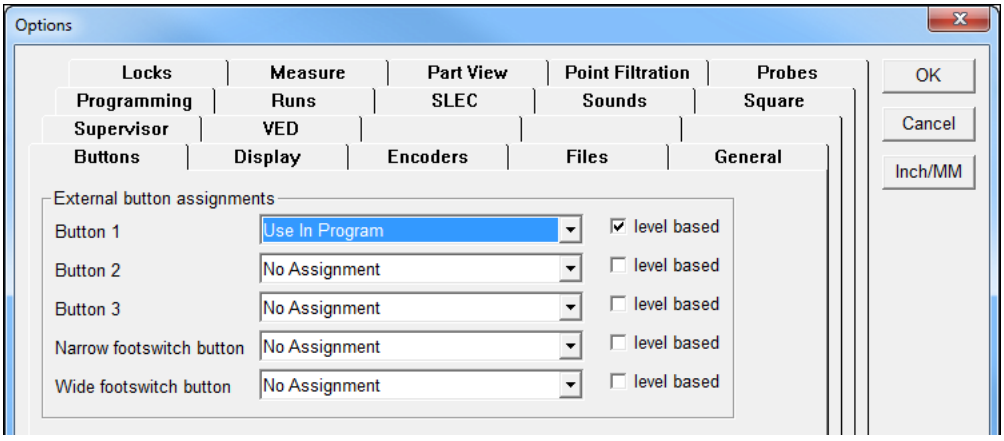
Enter the supervisor password in the **Password** field

- Select the **Buttons** tab
- Assign an external button input by selecting **Use In Program** from the drop-down menu and checking the **level based** checkbox

Button	Connector		Pin	Assignment
Button 1	B[1)]	I[2)]	15	Button 1 TTL in
Button 2	B[1)]	I[2)]	6	Button 2 TTL in
Button 3	B[1)]	I[2)]	24	Button 3 TTL in
Narrow footswitch button	A[1)]	H[2)]	30	Foot switch 1
Wide footswitch button	A[1)]	H[2)]	15	Footswitch 2

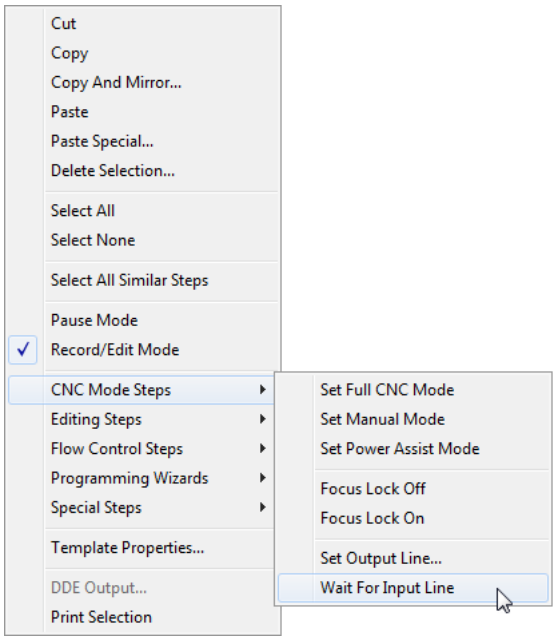
[1)] IK 5000 QUADRA-CHEK Installation Instructions (ID 1035034-9x)

[2)] IK 5000 QUADRA-CHEK Installation Instructions (ID 1108935-9x)



Select the **Buttons** tab, select **Use In Program**, and check **level based**

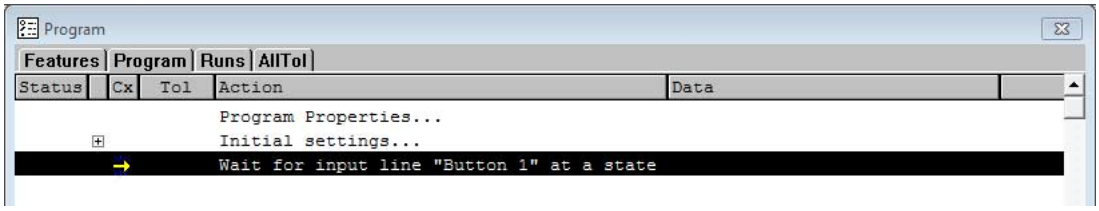
- While recording a program, right-click in the program template window and select **CNC Mode Steps>Wait For Input Line**



Right-click in the program template window and select **CNC Mode Steps>Wait For Input Line**

A new program step **Wait for input line <button> at a state** is added to the program.

When playing back the program, the program will wait until the state of **<button>** changes when the program step **Wait for input line <button> at a state** is encountered. A CNC option is required for this program step to be available.



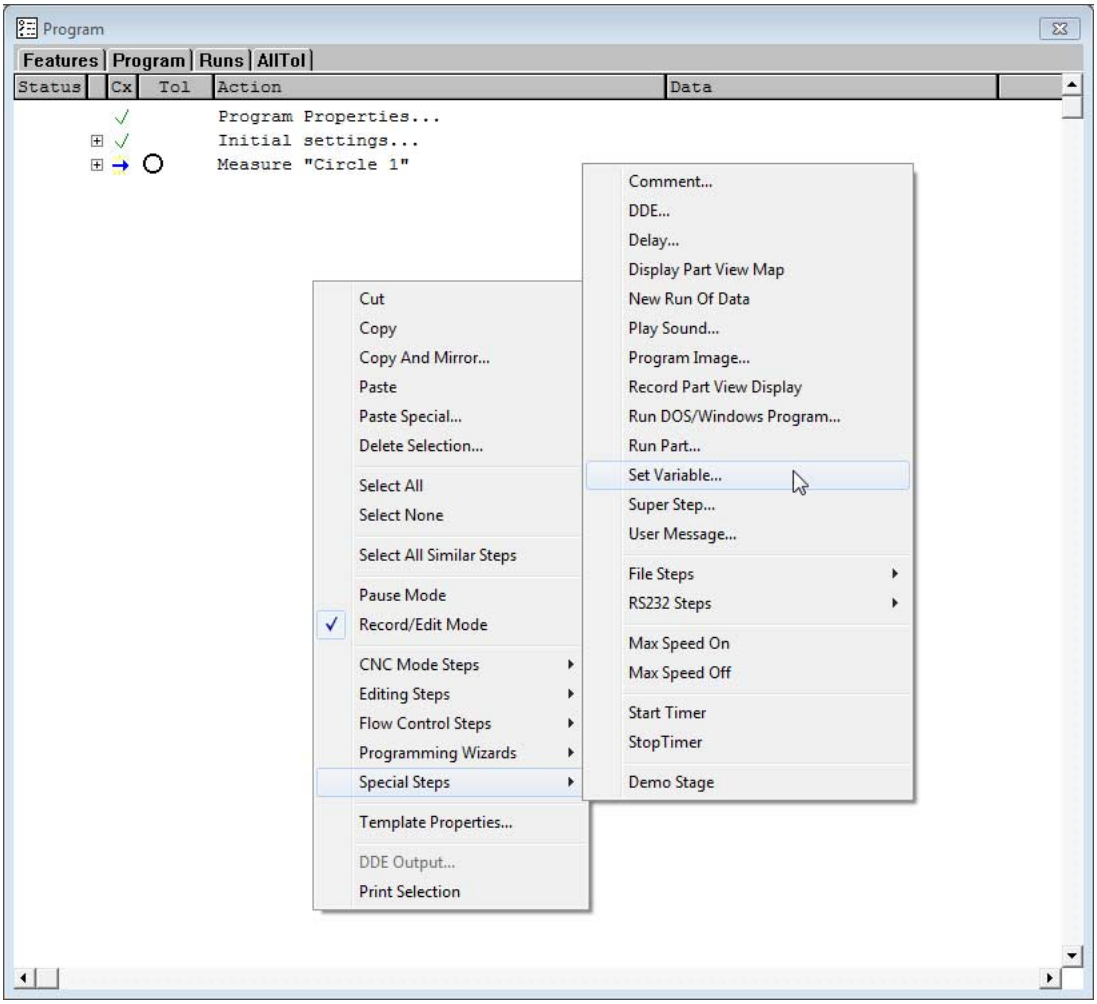
A new program step **Wait for input line <button> at a state** is added to the program.

Creating and Using Global Variables

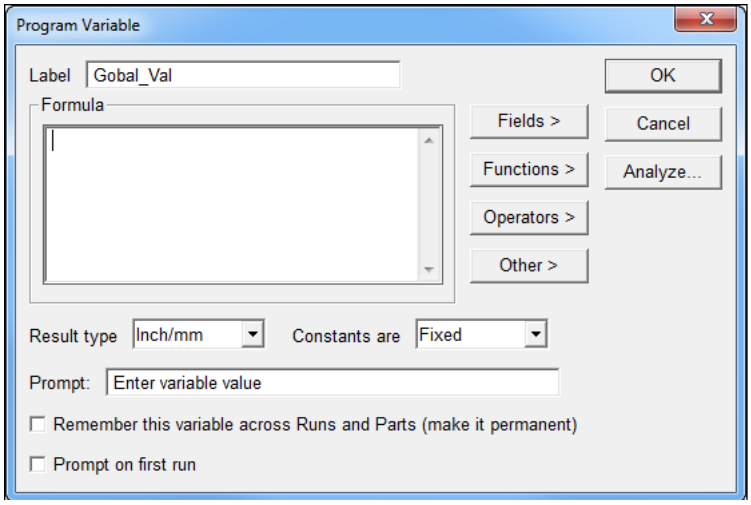
A program variable can be set to be used in multiple programs. It can be set to be edited every time the program is run, only the first time the program is run, or only once.

To set program variable options:

- Record a program
- Right-click in the **Program** template and select S**pecial Steps>Set Variable…** to add a program variable



- Enter the variable **Label**, **Formula**, and **Prompt**
- Select the **Result Type** and **Constant Type**
- Check the **Remember this variable across Runs and Parts (make it permanent)** or **Prompt on first run** or both options to set the desired option for the variable



Select the options for the variable

Cross-calibrating Probe Families, Groups/Cameras, and Tips/Magnifications

The IK 5000 can cross-calibrate between probe families (in a multi-sensor system), different groups/cameras in a probe family, and various probe tips/magnifications within a group/camera.

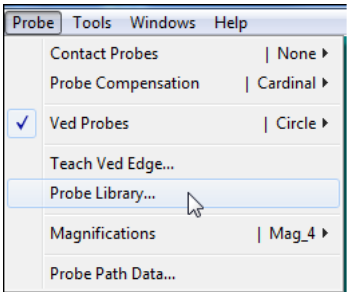
Any probe family (in a multi-sensor system), group/camera (in a probe family), or probe tip/magnification (in a group/camera) can be assigned to be the reference.

A multi-sensor system with Video Edge Detection (VED) and Contact Probe family of probes is an example. The system may be configured with one camera (Camera_1) for VED with two magnifications (Mag_1 and Mag_2), and two groups (Group_1 and Group_2) for Contact Probe with two probe tips each (Tip1_G1, Tip2_G1, Tip1_G2, Tip2_G2). Each level is automatically assigned a reference. This can be changed by going to the probe library and selecting a different item to be the reference.

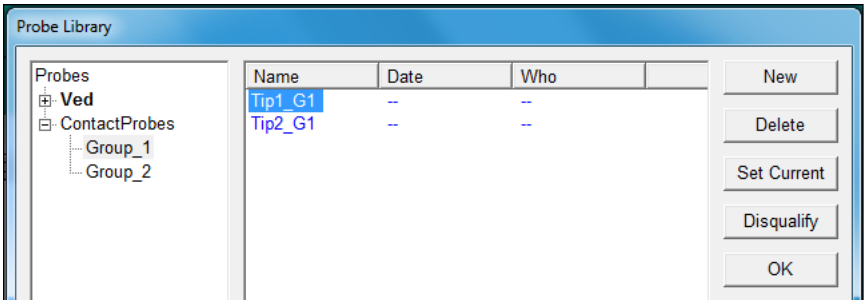
Example: Change the Reference of a Probe Tip from Tip1_g1 to Tip2_g1

To change the reference of a probe tip:

- Click **Probe>Probe Library…**
- Select **Contact Probe** on the left side and double-click on **Group_1** to show probe tips Tip1_G1 and Tip2_G1

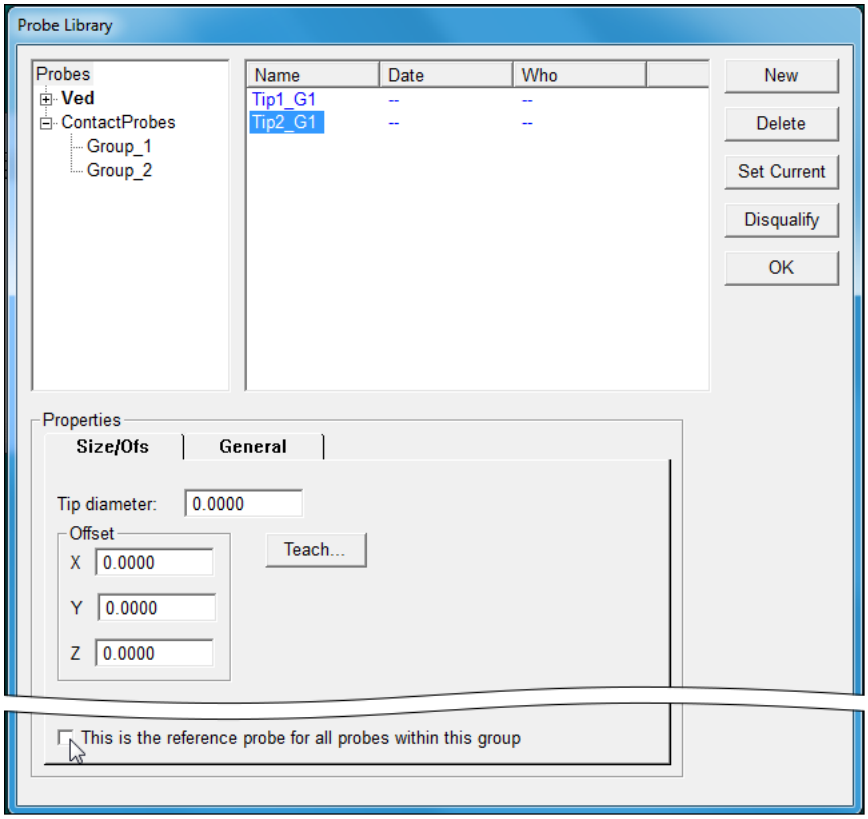


Click **Probe>Probe Library…**



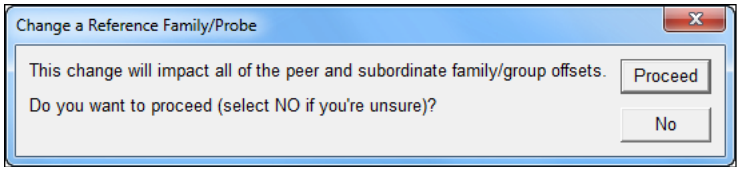
Select **Contact Probe** on the left side and double-click on **Group_1** to show probe tips Tip1_G1 and Tip2_G1

- Select the **Size/Ofs** tab of **Tip2_G1** and check **This is the reference probe for all probes within this group**



Select the **Size/Ofs** tab and check **This is the reference probe for all probes within this group**

- Click **Proceed** in the Change a Reference Family/Probe window



Click **Proceed**

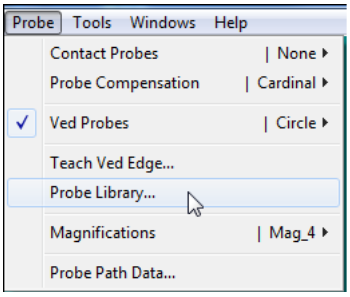
- Click **OK** in the Probe Library window to save the changes and exit the window

Example: Set the Contact Probes to be the Reference Family

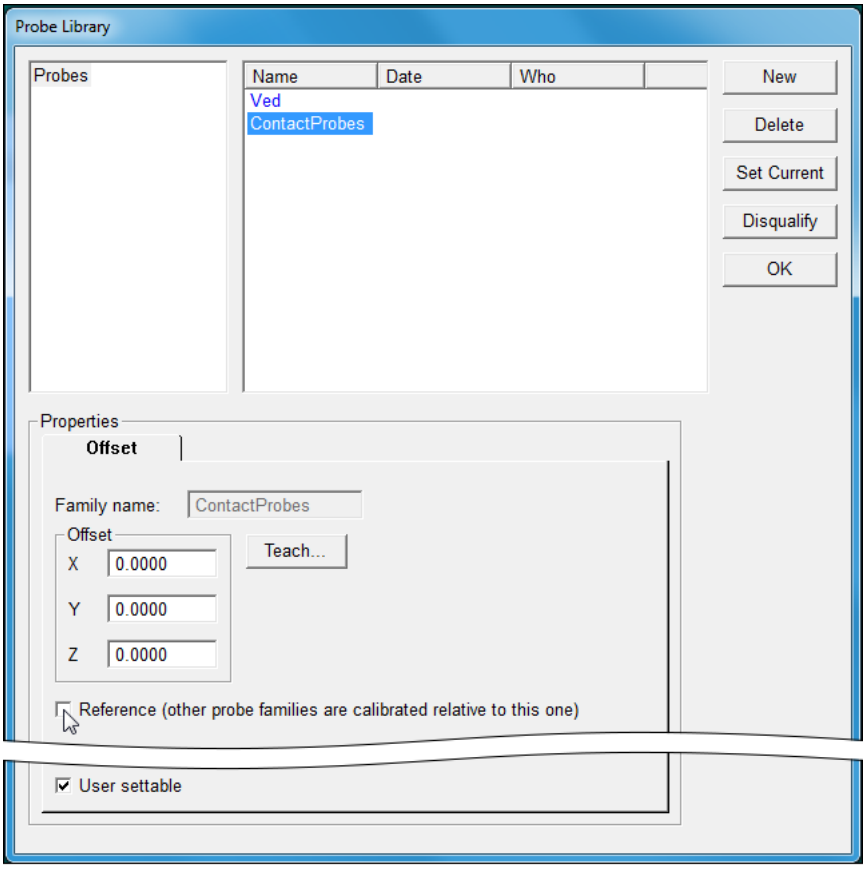
To cross-calibrate between probe families:

- Click **Probe>Probe Library…**
  The Probe Library window opens.
- Select **Probes** on the left and then select **ContactProbes** on the right
- Check **Reference (other probe families are calibrated relative to this one)**

At every level of the probe hierarchy, there must be a reference item. There must be a reference family, reference group/camera, and reference probe tip/ magnification.

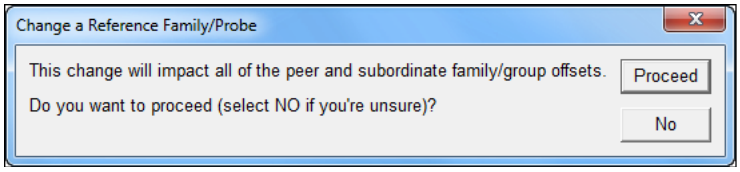


Click **Probe>Probe Library…**



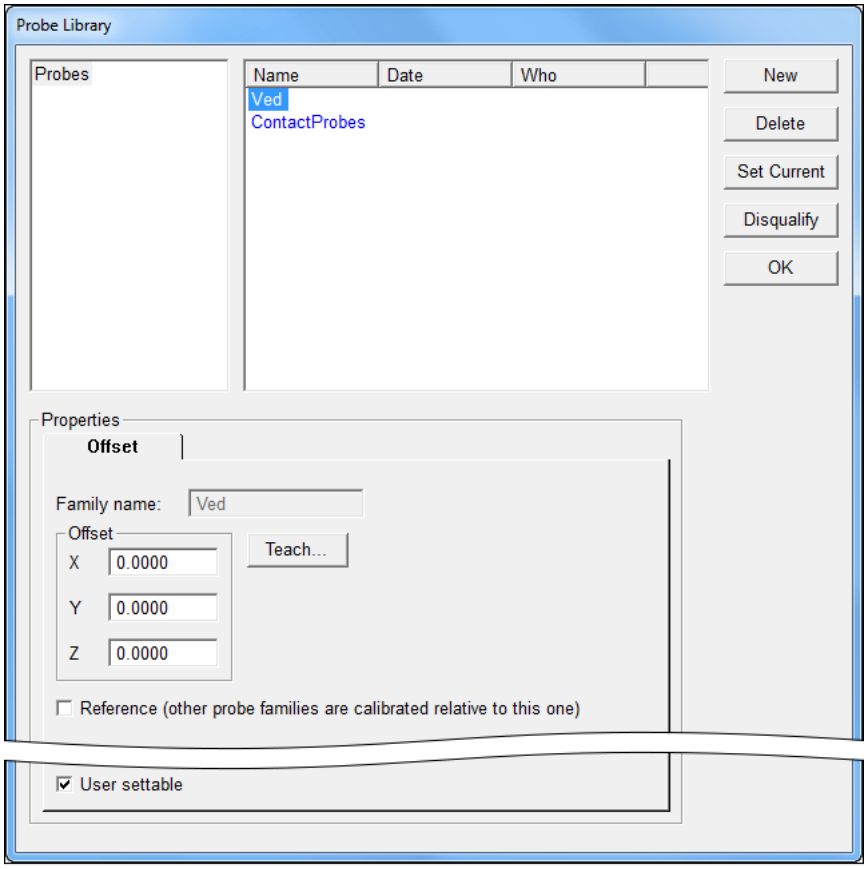
Select **Probes**, then **ContactProbes** and check **Reference (other probe families are calibrated relative to this one)**

- Click **Proceed** in the Change a Reference Family/Probe window



Click **Proceed**

- Select **Probes** on the left side and then select **Ved** on the right side
- Select the **Offset** tab and Click **Teach**



Select **Probes**, then **Ved**, Select the **Offset** tab and click **Teach**...

- Follow the prompts until the cross-calibration is completed.
  If the cross-calibration is successful, the system will insert values (most likely non-zero values) in the XYZ offset group box.
- Click **OK** in the Probe Library window to save the changes and exit

Master User Profile

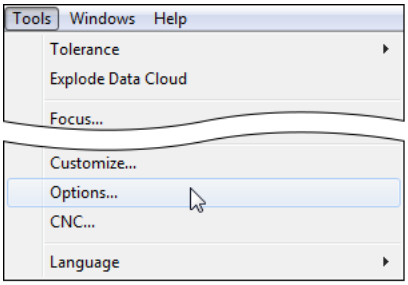
Creating the Master User Profile

A supervisor can create an IK 5000 master user profile that can be loaded for different Windows OS users. When this master user profile is loaded, it gives the IK 5000 a uniform look as designed when the profile was saved.

The profile is saved to the IK 5000 folder located in the ProgramData system folder.

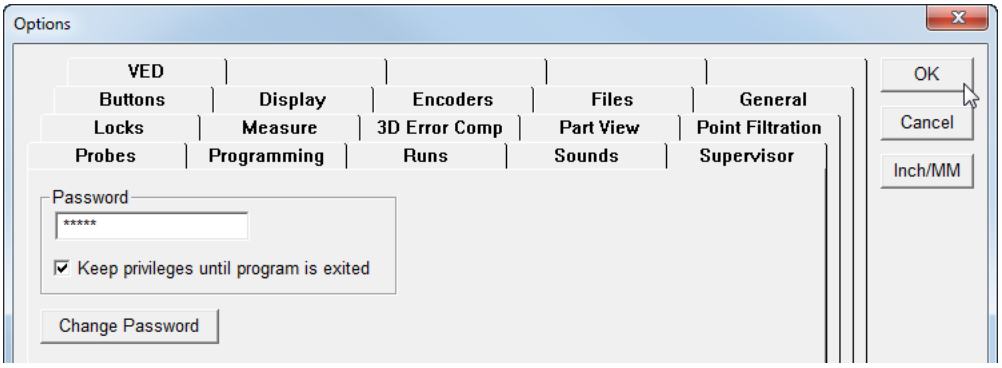
To create the master user profile:

- Click **Tools>Options...**



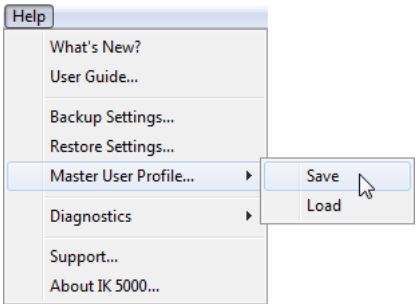
Click **Tools>Options…**

- Select the **Supervisor** tab and enter the supervisor password in the **Password** field
- Check **Keep privileges until program is exited**
- Click **OK** to save the changes and exit the **Options** window

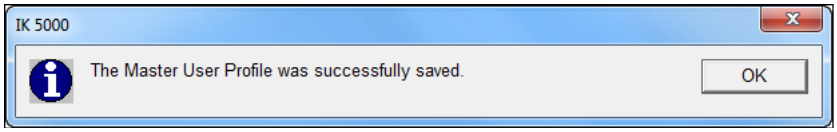


Select the **Supervisor** tab, enter the supervisor password, check **Keep privileges until program is exited**, and click **OK**

- Click **Help>Master User Profile...>Save**
  A pop-up message is displayed indicating that a Master User Profile was saved.
- Click **OK** to close the message window



Click **Help>Master User Profile...>Save**



Click **OK**

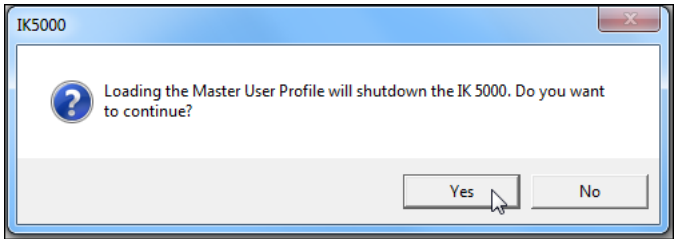
Loading the Master User Profile

To load the Master User Profile:

- Click **Help>Master User Profile...>Load**

A pop-up message is displayed indicating that loading the Master User Profile will shutdown the program.

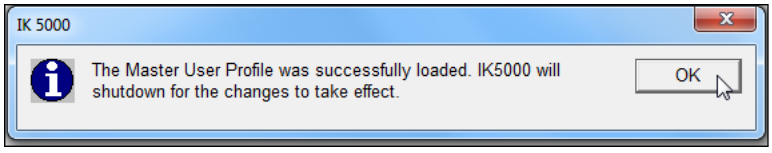
- Click **Yes**



Click **Yes**

A pop-up message is displayed indicating that the Master User Profile was loaded successfully and the program will shutdown for the changes to take effect.

- Click **OK**



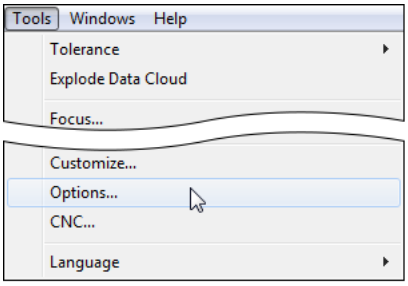
Click **OK**

Enabling Load Master User Profile at Startup

The Master User Profile can be loaded on startup for situations when a supervisor wants the user to always have the same profile at startup.

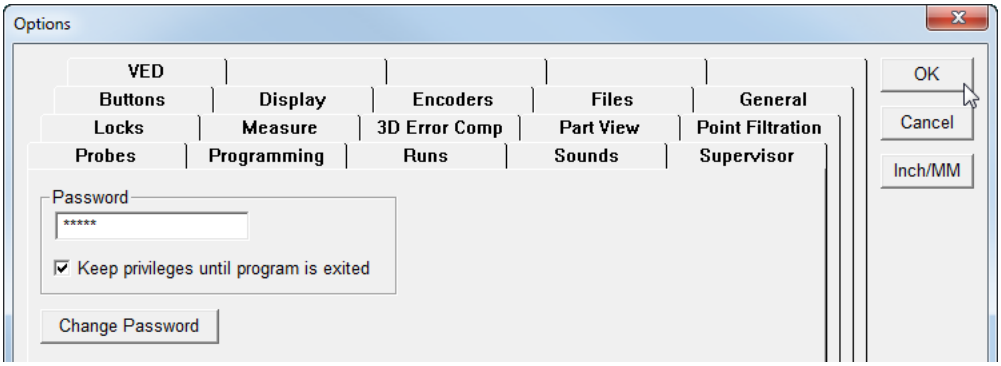
To enable **Load master user profile at startup**:

- Click **Tools>Options...**



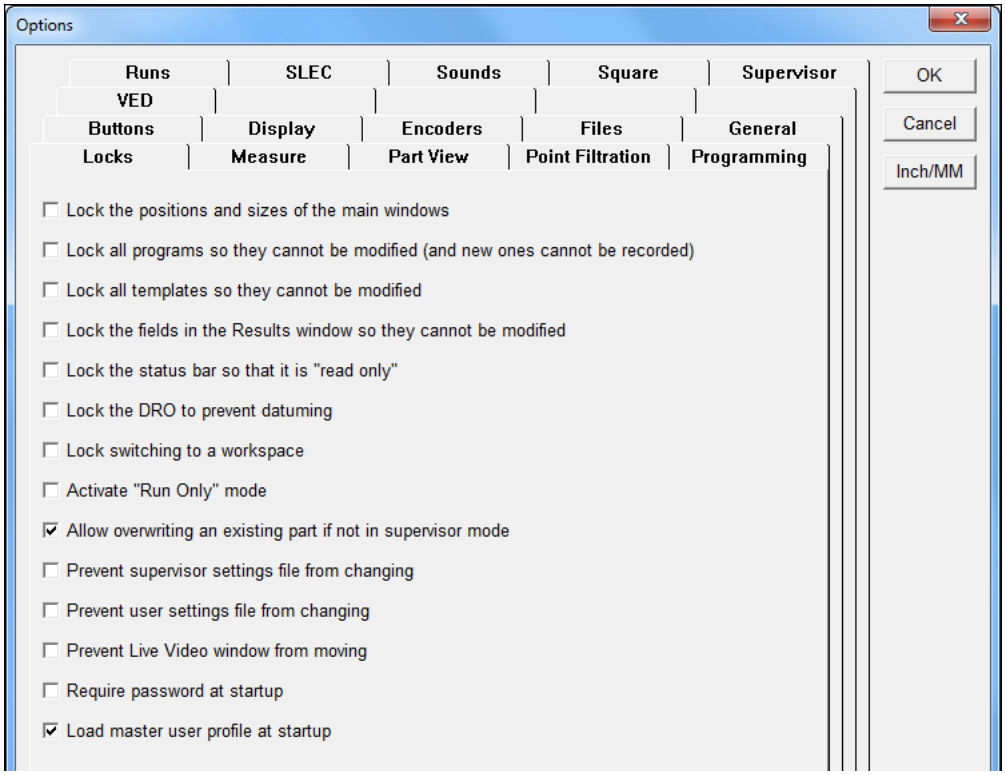
Click **Tools>Options…**

- Select the **Supervisor** tab and enter the supervisor password in the **Password** field



Select the **Supervisor** tab and enter the supervisor password

- Select the **Locks** tab
- Check **Load master user profile at startup**



Select the **Locks** tab and check **Load master user profile at startup**

- Click **OK** to save the changes and exit the **Options** window

Setting Focus Offset

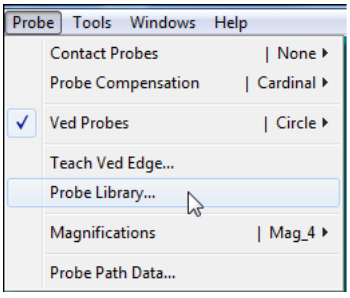
Focus Offset lets you move the focal point from the one found when performing autofocus.

For example, you may want to do this when a camera has an inbuilt delay in the posting of image frames. The target focal point will be off because of the delay. The distance that the focal point is off can be corrected by applying a focus offset.

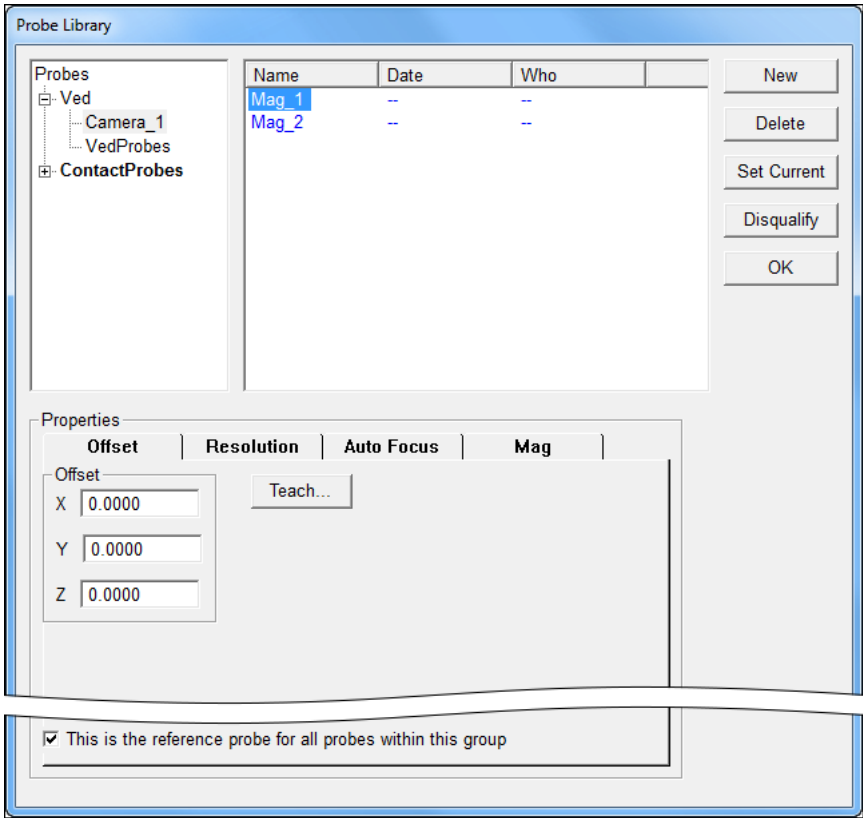
An offset can be set for each magnification of each camera in the system.

To set a Focus Offset value:

- Determine the offset value that needs to be applied to the acquired focus point for a magnification
- Click **Probe>Probe Library…**
- Select **Ved**, the desired camera, and the desired magnification

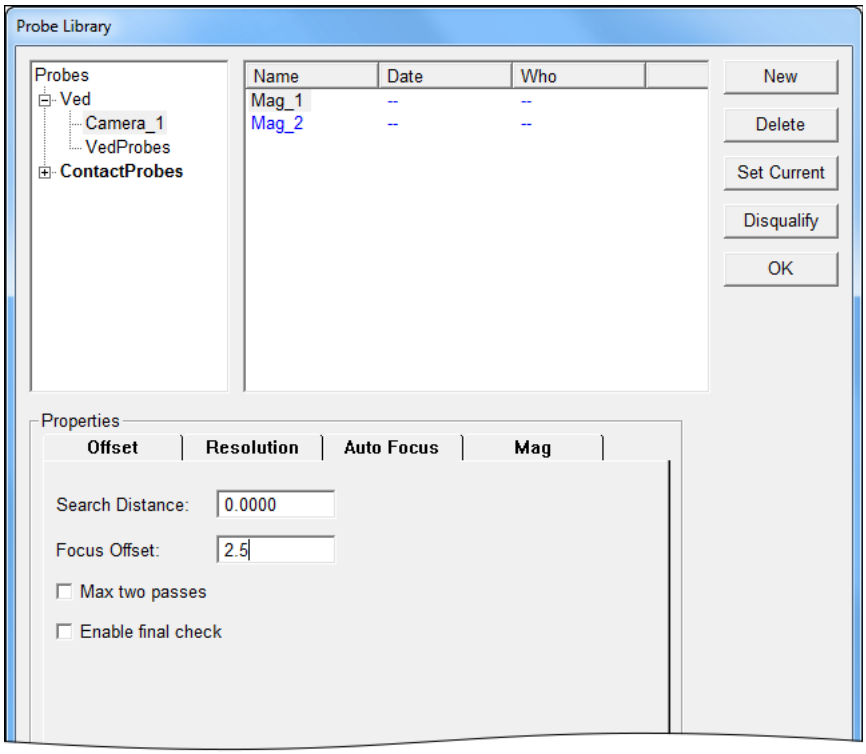


Click **Probe>Probe Library…**



Select **Ved**, the desired camera, and the desired magnification

- Select the **Auto Focus** tab and enter the offset value in the **Focus Offset** field



Select the **Auto Focus** tab and enter the offset value

- Click **OK** in the Probe Library window to save the changes and exit

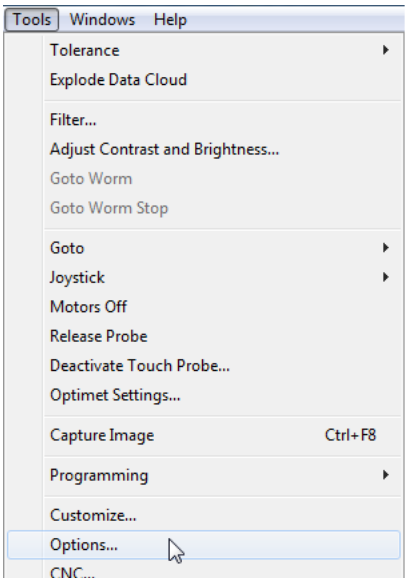
Loading an NLEC (Non-Linear Error Compensation) file

NLEC information is stored in a file in the system configuration folder.

An NLEC file must be created by an IK 5000 system or structured in the same format as an IK 5000 NLEC file.

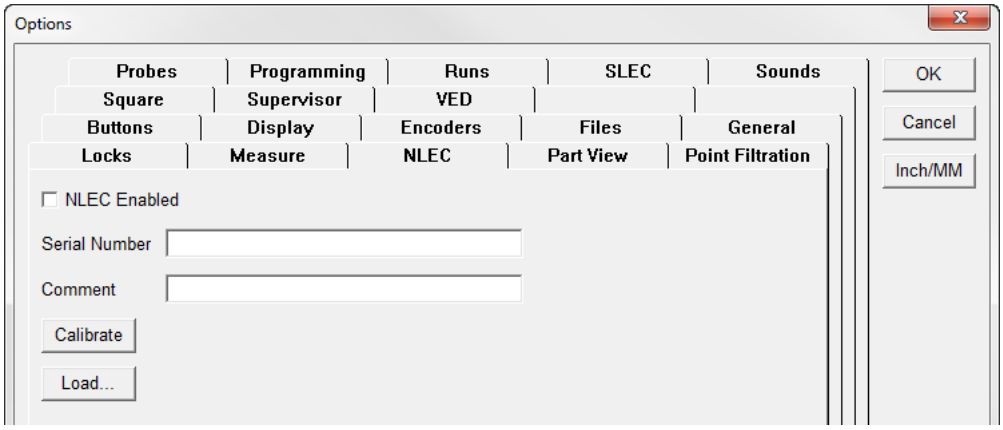
To load an NLEC file:

- Replace the NLEC.txt file in the system configuration folder with the new NLEC file. The new file must be named NLEC.txt.
- Click **Tools>Options…**



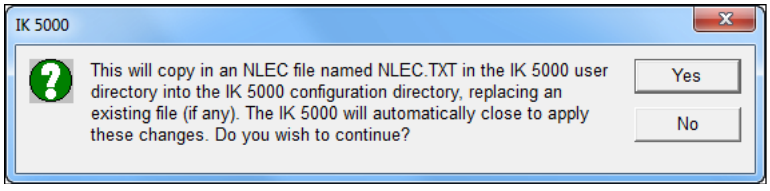
Click **Tools>Options…**

- Select the **NLEC** tab
- Verify that the **NLEC Enabled** check-box is checked
- Click **Load...**



Click **Load...**

- Click **Yes** in the prompt to confirm loading



Click **Yes**

- Browse to the system configuration folder and select the new NLEC.txt file
- Click **OK** to save the changes and exit the Options window

2D Profile Maximum Form Error

When a 2D profile feature has been measured, applying a tolerance can result in a failure if outlier data points are present.

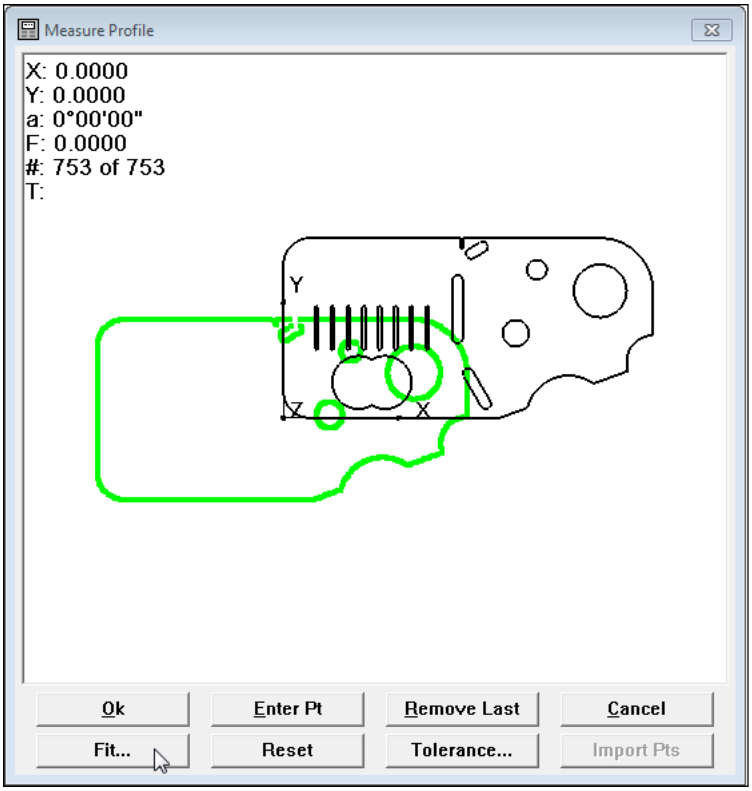
The Maximum Form Error function can be used to filter outlier data points from tolerance test consideration.

To use Maximum Form Error and filter outlier data points:

- Start measuring a 2D profile feature

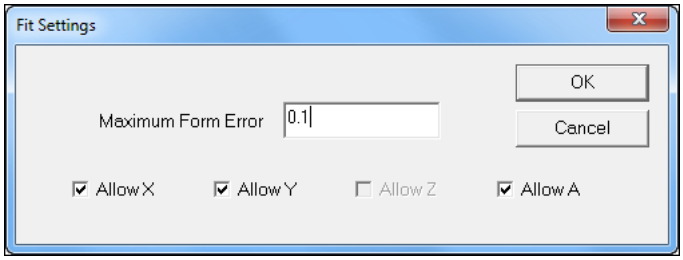
While in the Measure Profile screen:

- Click **`Fit...`**



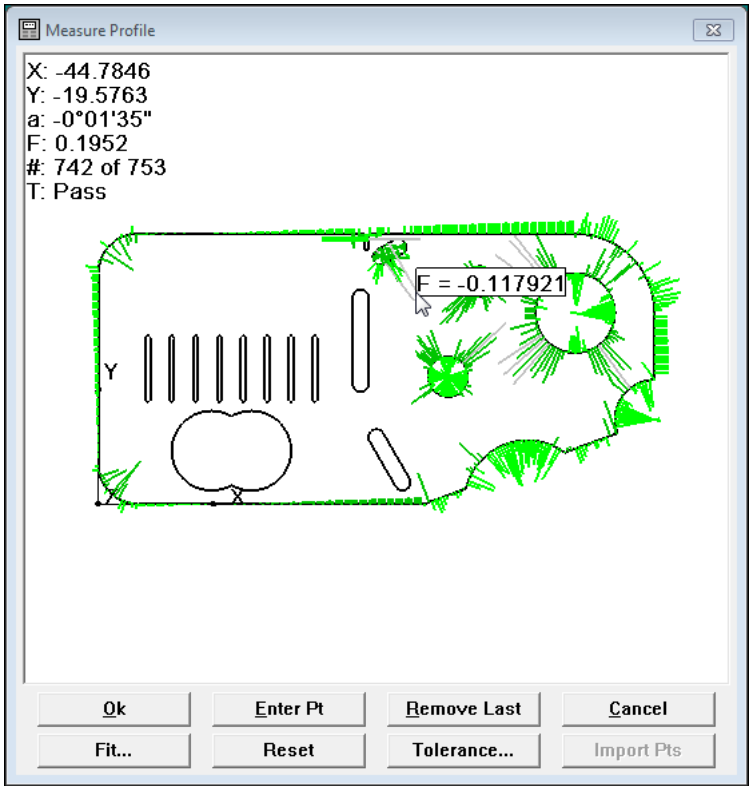
Click **`Fit...`**

- Enter the maximum allowed form error that will be acceptable for data points to be included in the tolerance pass/fail test in the **Maximum Form Error** field
  Any data point that is greater in absolute value than the maximum form error value will be filtered from the overall set of points used in the fit of the profile.
- Enable/disable applicable fit options **Allow X**, **Allow Y**, **Allow Z**, and **Allow A**.
- Click **OK** in the **Fit Settings** window to save the changes, exit the dialog, and perform a fit calculation. Notice the results data on the upper left corner (if display data is enabled).



Enter the Maximum Form Error and click **OK**

- Complete the 2D profile measurement



Complete the measurement

Editing VED Tools in a Program

VED tools can be edited to change the size, number of scans, scan direction and edge type.

For tools with two sides—inner and outer or left and right—the size of each side can be changed independently using the Inner and Outer size change parameters.

For probes like the circle tool, the Inner size change applies to the inner circle and the Outer size change applies to the outer circle.

For probes like the buffer tool, the Inner size change affects the start of scan side and the Outer size change affects the end of scan side.

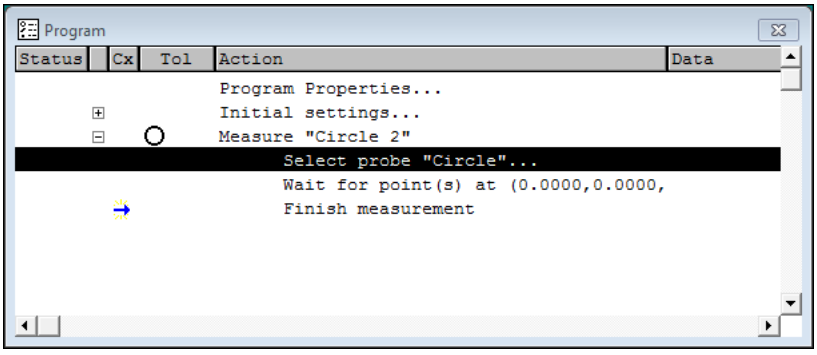
The Inner size change cannot be applied if the change is going to make the inner side larger than the outer side.

The Outer size change cannot be applied if the change is going to make the outer side smaller than the inner side.

Example: Recording a program with features measured using the circle VED tool

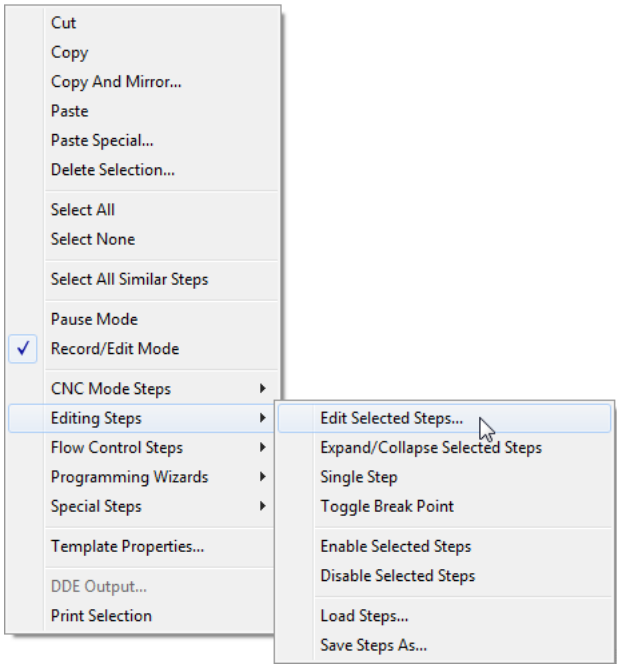
To edit VED tools in a program:

- Start recording a program using the VED circle tool
- Highlight the **`Select probe "Circle"…`** step in the program

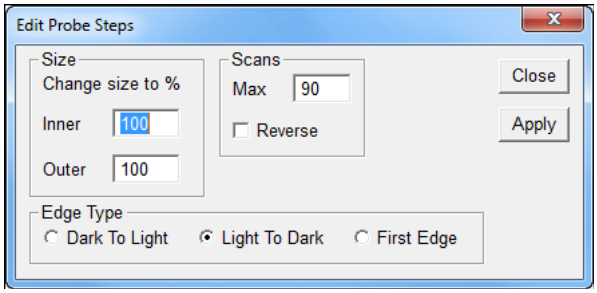


Highlight the **`Select probe "Circle"…`**

- Right-click in the program window and select **Editing Steps>Edit Selected Steps…**
- Enter the size percentage for the selected edge tool in the **Inner** and **Outer** fields
- Make any other neccessary changes to the parameters in the Edit Probe Steps window

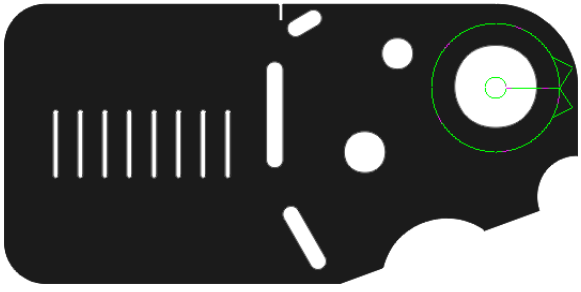


Right-click on the program template and select **Editing Steps>Edit Selected Steps…**

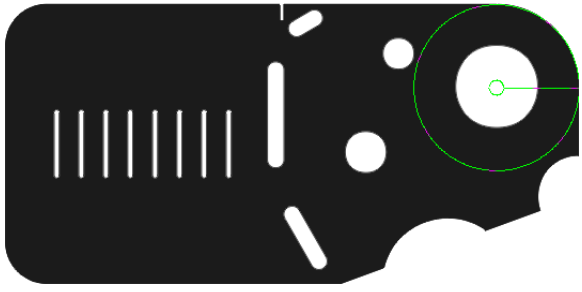


Enter the Inner and Outer size percentage

- Click **Apply** to save the changes and then click **Close** to exit.
  Note the changes on the tool—before and after.



Note the changes on the tool—before.



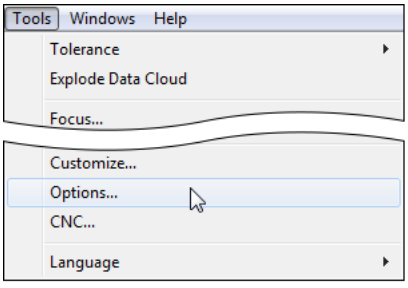
Note the changes on the tool—after.

Changing the Size of Continuous Fire Function Stake Marks

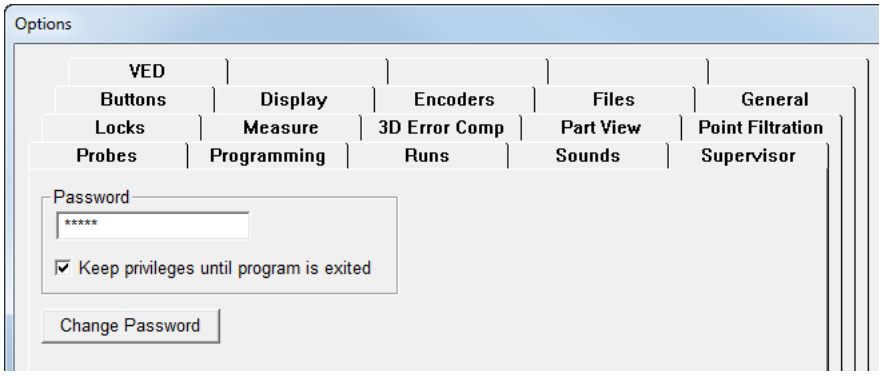
The Continuous Fire VED function uses stake marks to show where an edge has been found. The size of the stake marks can be set from 1–30 pixels.

To change the stake mark size:

- Click **Tools>Options…**
- Select the **Supervisor** tab and enter the supervisor password in the **Password** field

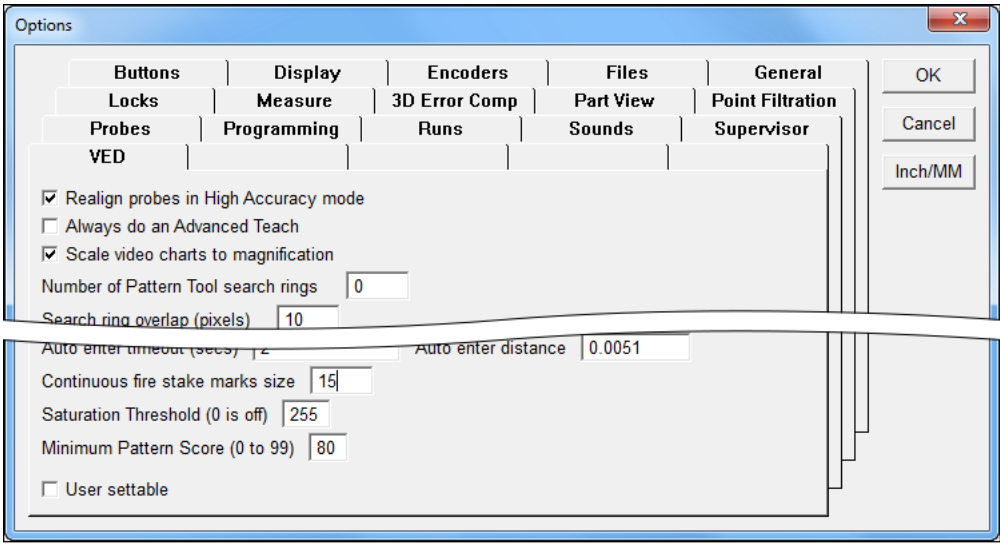


Click **Tools>Options…**



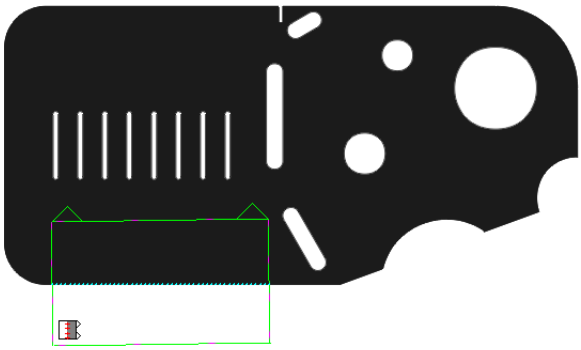
Select the **Supervisor** tab and enter the password

- Select the **VED** tab
- Enter the desired pixel size in the **Continuous fire stake marks size** field. The value needs to be set from 1–30 pixels

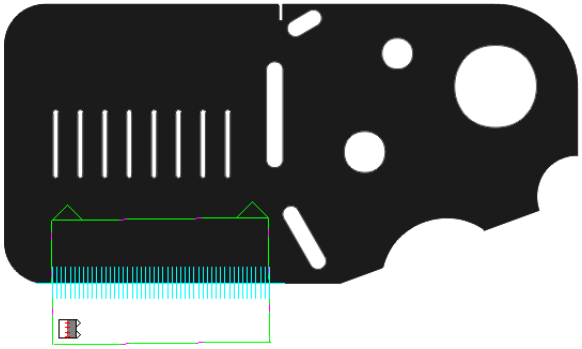


Select the **VED** tab and enter the **Continuous fire stake marks size**

▶ Click **OK** to save the changes and exit the Options window.
Note the changes on the tool—before and after



Note the changes on the tool—before.



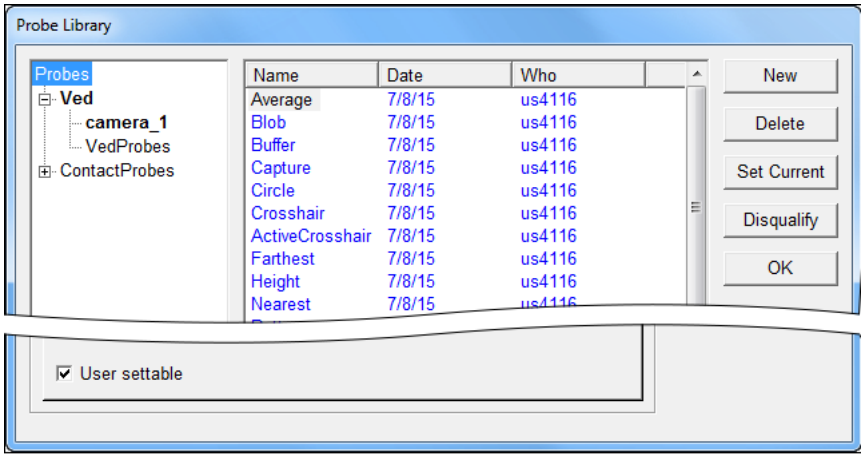
Note the changes on the tool—after.

Enabling User Access to Settings

Most IK 5000 configuration settings are set by the OEM or a supervisor. A supervisor can allow the User to access **User settable** settings if the option is available for that parameter. The **VedProbes** in the **Probe Library** are an example of a **User settable** parameter.

To enable **User settable** settings:

▶ Check the **User settable** checkbox



Check the **User settable** checkbox

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 (8669) 31-0
FAX +49 (8669) 5061
e-mail: info@heidenhain.de

www.heidenhain.de

809541-29 · Ver09 · 5/2017