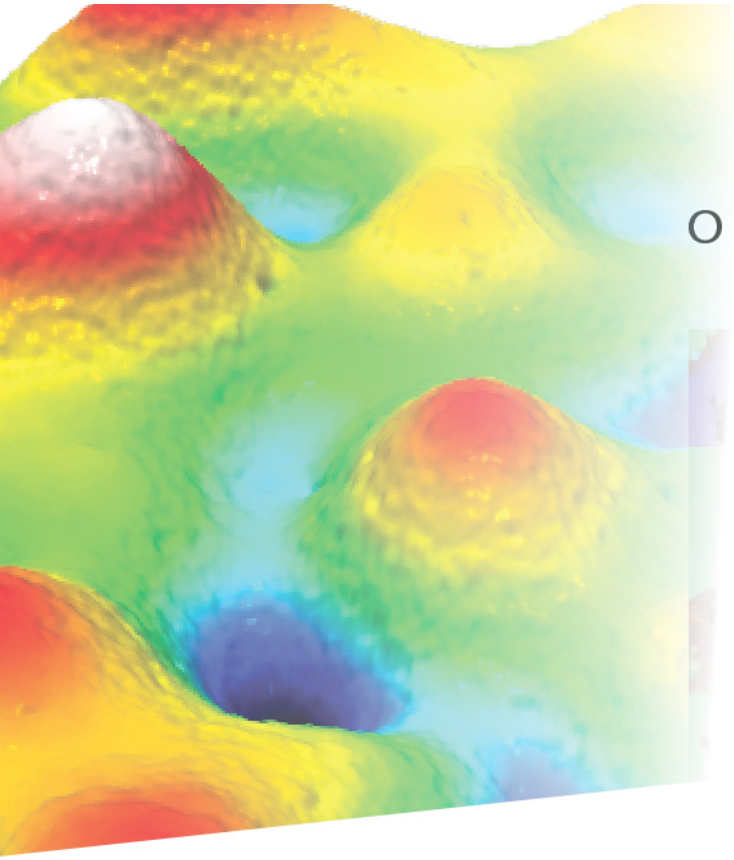


Profilm

OPTICAL PROFILING SOFTWARE

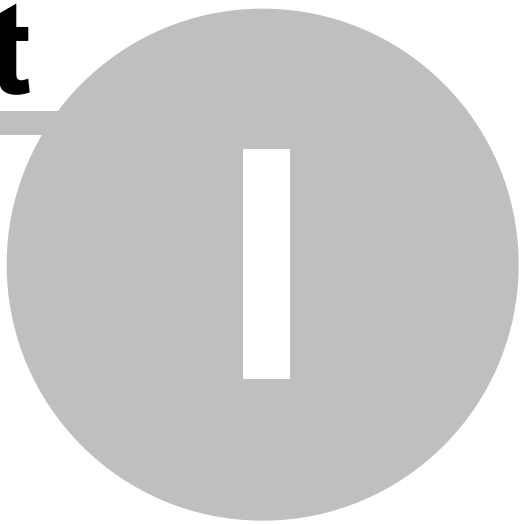
FILMETRICS

Profilm User Manual v.4.0.6.3

Generated: February 2020

Table of Contents

Part



1 Welcome

What is the Profilm software used for?

The Profilm software package is used to analyze sample surface profiles with the goal of quantifying surface topography and film thickness of a wide range for materials. This is achieved through a variety of tools and features which will be described in this manual.

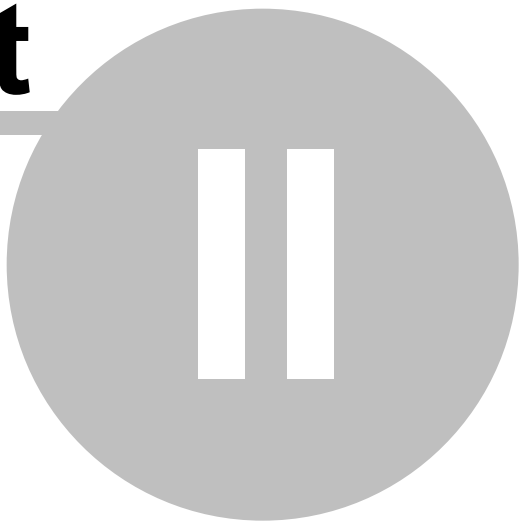
We welcome user suggestions on ways to improve our software and hardware. Please send us any suggestions for new features you would like to see or improvements in the help file.

We may be reached by phone at +1(858)573-9300, through the Contact Us Now form on our website, or by e-mail at support@filmetrics.com.

Thank You,

Filmetrics - A KLA Company
10655 Roselle St
San Diego, CA 92121
Phone +1(858)573-9300
Fax +1(858)573-9400

Part



2 Safety Warnings

WARNING: Improper use may result in injury or damage to the equipment.

Warning Symbols

Shock hazard. This area should only be serviced by a trained service technician.

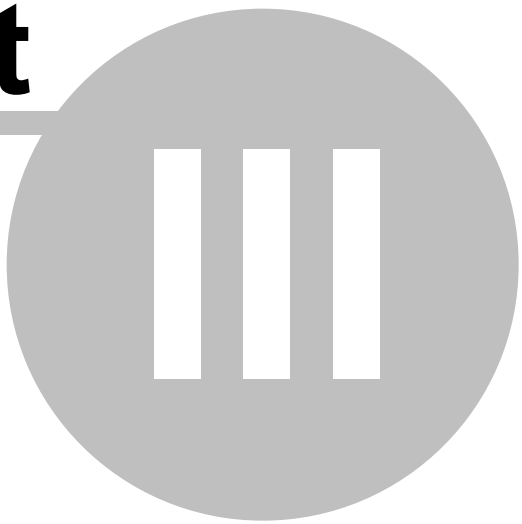
Shock hazard. This area should only be serviced by a trained service technician.

Pinch point. Keep fingers and hands clear when the system is active to avoid injury.

Instrument model and serial number. This information may be necessary when contacting Filmetrics for assistance.

Connect main supply cord only to a properly measured supply. Only use three-wire main supply cord provided with the unit.

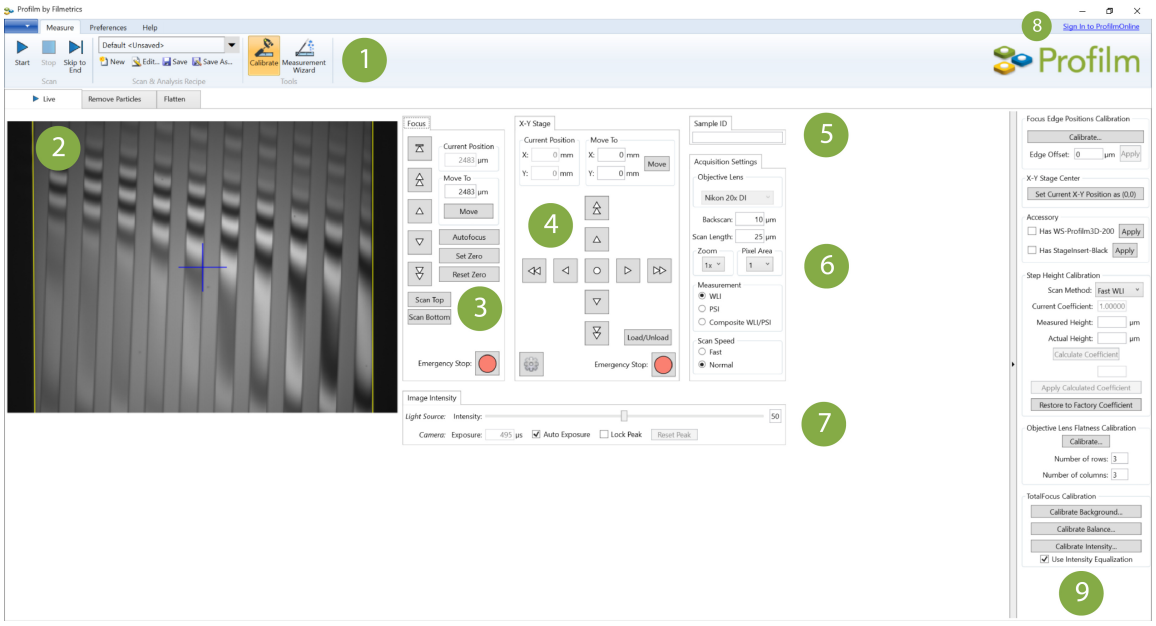
Part



3 Software Overview

As a new user of the Profilm software, the first step is to acquaint yourself with the various interfaces and features. The software is primarily broken up into three sections: the **Live** and **Image** windows for data acquisition and analysis, **Ribbons** for selecting tools and functions, and the **Home Menu** for managing your data.

3.1 The Live Window



*The **Live** window is the first screen that loads on launch of the Profilm software.*

1. Ribbons: Select between the Measure, Preferences, and Help ribbons, or open the Home menu.

2. Video Image: The live video image from the camera is displayed here. Moving the mouse over the video image will display the X, Y coordinates for the given point. These values can be entered into the *X-Y Stage* section to center the image over that point.

3. Focus: These controls adjust the camera focus height. The single arrow is for slow motion and the double arrow is for fast motion. The topmost button will raise the head to its maximum height. Click **Set Zero** to change the current z location to 0, and **Reset Zero** to set it back to the factory 0. **Scan Top** and **Scan Bottom** are used to set your scan length, more details can be found in the Editing Recipes section. Click **Emergency Stop** to stop any active motion in X, Y, or Z.

4. X-Y Stage: These controls move the automated stage. Enter specific X, Y coordinates or click the circle button in the center to automatically center the stage. Click **Emergency Stop** to stop any active motion in X, Y, or Z. Click the settings cog icon to open the X-Y Stage Settings dialog.

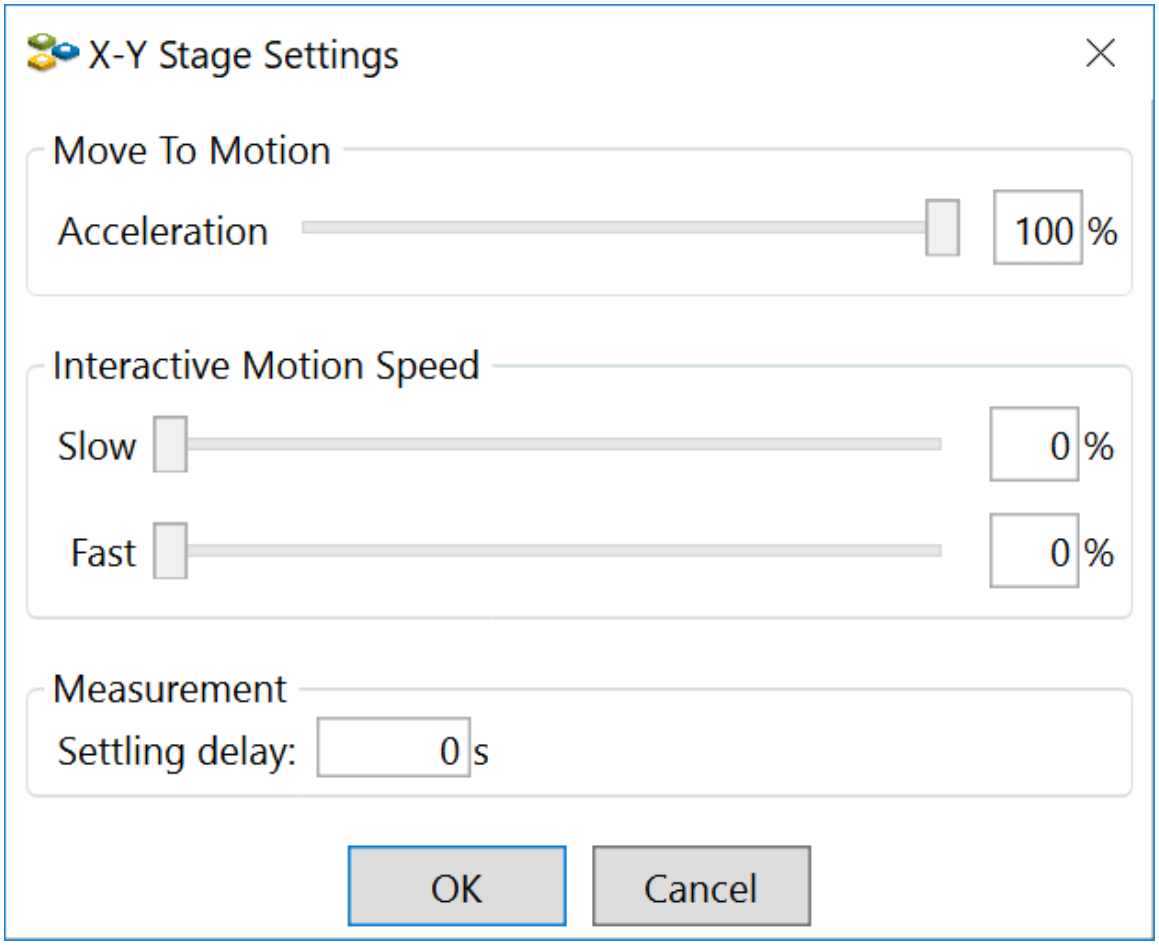
5. Sample ID Enter an identifying sample name here. Subsequent images will have this name followed by a serialized number for all images after the first (e.g., Example, Example 1, Example 2, etc.).

6. Acquisition Settings: These controls adjust the measurement acquisition settings. More details can be found in the Editing Recipes section.

7. Image Intensity:	Adjust the slider left to right to control the light source intensity. Select *Auto Exposure* to have the software determine the best intensity for the currently loaded sample. *Lock Peak* fixes exposure based on the highest intensity seen.
8. ProfilmOnline Sign-in:	Sign in to your ProfilmOnline account to enable uploading of images to the site.
9. Tools:	The section displays settings for the currently selected tool in the **Measure** ribbon. The Help features also appear in this section, when enabled. Click the arrow divider to hide or expand this section, depending on its current state.

3.1.1 X-Y Stage Settings

These options control the acceleration and movement speed of the X-Y stage when it is moved using the **Move To** button, the arrow controls, or during grid scans.

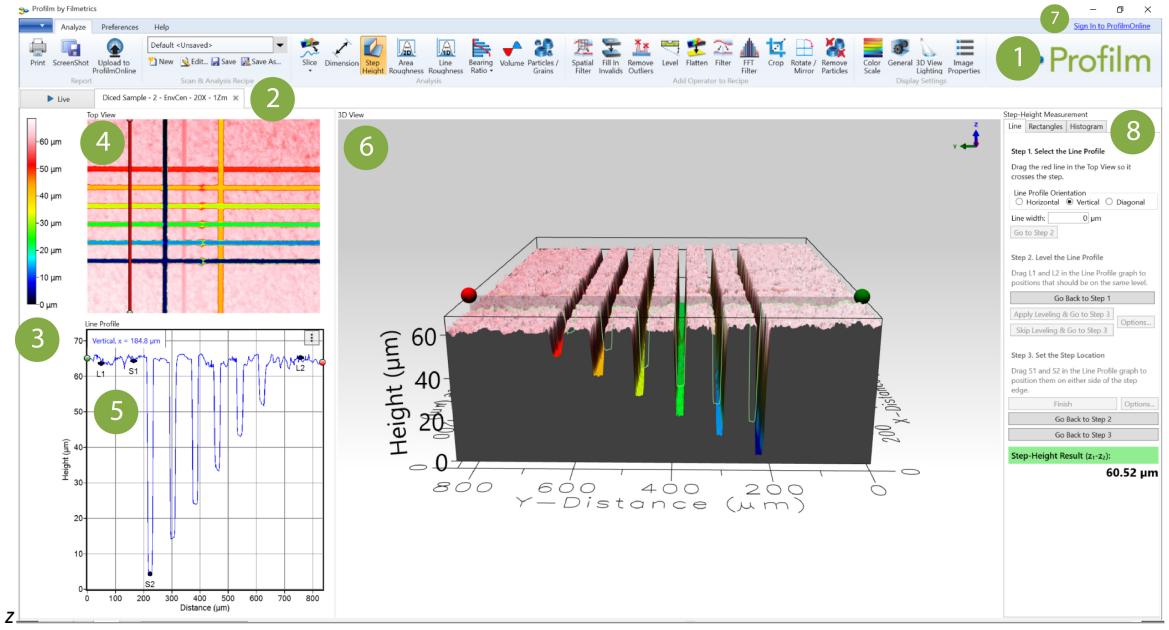


Move To Motion is used to control the **Move To** button acceleration speed. Lowering the acceleration speed can help if samples are shifting during stage movement.

Interactive Motion Speed adjusts the stage movement speed when the arrow controls are pressed, from 0% (slowest) to 100% (fastest). Higher power objectives may use lower speeds to more easily track motion across the sample, and lower power objectives may benefit from increased speeds. Speeds can be set independently for the Slow (single arrow) and Fast (double arrow) controls.

Settling delay is used during grid scans to insert a pause between the X-Y coordinate stage movements, and when taking a measurement.

3.2 Image Window(s)



Once a scan is completed, it will open as an individual tab.

1. Ribbons: Select between the Analyze, Preferences, and Help ribbons, or open the Home menu.

2. Tabbed Browsing: Toggle between currently open images and the Live window by clicking on the corresponding tab.

3. Color Scale: Shows the correlation between the measured height of a given pixel and the color displayed in the *Top* and *3D* views.

4. Top View: A 2D top-down view of the surface; used to select where to perform sample analysis. Zoom in and out from the mouse pointer location using the mouse scroll wheel or trackpad scrolling. See Making Measurements for more details.

5. Line Profile: The linear plot of the current line slice. In horizontal mode, the x-axis correlates to the x-axis of the top view, while the y-axis shows measured height. In vertical mode, the x-axis will instead correlate with the y-axis of the top view. In diagonal mode, 0 will correlate to the start point, while the x-axis max value will match up with the end point.

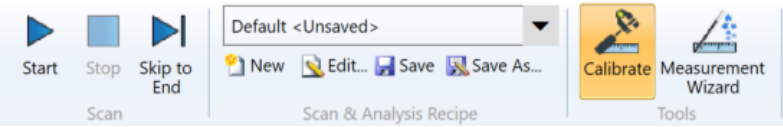
6. 3D View:	This window shows a 3D image of the sample. See Controlling The Camera for more details on how to interact with the 3D view.
7. ProfilmOnline Sign-in:	Sign in to your ProfilmOnline account to enable uploading of images to the site.
8. Analysis Results:	This side panel contains controls, settings, and results for the various analysis functions. The functions are explained in greater detail under Making Measurements.

3.3 Ribbons

Access to the various features and controls in the Profilm software are separated into a collection of different ribbons, each of which are organized around a specific task. This section will review the individual controls in each ribbon and how they are used.

Measure
Analyze
Preferences
Help

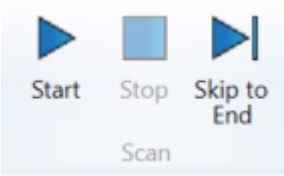
3.3.1 Measure



The **Measure** ribbon contains three sets of controls, which are used to set up and acquire the scan. Note that the **Measure** ribbon is only available when the Live window is selected. Once a scan has been completed and loaded the **Measure** ribbon is replaced by the Analyze ribbon.

Scan
Scan & Analysis Recipe
Tools

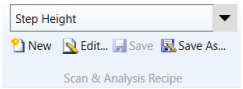
Scan



The **Scan** tab is used to start a new scan, stop a scan currently in progress, or skip to the end an in-progress scan. If a scan is stopped while in progress, no data will be displayed. If multiple scans are required (e.g. scan averaging, grid scan, etc.) text will display next to the start button showing the number to be completed.

Scan & Analysis Recipe

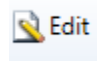
In the Profilm software, recipes are used to store different acquisition and analysis settings. Multiple recipes can be saved for more than one product or application for maximum versatility.



More information on how to create and use the recipes will be included in the Editing Recipe section.

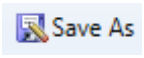
Shows the currently selected recipe. Click the drop-down arrow to open a different recipe.

Click to start a new recipe.



Click to open the Recipe dialog box.

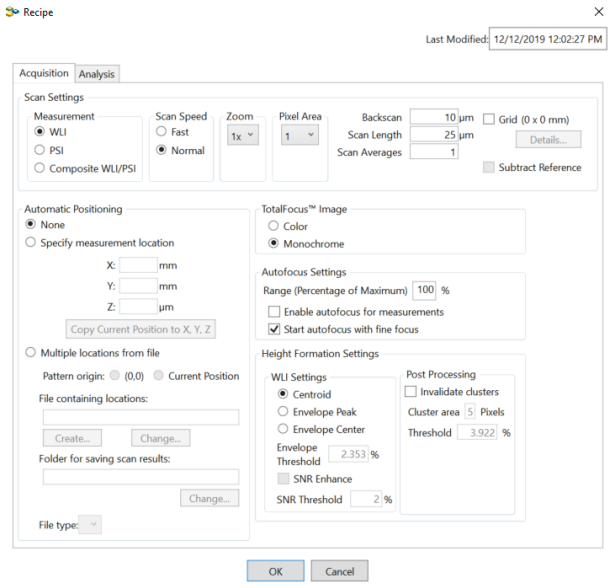
Click to save the current scan settings to the currently selected recipe.



Click to save the current scan settings as a new recipe.

Acquisition

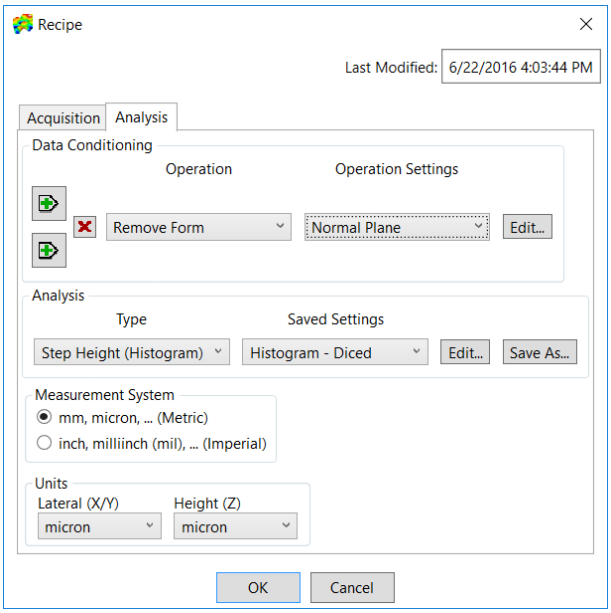
The *Acquisition* tab contains all settings required to measure a sample. The functions are split into multiple sections. The *Scan Settings* are used to select the measurement method and control the motion of the stage and measurement head. *Automatic Positioning* is used to map one or more specific positions. The *TotalFocus™ Image* provides large depth-of-field monochrome or color images (requires UPG-TotalFocus or UPG-ColorImaging). *Autofocus Settings* modify the behaviors of the autofocus routine. The *Height Formation Settings* determine how the software will interpret the measured data.



These controls are explained in further detail in the Editing Recipes section.

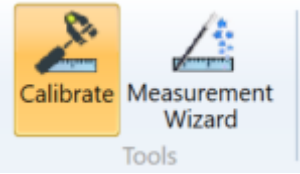
Analysis

Like the Acquisition tab, the *Analysis* tab is split into two sections. The *Data Conditioning* contains the various operators that can be applied to the image such as leveling, form removal, or filtering. The *Analysis* section is used to select the type of analysis to be applied, and how it should be used. The Units used in the image are also set here.



See the Editing Recipes section for further detail on how these features work.

Tools



The **Tools** menu is used to access system calibration options and an example measurement wizard.

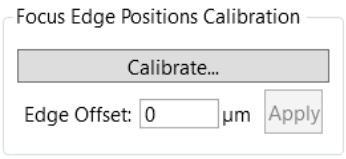
Calibrate
Measurement Wizard

Calibrate

The **Calibrate** options are used to correct selected features in the software that may have changed with time or in shipping. Do not change these settings without first seeking assistance from Filmetrics. Contact us by phone at +1(858)573-9300, through our website, or by email at support@filmetrics.com.

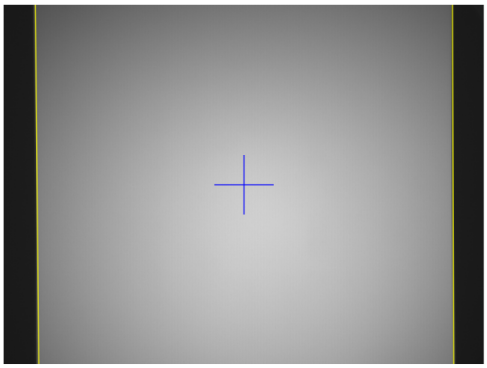
Focus Edge Position Calibration
X-Y Stage Center
Accessory
Step Height Calibration
Objective Lens Flatness Calibration
TotalFocus Calibration (UPG-TotalFocus or UPG-ColorImaging required)

Focus Edge Position Calibration



The camera has a pair of focus edges set in the light path to help assist the autofocus routine. In order for measurements to be made as accurately as possible, it is important that the software knows where these edges are. This is accomplished by completing the *Focus Edge Positions Calibration*.

To access this feature, first select the **Measure** ribbon and then go to the **Tools** section. Click on the **Calibrate** button to open the calibrations side bar. A pair of vertical yellow lines on the live camera image indicate the current location where the software believes the focus edges to be. See the image below for an example alignment.



The focus edges are well aligned in this example image. You must be in 1x zoom to see the focus edges.

In this case, the focus edges (black sections on either side of the image) are in good alignment with the locations estimated by the software. If they are not in good alignment click the **Calibrate...** button to have the software automatically determine their location. Follow along with the instructions on the screen and the system will go through a focus routine and show a new estimated alignment indicated by a pair of blue vertical lines. Assuming the alignment is good click **OK** to set the new locations or **Cancel** to try again. If the software continually fails to find the proper alignment, or if one or both of the black sections are not visible in the camera image, contact Filmetrics for assistance.

The offset can also be entered manually using the **Edge Offset** text box. Click **Apply** to update the displayed position.

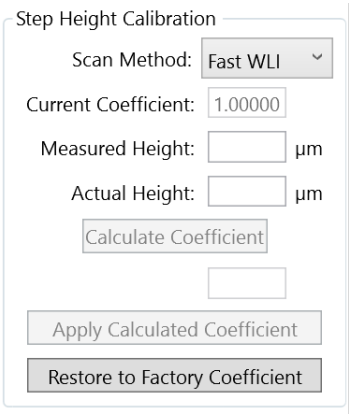
X-Y Stage Center

Allows for the stage center (0,0) position to be set arbitrarily for the X-Y stage. **Set Current X-Y Position as (0,0)** sets the new center point at the current position of the blue cross-hair in the live video image.

Accessory

Indicates the presence of a wafer stage accessory (WS-Profilm3D-200), or the low reflectance stage insert (StageInsert-Black).

Step Height



The **Step Height** calibration tool is used to account for disparities between measurements taken on the optical profiler and measurements taken from another instrument.

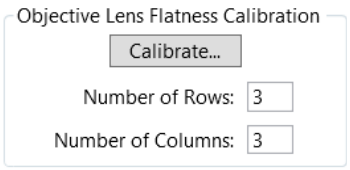
To access this calibration, select the **Measure** ribbon then click on **Calibrate** under the **Tools** section. The **Step Height**, **Focus Edge Settings**, and **Subtract Reference** sections will become visible to the right of the video image.

First, select which scan speed and measurement method to apply the calibration to using the drop-down menu. Next, enter the **Measured Height** value from the Filmetrics optical profiler, then enter the **Actual Height** value taken from the other instrument. Click **Calculate Coefficient** to calculate and display a correlation coefficient to correct for the disparity. Click **Apply Calculated Coefficient** to use the correction. To reset the system to its default settings, click on the **Restore to Factory Coefficient** button.

It is important to note that this calibration will only be applied to measurements taken after the correction was applied. Any previously saved images will continue to display non-corrected thicknesses.

Since the software will only accept a change of ±10% of thickness or less, if a value greater than 10% is entered an error will be displayed and the calibration will be canceled. Contact Filmetrics for further assistance at this point.

Objective Lens Flatness Calibration



On very smooth or very flat surfaces, it is possible that an image artifact may appear in images as a result of imperfections in the reference mirror within the objective. The **Objective Lens Flatness Calibration** can be used to remove this artifact from the data. Before continuing with this calibration, contact Filmetrics to verify the correction is required.

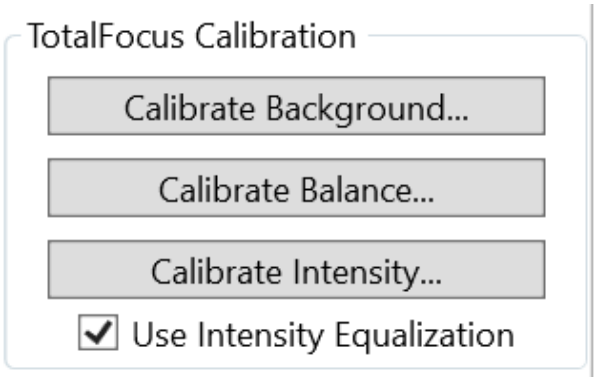
The calibration works by taking a series of measurements across a given sample, determined by the values entered into the **Number of Rows** and **Number of Columns** field. For most applications the default 3x3 grid can be used.

The calibration should be completed using an optical flat, though using your own sample where the artifact is apparent can work. Set up the profiler with the sample leveled as best as possible using the tip/tilt adjustments on the stage. Focus to the sample (i.e., when the interference fringe is visible in the camera image). Click the **Calibrate** button to start the process. The software will automatically move across the defined grid, completing a scan at each location. When it is done, a message will indicate it has completed the calibration successfully.

The **Subtract Reference** button should now be active underneath the Scan Settings in the recipe. Toggle it on and off to apply the correction as needed.

TotalFocus Calibration

Improves the quality of the finished TotalFocus™ image. **Calibrate Background** and **Calibrate Balance** are available for systems with UPG-ColorImaging. Calibrate Background is available for systems with UPG-TotalFocus. Systems with upgrades included at the time of purchase will ship with completed calibrations.



Calibrate Background:

Removes the effect of background signal (noise, room lights, etc.) from the measured image. Click **Calibrate Background** to open a dialog with instructions to complete the calibration, then **OK** to begin the calibration, or **Cancel** to exit without applying changes. The automated process takes approximately 10-15 minutes and confirmation is shown when com-plete. Calibration must be completed for each objective.

Calibrate Balance:

Used to color balance the objective (only available for systems with UPG-ColorImaging). Click **Calibrate Balance** to open dialog will open with instructions to complete the calibration, then **OK** to begin the calibration, or **Cancel** to exit without applying changes. The automated process takes approximately 5-10 minutes and confirmation is shown when complete.

Calibrate Intensity:

Used to balance intensity variations across the image area. Click **Calibrate Intensity** to open a dialog with instructions to complete the calibration, then OK to begin the calibration, or Cancel to exit without applying changes. The automated process takes approximately 5-10 minutes and a confirmation is shown when complete.

Measurement Wizard

The **Measurement Wizard** provides an easily accessible guide for making a measurement on the Profilm3D. For more detailed instructions on measurements, see Making Measurements.

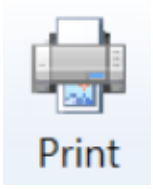
3.3.2 Analyze

The **Analyze** ribbon contains options to analyze data and output the results. Note that the **Analyze** ribbon is only available when an image is selected. If the Live window is selected, the Measure

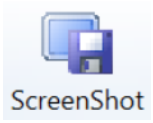
ribbon will be displayed instead. To access the **Analyze** ribbon either click to an already open image, take a new scan, or load a saved image.

Report

The **Report** tab is used to output data to a printer, image file, or ProfilmOnline.



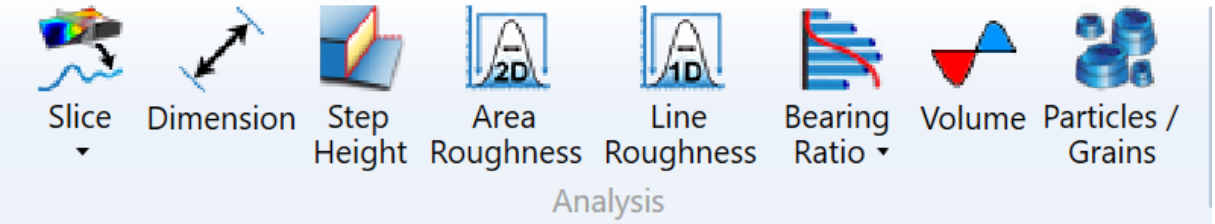
Opens the printer wizard. The printout includes the *Top* and *3D* views, line plot, and results of the currently selected analysis mode.



Takes a screenshot of the currently active image and results. Files are saved as .jpg by default, but other file formats are available.

Uploads the selected image to ProfilmOnline. A ProfilmOnline account is required. If not already logged in through the software, you will be prompted to log in before uploading the file.

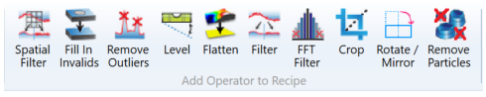
Analysis



The *Analysis* tab is used to alternate between the available analysis modes in the software: Slice, Dimensions, Step Height, 2D Area Roughness, 1D Line Roughness, Bearing Ratio, Volume, and Particles/Grains. The currently active mode will be highlighted and only one can be active at a time.

Add Operator to Recipe

The *Add Operator to Recipe* tab includes a variety of tools used to modify the measured image. Any operator that is applied can be saved as part of a recipe and automatically used on future images.



To see what operators have been used on a given image, click on the Image Properties button in the Display Settings section.

Spatial Filter

The **Spatial Filter** operator applies a matrix filter to the surface data. Six filters are available: *Median*, *Mean*, *Maximum*, *Minimum*, *Gaussian*, and *Laplacian*. These filters are used to improve the image or search for certain surface features.

To demonstrate the basic filtering process, we'll use the example matrix below to represent a 5x5 pixel image where the pixel highlighted in gray (2,4) is the target.

5	4	7	1	6
5	4	3	6	4
8	7	2	2	7
7	7	9	8	7
4	2	3	1	7

The *Mean* filter, set for a 3x3 size, uses the average of the surrounding data points to calculate the resulting value. As a result, the sum for the 3x3 grid is 38 (7+1+6+3+6+4+2+2+7) for the 3x3 grid, which is then divided by 9 to give a final value of 5.4.

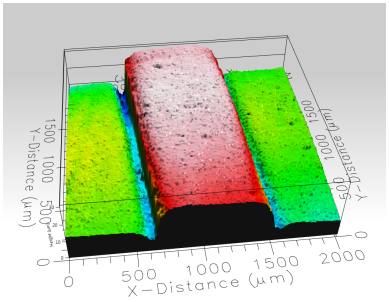
The primary uses of all filters are as follows:

Median Filter

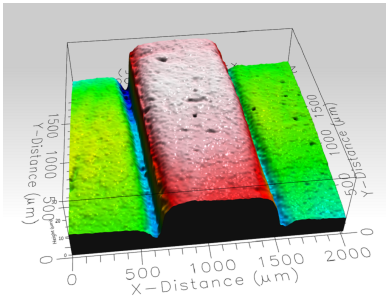
Removes noise from a surface by arranging the data points in the region of the matrix in order from least to greatest. The middle value of the distribution is then used as the filtered value.

Mean Filter

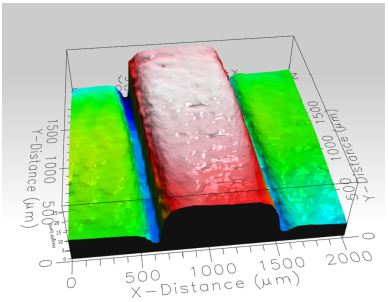
Smooths the surface by averaging each point against its nearest neighbors. The larger the filter matrix size, the larger the smoothing effect.



Original Image



Mean 3x3



Mean 9x9

Maximum Filter

Selects the data point with the maximum value within the matrix region and uses that as the new value. The residual surface created by this filter is often used to find peak outliers on the surface.

Minimum Filter

Selects the sample with the minimum value within the matrix region and uses that as the new value. The residual surface created by this filter is often used to find valley outliers on the surface.

Gaussian Filter

Similar to the *Mean* filter in that it is used to smooth the data. However, instead of using the average value it will apply a Gaussian matrix kernel. An example of the 3x3 matrix kernel is shown below.

1	2	1
2	4	2
1	2	1

Laplacian Filter

Detect sharp transitions in the surface data. An example of the 3x3 matrix kernel is shown below.

-1	-1	-1
-1	8	-1
-1	-1	-1

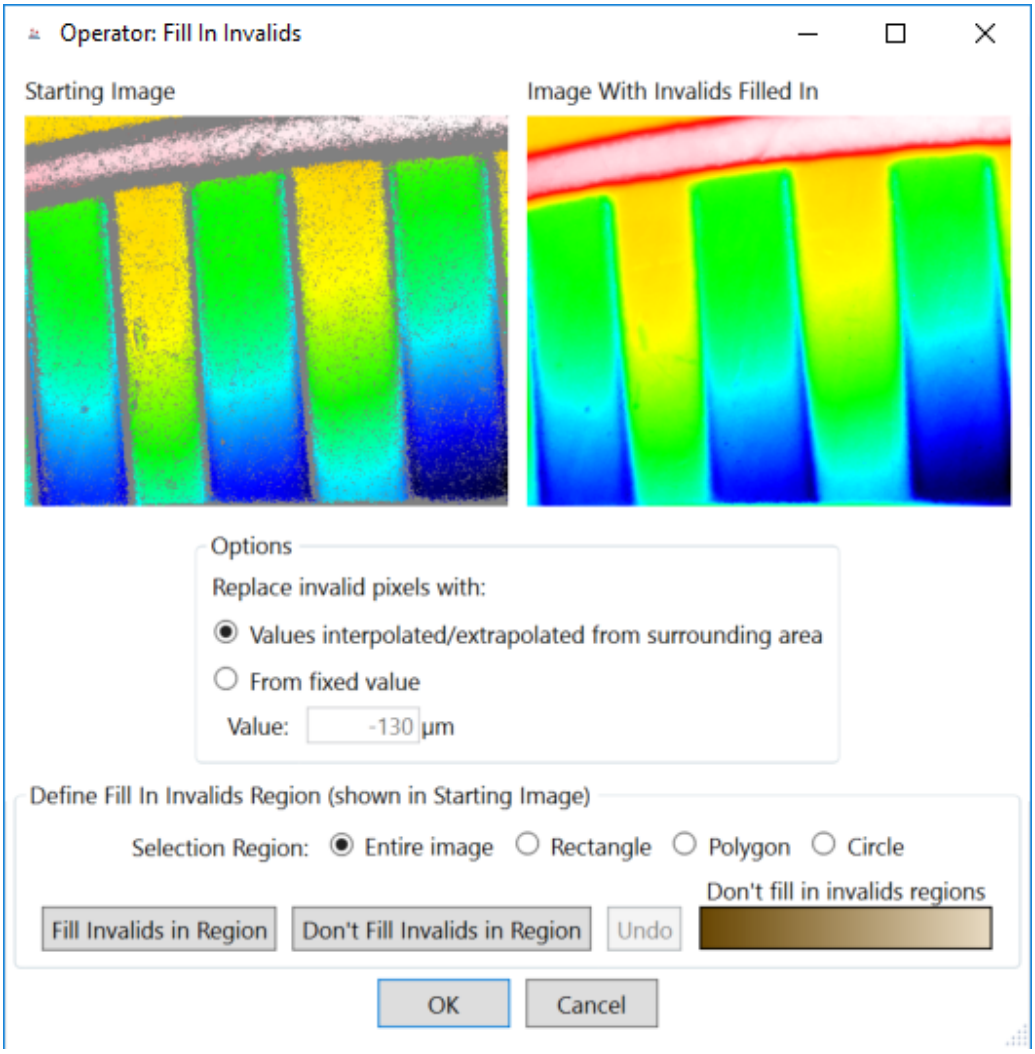
The available filter sizes are 3x3, 5x5, 7x7, 9x9 and 13x13. In general, the larger the filter size the greater the filter effect. The *Smoothed Image* is the result of the *Noise Image* subtracted from the *Starting Image*. The *Noise Image* contains the surface detail to be removed by the filtering process. Select which image to output using the radio buttons below the images. The *Smoothed Image* is selected by default.

Once the *Filter Method*, *Filter Size*, and output image have been selected, click **OK** to apply the filter to the image.

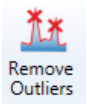
Fill In Invalids

The **Fill In Invalids** tool is used to fill in invalid points based on user input. Select to replace invalid pixels using values interpolated/extrapolated from surrounding data, or from a fixed height value.

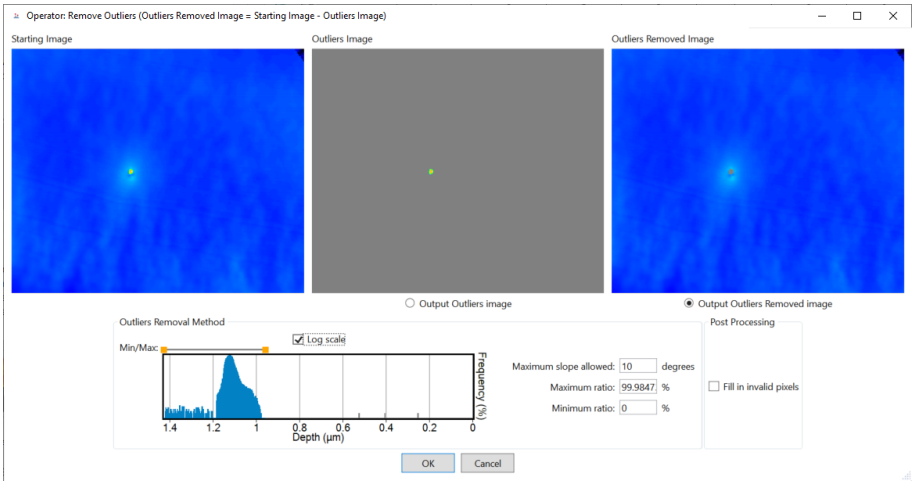
Specific regions can be defined to be filled, similar to other tools like Flatten and Crop. Click **OK** to apply or **Cancel** to exit.



Remove Outliers

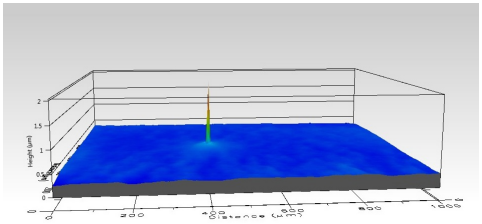


The **Remove Outliers** tool is used to identify and remove outlying data points from the surface by comparing them against the nearest neighboring points.

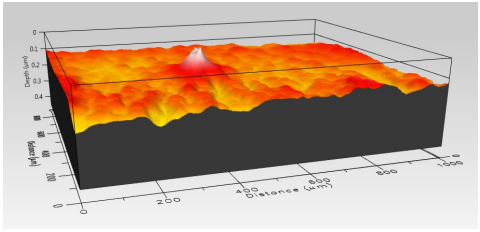


The *Outlier Removal Method* dialog provides controls to define which points are identified as outliers for removal. The graph shows a histogram of the distribution of pixels by their depth in the image with 0 being the highest point. Select to show the display on a logarithmic scale using the *Log Scale* checkbox. *Maximum slope allowed* is the acceptable value, in degrees, allowed between neighboring pixels before a point is considered to be an outlier. The *Maximum* and *Minimum ratio values* refer to the percentage of all pixels used in the analysis. Adjust these values by entering a number into the appropriate text box, or by adjusting the slider above the graph.

For example, in the images below there is a flat field with a strong central spike which is the outlier data point. By setting the maximum slope value at 45 the strong central peak is removed, while the base of the feature remains.



An example surface with a single outlying point



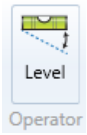
The same surface, post processing.

Any removed points are changed to invalid points in the data. By clicking the *Fill in invalid pixels* checkbox, the software will instead interpolate those data points based on the pixels around them.

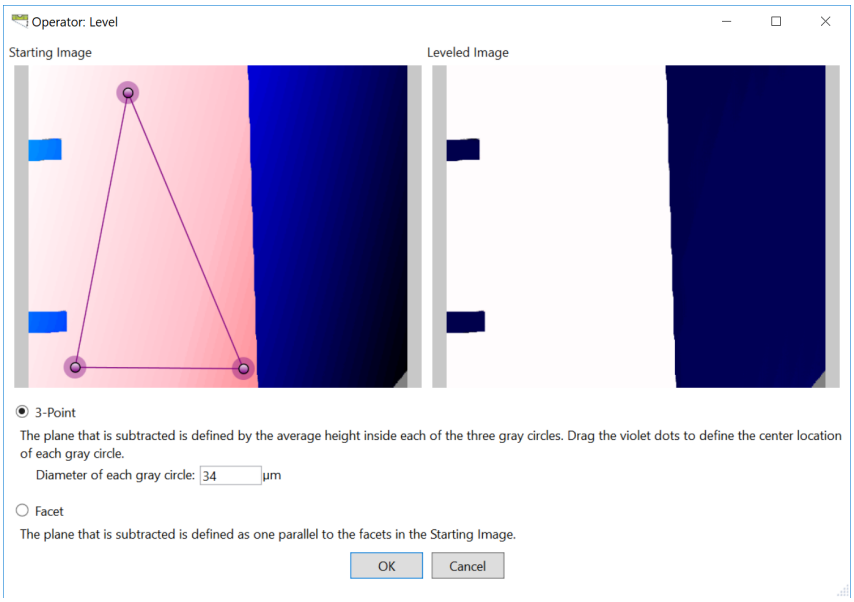
Similar to the other operator features, the user is given the option to choose what image will be shown after the **Remove Outlier** operation is completed. The *Outliers Removed Image* indicates the

remaining points after removing the outliers, and the *Outliers Image* shows which points have been removed.

Level



The **Level** tool is used to remove any tip or tilt that is present in the completed image. To access it first load an image or take a new scan, then click the **Level** button under the **Analyze** ribbon. This will open the *Operator:Level* dialog shown below. Note that the level tool is best used for planar surfaces.

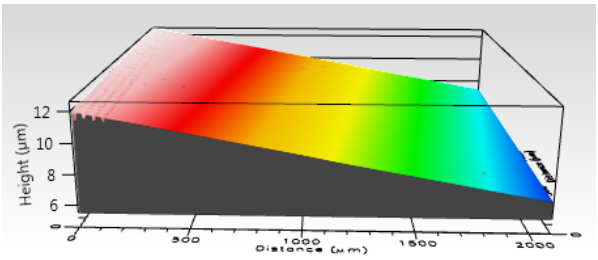


The Operator: Level dialog box.

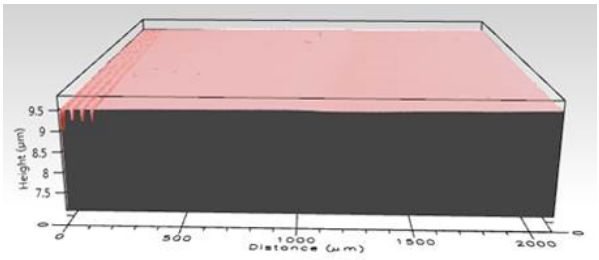
The dialog will display two images, the *Starting Image* and the post-processing *Leveled Image*.Then select one of the leveling methods, *3-Point* or *Facet*.

3-Point leveling is performed by fitting a plane through the average height of the three shaded circular areas surrounding the dots in the source surface image. The radius of these dots can be controlled using the text box underneath *Leveling Method*. The locations of the three points can be changed by clicking and dragging the center of each dot. As the radius and location are changed, the leveled surface updates to reflect the changes. When at the desired level correction, click **OK** to apply it to the image.

Facet leveling is intended for use on samples with more than one planar surface; the step height standard is a good example. Instead of using three points to determine the level, the software will subtract a plane that is defined as parallel to the facets in the starting image.



Example surface before leveling.



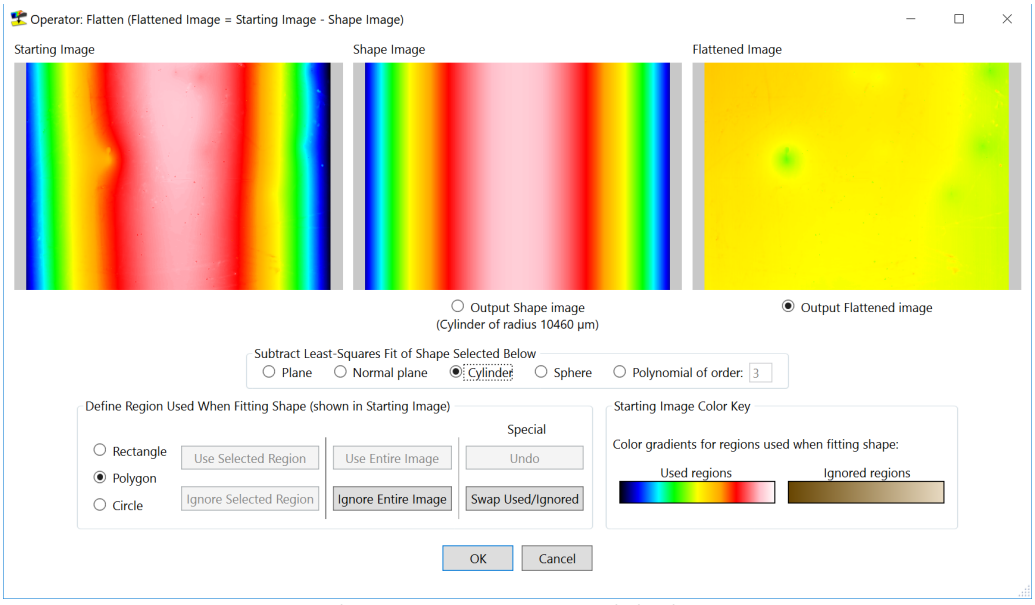
Example surface after leveling.

This correction will remain as long as the image is open, but the file must either be saved or saved as a new file to permanently apply the changes.

Flatten

The **Flatten** tool is used to remove the inherent shape of the sample from the texture of the surface. Profilm has five methods by which this function can be completed: *Plane*, *Normal Plane, Sphere*, *Polynomial*, and *Cylinder*.

When using this tool, select the image to load after analysis options are selected. The *Flattened Image* is the post-processing surface, while the *Shape Image* shows the form that was subtracted from the surface.



The Operator: Remove Form dialog box.

Plane: Works in a similar manner to the Level - 3-Point function by selecting three points to generate a plane through the image, which is then removed from the surface. The Profilm software automatically selects the points for the plane.

Normal Plane: Works in a similar manner to the Level - Facet function by calculating a plane parallel to the facets in the surface. Works best on surfaces with multiple planes at varying heights, like the step height standard.

Cylinder	The cylinder method detects and removes any inherent cylindrical shape from the measured sample. This feature works best the closer the ridge of the cylinder is to a perfectly vertical or horizontal orientation. The calculated radius of the removed form will display when this option is enabled.
Sphere:	The software will detect and remove the best-fit sphere from the selected image. The calculated radius of the removed form will display when this option is enabled.
Polynomial:	The polynomial method is used to compute a best-fit polynomial curve through the measured surface. The more complex the surface to be removed, the greater the polynomial order needed. A total order of 1 to 13 can be entered into the text box.

Defined regions of the image can be removed from the form removal analysis. Choose to add a user-defined polygon, where clicking on the *Starting Image* to define points to create an exclusion range (minimum of 3 points required), or by placing an adjustable rectangular or circular contour on the sample. Multiple contours can be used at one time.

Once the contours have been defined click on the **Ignore Selected Region** button to remove the desired feature, or **Use Selected Region** to replace it. Ignored areas will be highlighted with a sepia tone scale, while used areas will use the rainbow scale. Use the **Swap Used/Ignored** button to switch the used and ignored sections. Click **Use Entire Image** to remove all contours.

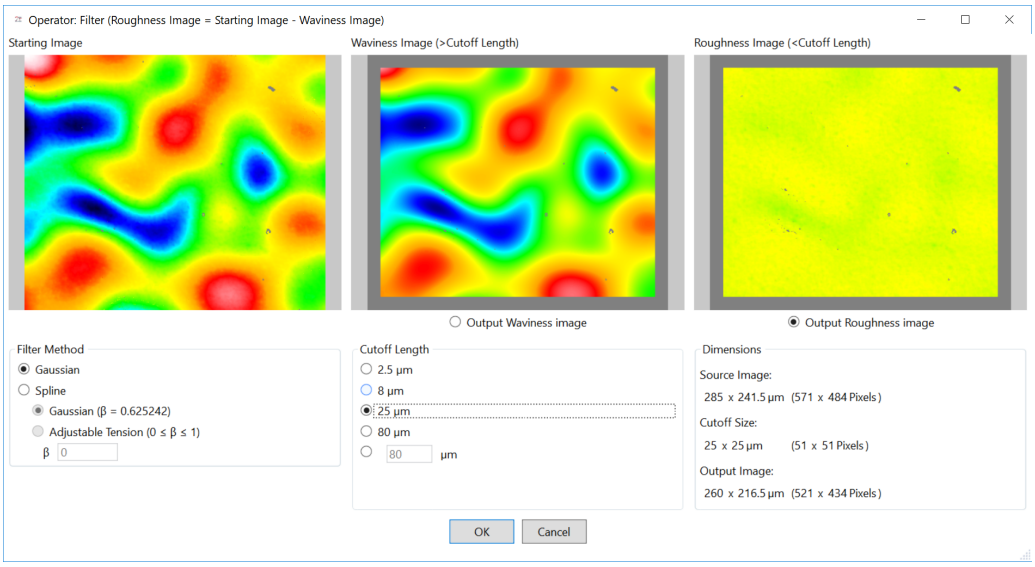
Filter

The **Filter** tool is used to remove features from the surface by applying a low-pass and high-pass, two-dimensional filter. Filtering is determined by selecting one of five predetermined cut-off lengths, or by entering a custom value. Active cut-off lengths are determined by the objective and zoom setting selected. There are two filtering methods available, *Gaussian* and *Spline*.

The *Spline* filter is the more flexible of the two options, and does not result in a smaller image after application. Variation in the spline method is achieved by adjusting the tension (β), which can be set from 0 to 1 depending on user input. By default, the tension will be set to β = 0.625242, which will closely approximate the effect of the *Gaussian* filter.

The *Gaussian* filter will result in a smaller image when applied, as it averages the surface over a particular area. The region that will be lost is shown as a grey band around the *Roughness* and *Waviness* output windows. The size of this region is determined by the cut-off length that has been selected.

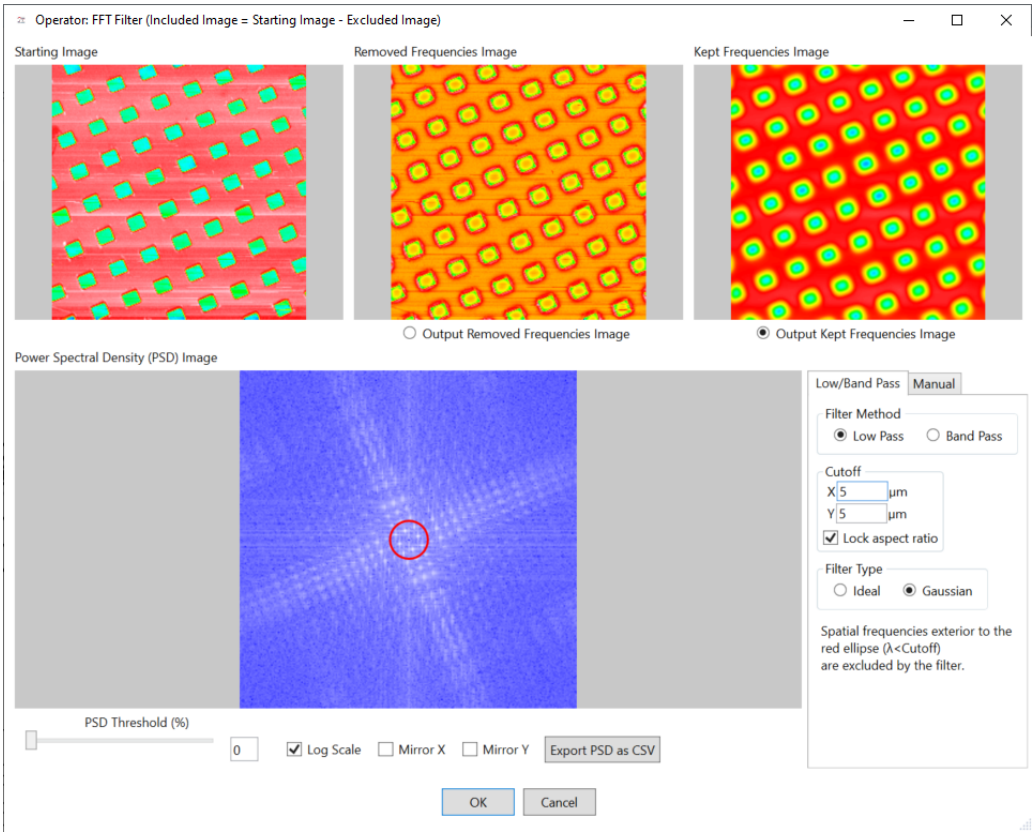
Select the resulting surface to be displayed after the filtering process by clicking the radio button for *Output Waviness image* or *Output Roughness image*.



The Operator: Filter dialog box.

FFT Filter

The **FFT (Fast Fourier Transform) Filter** operator separates short wavelength components from long wavelength components in an image. It also separates components of a surface that lie in a specific direction. FFT Filter must be used on an image with no invalid pixels. Use Fill In Invalids or Remove Outliers operators to fill invalid pixels.



The FFT Filter dialog box contains the *Starting Image*, *Removed Frequencies Image,* and *Kept Frequencies Image* output previews.

Removed Frequencies Image shows values that are to be removed based upon the selected method and cutoff values. *Kept Frequencies Images* shows the values that are retained, and is the default selection.

Power Spectral Density (PSD) Image is a 2D representation of the filtered spatial frequencies. Higher spatial frequencies are shown in the center, while lower frequencies are shown at the edges. The blue to white gradient shows how often a frequency occurs. Click **Export PSD as CSV** to save the PSD image as a .csv file for analysis in other software packages.

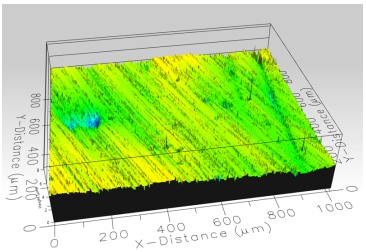
The *PSD Threshold (%)* slider filters noise in the PSD image based on the cutoff percentage selected. By default, these values are set to display on a *Log Scale* but can be toggled using the checkbox.

Values below the cutoff are set to the minimum value and displayed as blue pixels. Changing this value is intended as a visualization aid, it has no effect on the resulting image.

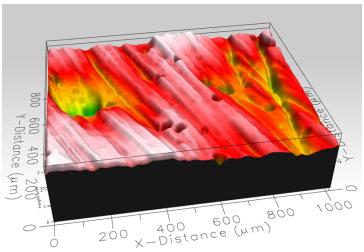
FFT Filter requires the image dimensions be a power of two (e.g. 256 x 512), otherwise the operator pads data from the edge to the next nearest factor of two. **Mirror X** and **Mirror Y** pad edge pixels by mirroring values around the selected axis. If neither are selected, a zero value is used to for all pixels padded to the edge, which can cause artifacts in the resulting image.

Low Pass filtering separates low and high frequency information in an image by varying the *Cutoff* wavelength in the X and Y direction. Spatial frequencies outside of the red ellipse are excluded from the filter. Cutoff values determine the size of the ellipse. **Lock Aspect Ratio** maintains the ratio between the X and Y cutoff values when either are changed.

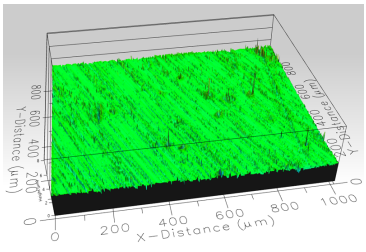
Low Pass requires a filter type be selected. The *Ideal* filter uses the cutoff values as a hard stop, while *Gaussian* applies a slight blur around the cutoff length which tends to look more natural to the eye. See the **Low Pass** filter example below:



Original Image



40 µm Low Pass Filter - Kept Frequencies image

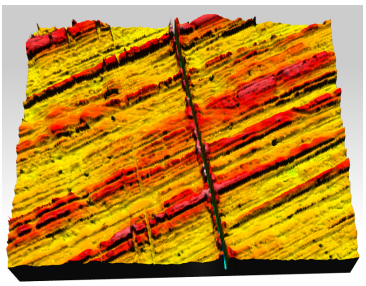


40 µm Low Pass Filter - Removed Frequencies image

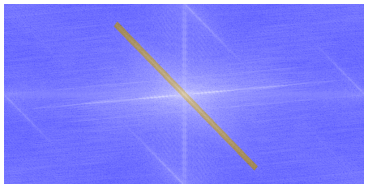
Band Pass adds the option to include a high cutoff value to pass a specific band of wavelength values, as opposed to a simple low pass. The low cutoff value will be displayed as a red circle, while the high cutoff value will be shown in yellow.

Manual filtering allows selection of the exact Fourier components to be attenuated by the FFT filter. Feature selection works similarly to the Flatten operator, where regions are selected using rectangular or circular contours, or a user-defined polygon. Click **Remove Selected Frequencies** to exclude the selected region, which is shown in sepia tone. **Keep Selected Frequencies** adds a defined region inside an excluded zone. Note, as the PSD image is mirrored around the x and y access, the software places a matching region in the conjugate spatial frequencies whenever an exclusion/inclusion region is defined.

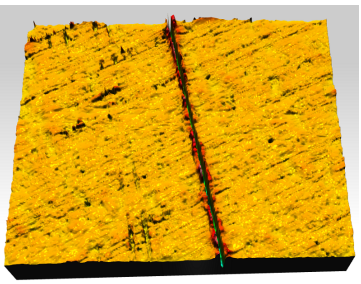
In the example below, the image contains a surface with a prevailing grain and a deep scratch. Using the **Manual** filter option, it was possible to remove the grain information from the image while leaving the feature of interest - the scratch - for further analysis.



Original Image

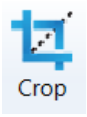


Power Spectrum Image with Excluded Region

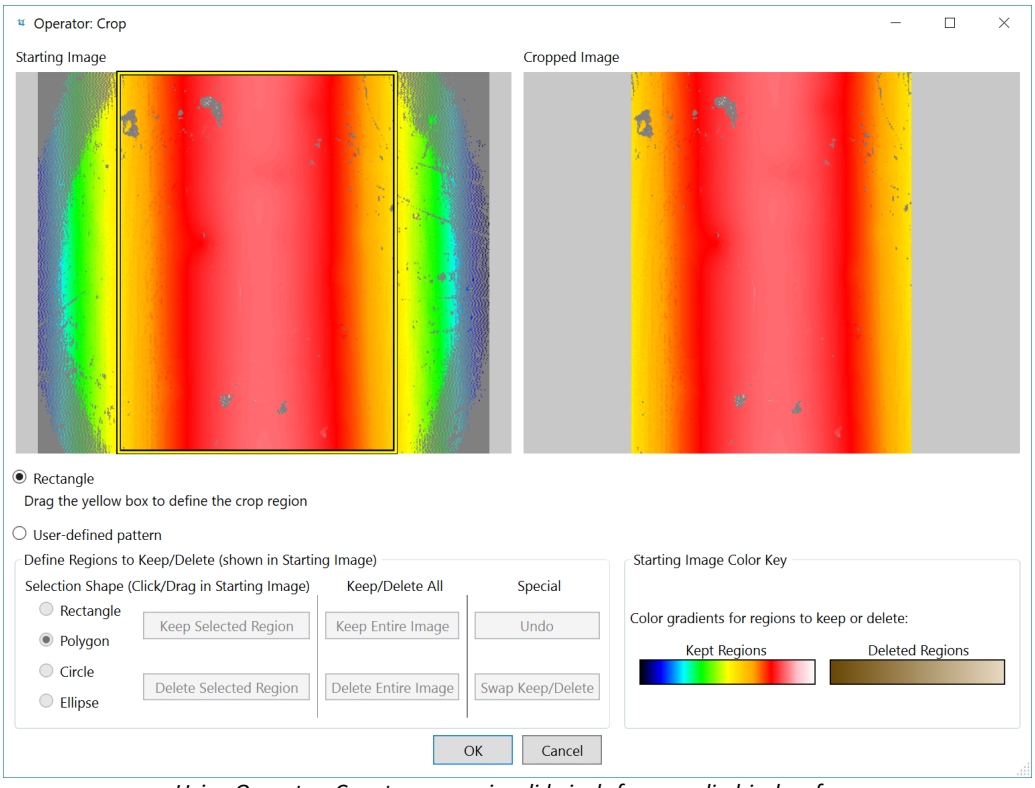


Output Included Image

Crop



The **Crop** tool is used to remove unwanted sections from a completed image.



Using Operator: Crop to remove invalid pixels from a cylindrical surface.

Rectangle removes information outside of the yellow rectangle on the *Starting Image*. Click and drag the yellow borders to resize the cropped area. Click and drag any area within the borders to move the rectangle without resizing.

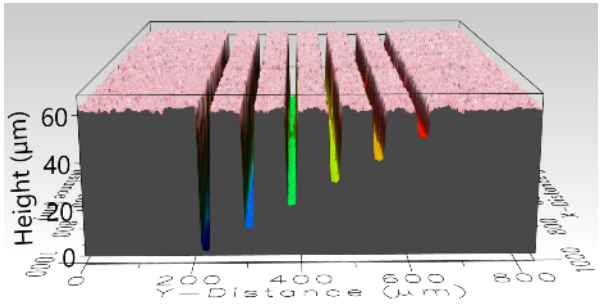
User-defined pattern functions as it does in Flatten. Choose to add an user-defined polygon, or adjustable rectangular, circular, or elliptical shapes. Define the user-defined polygon by clicking on the Starting Image to add points to create an exclusion or inclusion range (minimum of 3 points required). Multiple shapes can be used concurrently.

Once the shapes are placed, keep or delete a region using the included button controls. Kept data is shown in a rainbow color scale. Deleted regions are in sepia tone. Use smaller contours to remove outlying features that may be difficult to identify with the Remove Outliers tool.

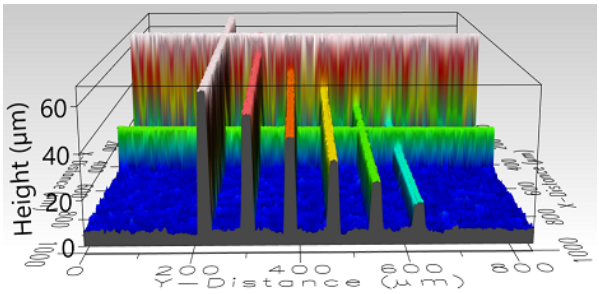
Click **OK** to apply changes, or **Cancel** to exit.

Rotate/Mirror

The **Rotate/Mirror** tool can either rotate an image surface at 90 angles, or mirror the surface around the x, y, or height axes.

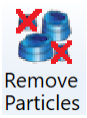


Example surface as measured.



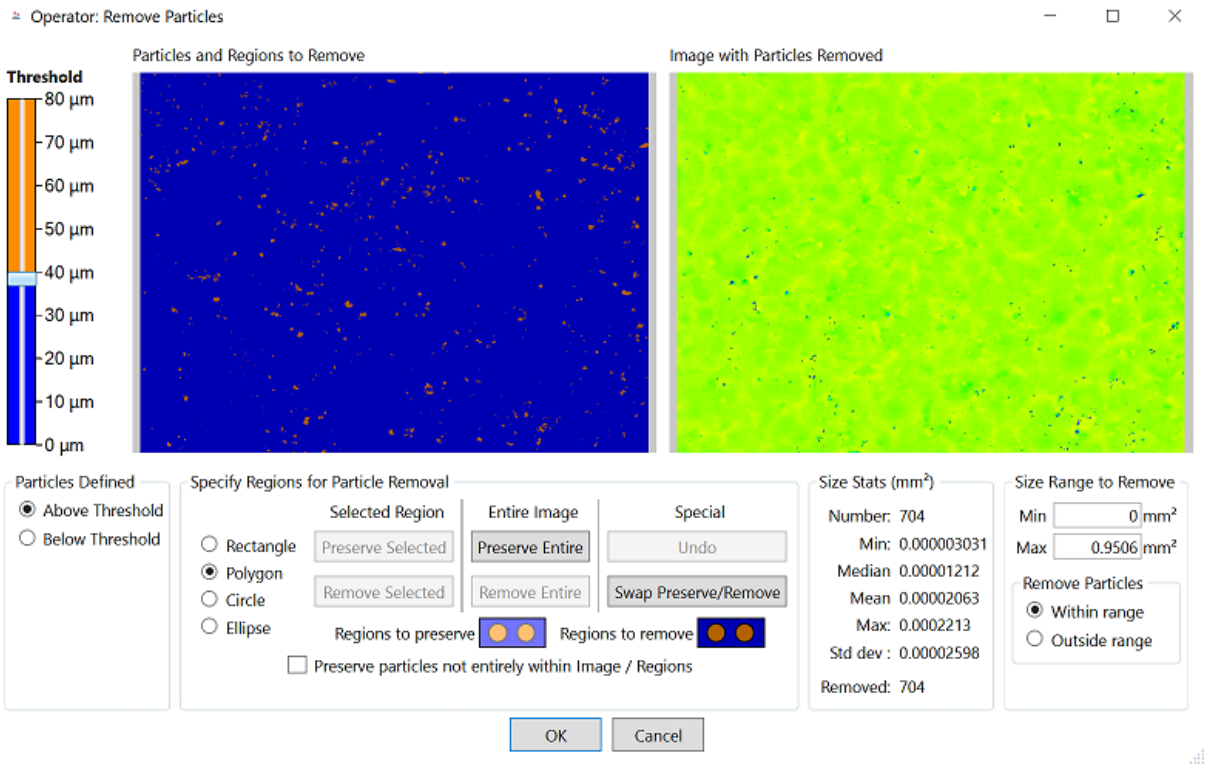
Same surface mirrored about the height axis.

Remove Particles



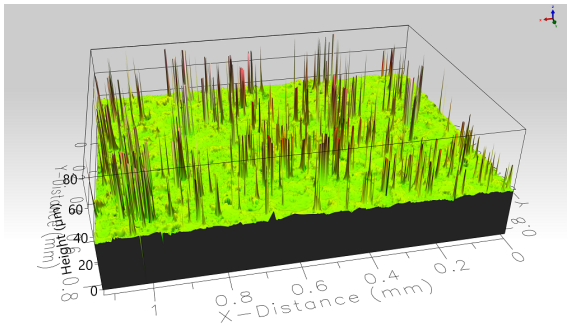
The **Remove Particles** filter removes features above or below a set height plane.

The dialog box displays a two-dimensional version of the image, colored in orange and blue. The region color is determined by whether the region is found above (orange) or below (blue) the height plane set by the *Threshold* value. Adjust this value by clicking and dragging the slider bar. The image updates as the changes are made.

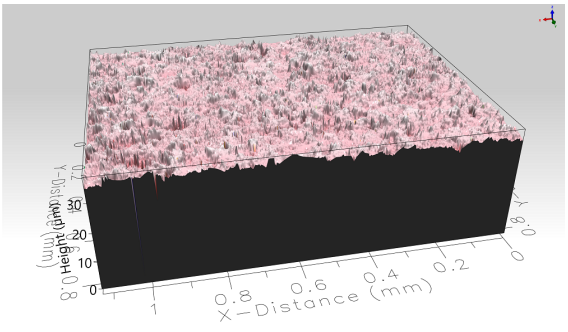


Particles may be removed or preserved from the entire image, or from a specific region. Choose to add a user-defined *Polygon* region by clicking in the *Particles and Regions to Remove* image to define the polygon points (minimum of 3 points required), or by placing an adjustable *Rectangle*, *Circle*, or *Ellipse* region on the *Regions to Remove* image. Multiple regions can be used at one time.

Once the regions have been defined, click on the **Remove Selected** button to remove the feature, or **Preserve Selected** button to replace it. The orange and blue colors will be shaded darker in removed areas, and lighter in preserved areas. Use the **Swap Preserve/Remove** button to switch the preserved and removed sections. Click **Preserve Entire** to remove all regions.



Original Image



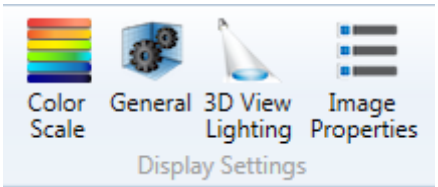
After using the Remove Particles operator.

Particles not entirely within the image or region may optionally be preserved.

Finally, particles that fall within or outside of a user-defined range may be removed. By default, the range will include all particles.

It is important to note how the **Remove Particles** operation will work on subsequent scans once it is saved to the recipe. The threshold (although displayed in height units) is actually saved as a percentage of the height range. Thus, if the threshold is set at a value halfway between the min and max values, this will still be the case when applied to a new scan, even if the height range of the new scan and the original scan do not overlap.

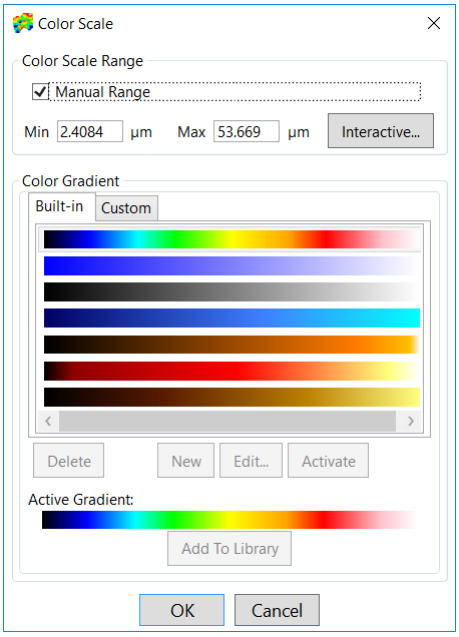
Display Settings



Display Settings include **Color Scale**, **General** and **3D View Lighting** parameters, and **Image Properties** display.

Color Scale
General Display Settings
3D Lighting Settings
Image Properties

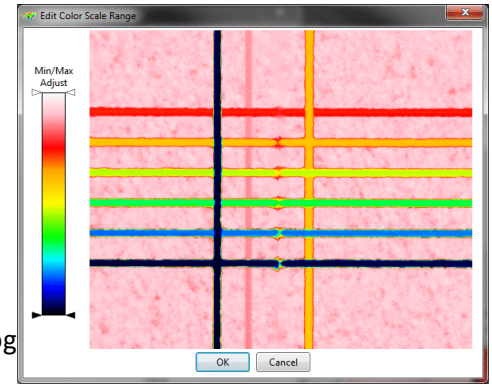
Color Scale



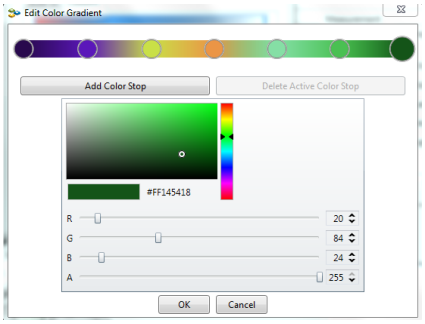
Color Scale can be used to choose between preinstalled multi-colored or multi-shaded gradients, and custom gradients you create. The software comes with several different gradients located in the **Built-in** tab of the *Color Scale* dialog box. Click to activate the desired built-in gradient.

After all desired changes are completed, click **OK** to apply or **Cancel** to exit the dialog without making changes.

Selecting *Manual Range* allows a user-defined range. Enter custom values, or click on the **Interactive...** button to open the **Edit Color Scale Range** dialog box. Adjust the sliders on the left-hand side of the image in the dialog box to determine min and max locations. The image will automatically update as the slider moves through the color range.



When the desired range is achieved, click the **OK** button to apply the changes. The *Min* and *Max* in the *Color Scale* dialog will update to match the new selections.



To add a new gradient, select the **Custom** tab, then click the **New** button to create a new gradient. Click on the new gradient to activate, then click the **Edit** button to open the *Edit Color Gradient* dialog box.

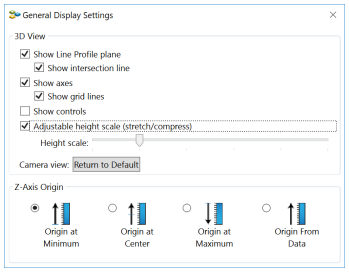
The *Edit Color Gradient* dialog box offers the following options:

1. Add a new color stop - Select **Add Color Stop** to create a new color point to the gradient. Continue to Step 2 to edit the color of the new stop.
2. Edit an existing color stop - Click on the color stop to activate, then use the vertical slider and adjacent moveable selection point to select the color family and shade. Alternately, you can use the Red, Blue, Green, and Alpha sliders (labeled R, G, B, and A respectively) to create a custom color, or enter the hexadecimal value to select a precise color.
3. Move an existing color stop - Click and drag the color stop to the desired location. Note that the end points cannot be moved.
4. Delete an existing color stop - Click on color stop to activate, then click the **Delete Active Color Stop** button. Note that the end points cannot be deleted.

When the desired color gradient is achieved, click **OK** to save the changes. Click the new gradient to select for use. You may create and save an unlimited number of custom gradients.

The built-in gradients cannot be directly edited, but can be copied into the custom library for modification. Click on the desired gradient to activate, and then click the **Add To Library** button.

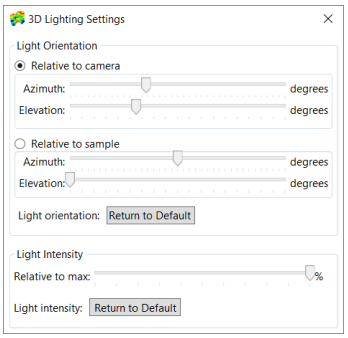
General Display Settings



General Display Settings provide options to change the way data is presented in the 3D View window. Changes made here do not affect the underlying data, only the presentation.

Show Line Profile plane in 3D View:	Toggle on/off to show the currently selected line profile plane from the *Top View* on the *3D View*.
Show intersection line:	Toggle on/off to show a line tracing where the profile plane touches the surface of the *3D View.*
Show axes in 3D View:	Toggle on/off to show X, Y, and Z axes and labels on the *3D View*.
Show grid lines:	Toggle on/off to show grid lines on the *3D View*.
Show Controls:	Toggle on/off to show image manipulation controls on the *3D View*.
Adjustable Height Scale:	Toggle on/off to enable scaling on the z-height scale. Set the height scale value for the image from 0 (flat) to 5 (strongly exaggerated). 1 is the default value.
Camera View:	Click **Return to Default** to revert image orientation changes (rotation, lateral or vertical motion, or zoom) to the default camera view.
Z-Axis Origin:	Select one of four points as your origin: Minimum - 0 is at the lowest measured point in the image. Center - 0 is at the center of the image. Maximum - 0 is at the highest measured point of the image. From Data - Range is set based upon absolute position in the scan range.

3D Lighting Settings

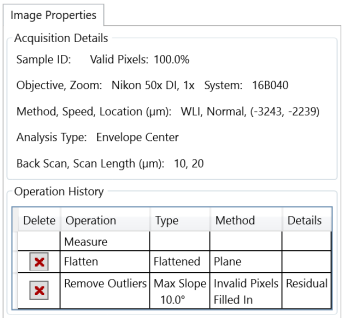


The **3D Lighting Settings** dialog is used to adjust the lighting settings used on the 3D view. Depending on the nature of the sample, adjusting these settings can make it easier to see certain features.

Light Orientation: Manually controls the angle of the *Azimuth* and *Elevation* of the light source relative to either the camera or sample. Clicking on **Return to Default** will reset to default values (0 azimuth, -45 elevation *Relative to camera*, 0 azimuth, -90 elevation *Relative to sample*).

Light Intensity: Manually controls the relative intensity of light hitting the 3D view. Clicking on **Return to Default** will reset to the default value (100% *Relative to Max*).

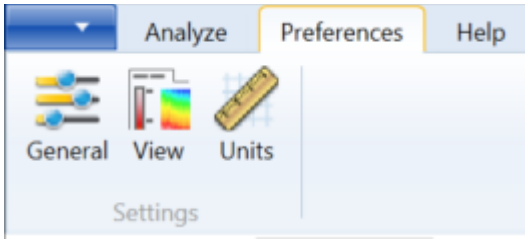
Image Properties



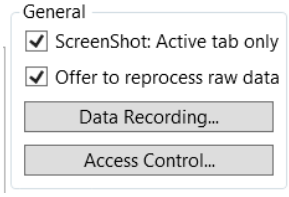
The **Image Properties** section contains a large selection of information relating to the currently loaded image. This includes everything from the objective and system used (by serial number), to the zoom settings, scan method, measurement speed, back scan, scan length, and the location on the sample stage where the measurement was taken.

Also included is the history of all operations that have been completed on the selected image, such as step height analysis, form removal, or filtering. This data is only available on images generated by a Profilm3D. Clicking on the red X next to a given layer will remove the effects of that operator. It is important to note that any subsequent operations will also be removed.

3.3.3 Preferences



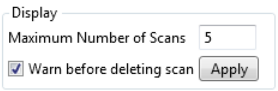
The **Preferences** ribbon provides options to customize selected aspects of the Profilm software to best match user needs.



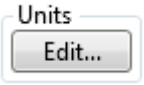
ScreenShot: Active tab only - Crops screenshot images so that only the active tab is visible. When disabled, the ribbons and any other open tabs will be visible in screenshots.

Offer to reprocess raw data - Provides a prompt to reanalyze the data when a new recipe is loaded. Otherwise, Profilm will apply the changes automatically. Only the most recently completed image or a loaded .fibcs file can be reprocessed.

This control can also enable the **Data Recording** and **Access Control** functions.



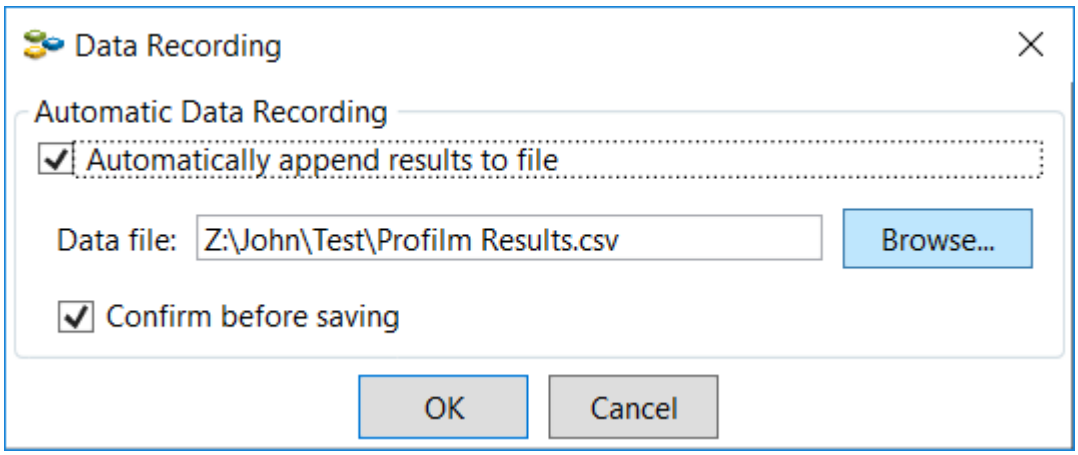
Controls the maximum number of scans the software will display at one time from 1-10. A warning notification prior to deletion of an active scan can also be enabled here.



Opens the **Edit Units and Numbers** dialog box to change units for motion control.

Data Recording

Data Recording provides a method by which results can be automatically exported into a .csv file for future use. To access this feature select the **Preferences** ribbon, then click on the **General** button under settings. A side panel will open. Click on the **Data Recording** button to access the dialog.



The Data Recording dialog box

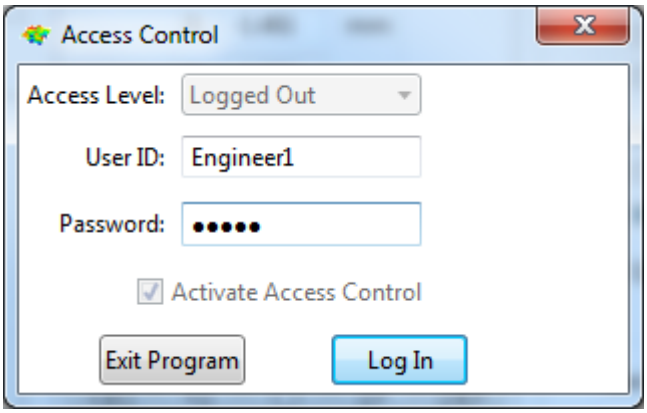
To enable, click the checkbox next to *Automatically append results to file*. Click the **Browse...** button to select the file to store the results. Select an existing file or create a new file. To have the software provide a prompt before adding results to the file, enable *Confirm before saving*. Click **OK** to save the changes, or **Cancel** to exit the dialog without saving changes.

With the feature enabled, Profilm will add the results from any analyses as individual lines in the .csv file until either a new file is selected through the dialog, or the feature is disabled. The following values are included with every analysis result written to the file: Recipe Name, Acquired Time, and X and Y position of the stage in microns. If other information is measured with the selected recipe, that information will be included as a separate column. This includes all step height, 2D area roughness, and 1D line roughness results.

Access Control

The Profilm software incorporates password protection to limit access to the measurement software and settings. When the software is initially installed, access control is turned off. The

software will default to *Engineer* level access, which enables access to all features of the program except turning on and off access control and adding and deleting users.



There are three access levels available; *Supervisor*, *Engineer*, and *Operator*.

Supervisor: Can access all features in the software, enable/disable access control, add new users, or delete existing ones.
Engineer: Can access all features available in the software, but cannot activate or deactivate access control or modify user accounts.
Operator: Can open previously created recipes but cannot edit the values. Has full access to analysis functions.

Turning on access control requires supervisor level access. The software is delivered with one user, a supervisor, in the list of authorized users as shown below:

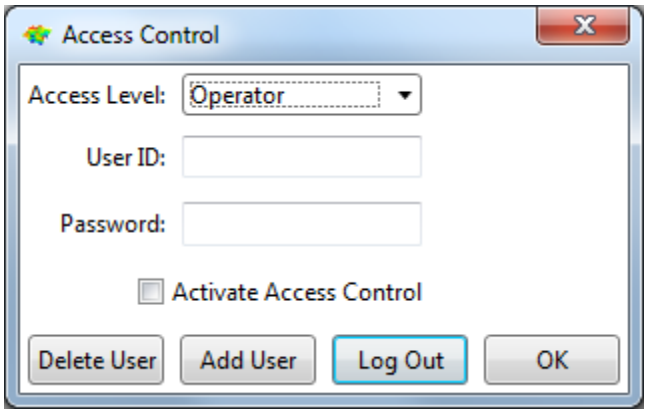
UserID: filmsuper

Password: filmetricsfff

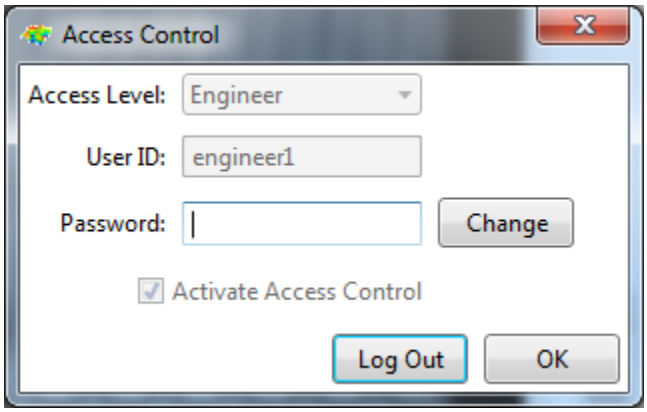
To turn on access control, log in as filmsuper using the password shown above. The **Activate Access Control** checkbox should selected, enabling access control functions. **User ID** and **Password** are case-sensitive.

To add a user, select the appropriate **Access Level**, enter a **User ID** and initial **Password**, and then click the **Add User** button.

To delete a user, enter the **UserID** and press the **Delete User** button.

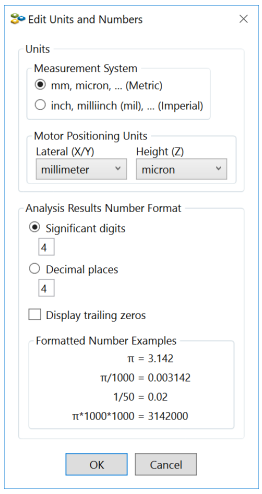


Operator and engineer-level users can change their passwords when they are logged in by entering their password into the password field and click the **Change** button. Supervisor-level users can change their password or the password of any other user by selecting the appropriate access level, entering the **UserID** and new password, then clicking **Add User**. If the program finds that a user already exists, it will delete the old entry for that user and create a new entry.



For maximum security, create a new supervisor-level user and delete the filmsuper user. In the event no supervisor-level user is able to log in (e.g., forgotten password), re-install the software and add all the users again.

Units

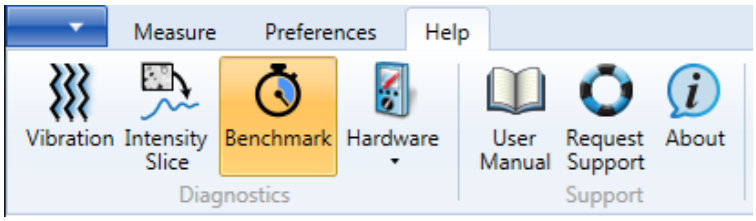


The **Edit Units and Numbers** dialog box allows the user to select from a list of metric or imperial measurement units for both the lateral (X/Y) and height (Z) directions.

Measurement units are selected using the various drop-downs shown in the display.

This dialog includes specifications for how the software displays numbers, including significant digits and decimal places used. As these values are changed, the *Formatted Number Examples* window will update to show the effects on measured results. By selecting the *Display trailing zeros* option, the software will add extra zeros to results as necessary to satisfy the selected significant digit and decimal place values.

3.3.4 Help



The **Help** ribbon contains the **Diagnostics** and **Support** tabs for use when troubleshooting. Most of these features will display under the Live window when active.

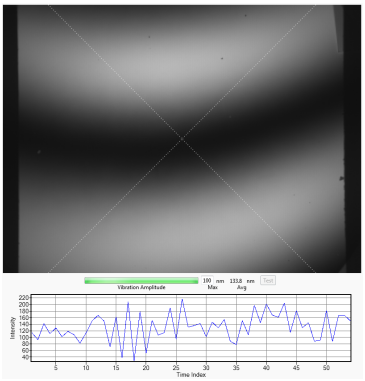
Diagnostics
Support

Diagnostics

Diagnostics functions are used to track the current status of the system, which is useful when troubleshooting measurement issues.

Vibration
Intensity Slice
Benchmark
Hardware

Vibration



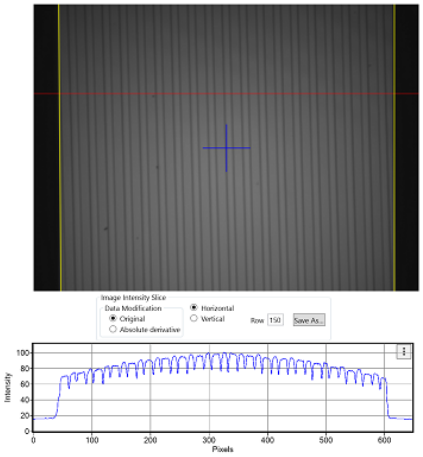
The **Vibration** tool is used to track the intensity of vibration detected by the system. This is displayed under the Live window, where an X pattern will be superimposed on the live camera image, along with a status bar and plot of the intensity value versus time.

Clicking on the drop-down button for the **Vibration** tool lists options and instructions to ensure the best possible measurement.For best results, the instrument should be on a clean, flat surface (the step height standard will work), and the surface should be flattened such that only 2 or 3 interference bands are present.

The status bar shows the highest current vibration intensity measured across the entire image. The **Test** button will move the z stage to confirm that vibration changes are being detected. The plot at the bottom shows the intensity plot versus time in milliseconds for the selected point (highlighted in red).

To update the plot, click on a new point or the same point again.

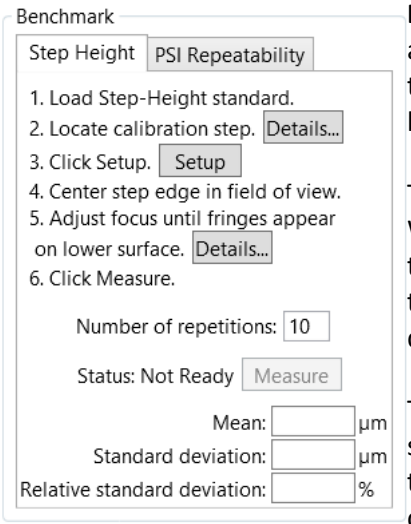
Intensity Slice



The **Intensity Slice** displays the current light source intensity at the detector on a line across the image field. To place an intensity slice, click at the desired location on the live camera image. The intensity will be plotted on the graph below. Alternate between a horizontal and vertical slice using the radio buttons beneath the live image field.

The maximum value the detector of capable of measuring is roughly 250 counts. Anything above that will saturate the detector. If the image is saturating enable the auto exposure function or lower the intensity slider to compensate.

Benchmark

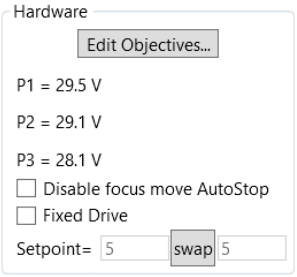


Benchmark checks that the Profilm3D is performing correctly after the system has been installed. There are two functions in this feature, **Step Height** and **PSI Repeatability**. Instructions for both are included in the software.

The **Step Height** function runs through an automated routine of WLI measurements to check the consistency of measurements on the step height standard. Results are output as the mean thickness of the measured surface and the measured standard deviation.

The **PSI Repeatability** function is a similar feature for the PSI scanning mode. The resulting repeatability number represents the change in Sq (RMS roughness) on the surface between consecutive measurements.

Hardware



The **Hardware** tab shows the current voltage status of the piezoelectric crystals that control the z-height of the measurement objective.

Edit Objectives defines various values for the objectives present on the system, such as turret position, magnification, etc. Other than turret position, values should not be modified without the assistance of Filmetrics.

P1, **P2**, and **P3** represent the voltages measured at the piezoelectric actuators. Ideally, the voltages values should be 30V ± 3V, though the system will still operate effectively at other values. If voltages drop to 0, or are showing 80 or above, contact Filmetrics for further assistance.

The **Disable focus move AutoStop** and **Fixed Drive** options should not be checked without the direct assistance of a Filmetrics employee.

When shipping the system, click the dropdown arrow to access the *Hardware Details* dialog box, which contains the **Move Z to Shipping Position** button. Click to move the head to the appropriate position for shipping. A prompt will appear as a reminder to remove all objectives before completing the move.

Support

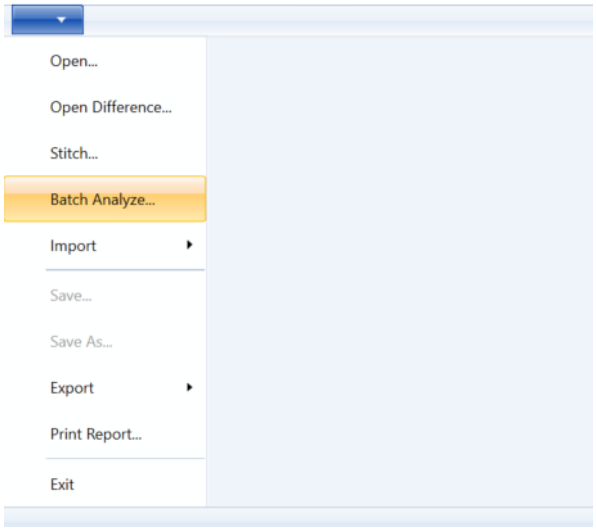
The **Support** tab is a resource for learning more information about the Profilm software, and lists options for contacting Filmetrics for support.

Provides access to an in-software copy of the user manual.

Launches a form for submitting a support ticket. If the computer connected to the instrument has access to the internet, the software will be able to submit support requests directly to Filmetrics. If not, a support request file can be generated and saved by clicking the **Create E-Mailable File...** button. E-mail the resulting file (.fipsa file type) to swsupport@filmetrics.com.

Displays software information including the current system serial number (if used with a Profilm3D), software version, and any enabled features. Have this information available when contacting Filmetrics for support to help us better assist you.

3.4 Home Menu



The **Home** menu contains basic controls for handling data coming into and out of the Profilm software. The **Open Difference** feature compares two images to see how the sample has changed between measurements. If the UPG-Stitching software has been purchased, the Stitching function will be available here.

Open:	Opens a previously saved file. A list of currently supported file types can be found in the appendices (Supported File Types).
Open Difference:	Opens and compares two different images, then generates a difference image. Used to compare pre- and post-process samples.
Stitch:	Opens the Stitching dialog box. UPG-Stitching required.
Import:	Imports recipes and license files.
Batch Analyze:	Load multiple scans, analyze them concurrently with a selected recipe, then save them as new files.
Save:	Saves the currently active image.
Save As:	Saves the currently active image under a new name.
Export:	Exports recipes, license, and system info files.
Print Report:	Prints out a report from the currently open images for presentation to clients. Explained in detail in Measurement Report.

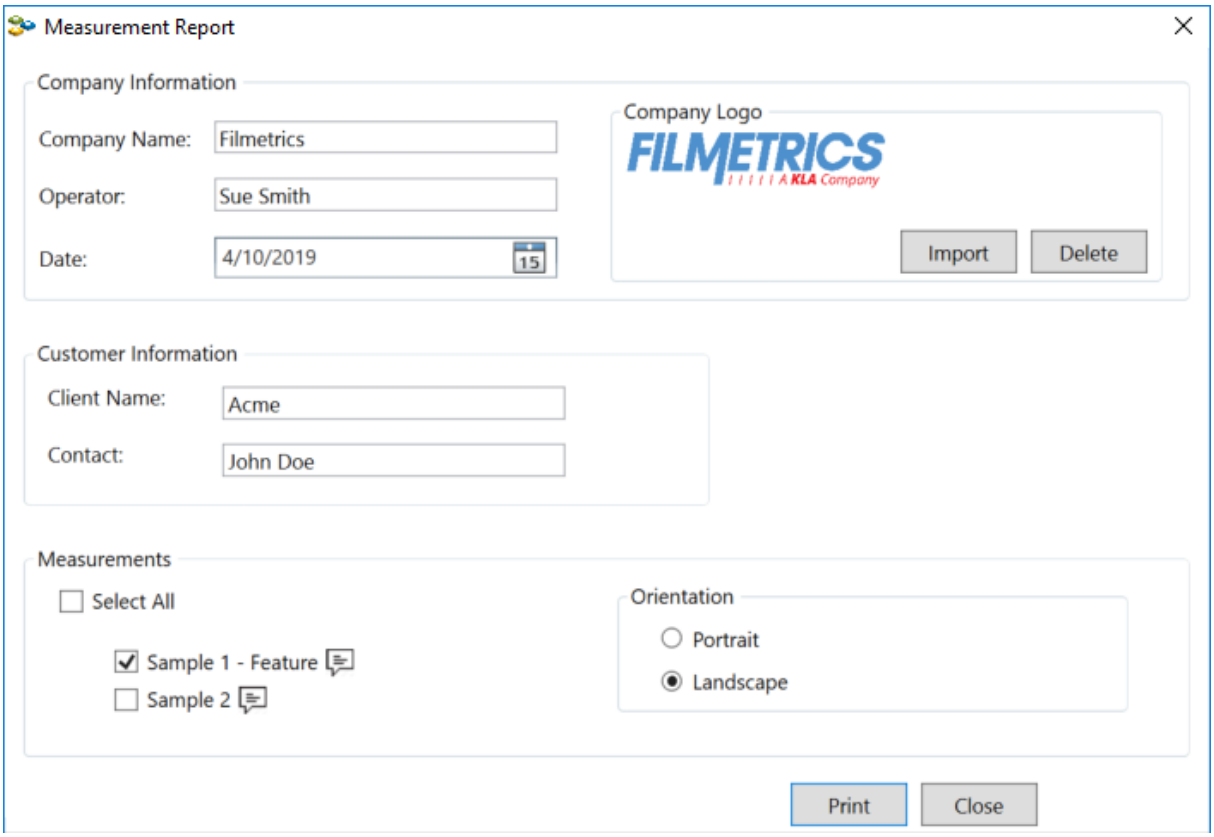
Exit: Exits the software.

3.4.1 Measurement Report

Selecting the **Print Report** option opens the *Measurement Report* dialog to automatically output your results for distribution to clients and colleagues.

Company Information refers to the entity that generated the data. Include an image of the company logo on the report using the **Import** button or **Delete** the currently loaded image. *Customer Information* refers to the recipient of the report.

Measurements automatically populates with any currently open images in Profilm. Click **Select All** to add all images to the report or select them individually using the check box next to the sample name. Click the comment icon next to an image to add additional context for the image. Select an output *Orientation* using the *Portrait* or *Landscape* radio button. Click **Print** to print the report, or **Close** to exit.



Part

4 Editing Recipes

In Profilm, **Recipe** files are used to control how the software will acquire and analyze the data from a given sample. This section will explain how each setting works, and best use for a given application.

Each recipe is made up of two parts, the Scan and Analysis settings. These are accessed in the Measure ribbon under the Measurement Recipe tab by clicking the **Edit** button next to the **Recipe** drop-down. Some settings, such as *Backscan* and *Scan Length*, are also available in the main Live Window.

Hundreds of individual recipes can be saved in the Profilm software to match applications as needed.

4.1 Creating a New Recipe

Profilm will ship with at least one default recipe for you to use that can be modified to suit different samples and applications. The following are step-by-step instructions for creating a new recipe from scratch using the Profilm software, starting with assigning scan settings.

4.1.1 Assigning Scan Settings

To begin, click on the Live tab to access the Measure ribbon.

Step 1: Accessing the Scan Settings.

Click on the **Edit** button in the **Scan & Analysis Recipe** section to open the **Edit Recipe** dialog box.

Step 2: Selecting Measurement Type.

With the **Recipe** dialog box open, select to scan in **WLI**, **PSI**, or **Composite WLI/PSI** mode. The WLI mode will be appropriate for most applications due to its robust capabilities. PSI should be reserved for situations requiring higher resolution, as its much higher precision is limited by a smaller scan range capability (0-3 µm) and greater sensitivity to surface irregularity. Profilm's unique **Composite WLI/PSI** mode can be used to expand the scan range even further, up to 10 µm. For in-depth details for these modes, see the WLI, PSI, and Composite WLI/PSI sections under the appendices.

Step 3: Selecting a Scan Speed.

There are two scan speed options, **Fast** and **Normal**, which determine the speed at which the piezoelectric assembly moves when performing a scan. **Normal** is recommended for best results with most applications. The **Fast** scan setting can be used for larger scans to decrease measurement time, but with a greater risk of invalid points in the data.

Step 4: Select a Zoom Setting.

Increasing the zoom focuses more tightly on a given feature without having to change the objective on the Profilm3D. For more information about how zoom affects a given objective, see the Objective Specifications page in the appendices.

Step 5: Setting your Backscan, Scan Length, and Scan Averages.

The last step in adjusting the scan settings is to determine the vertical range the scan should cover, and how many scans should be taken. The backscan determines how far the head of the Profilm3D will be moved down towards the stage using the stepper motors, while the scan length determines how far the objective will be raised by the piezoelectric assembly to complete the scan. Data is not collected during the backscan, and should be modified to ensure the scan length covers the entire range to be measured.

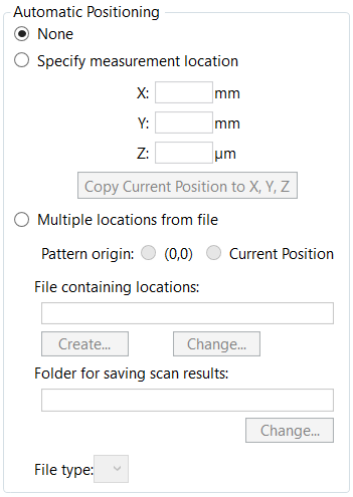
For example, when measuring a sample with an eight micron step that is focused to the bottom of the step using a backscan of five microns combined with a fifteen micron scan length will ensure measurement of both the lowest and highest points on the sample, while still giving a reasonable overall measurement time. However, setting a scan length of eight microns with the same backscan length would not include the top of the step as a result as it would still be five microns above where the scan ends.

To simplify this process, the **Scan Top** and **Scan Bottom** controls in the **Live** tab can be used. These controls set the top and bottom of the scan based on the position of the Profilm3D head. To use the above example, first focus to the bottom of the step and then continue focusing down until the last interference fringe disappears from the image. For this example lets say that occurs at a position of 10 um. Click the **Scan Bottom** button, which will then display 10 um as the bottom of the scan. Next, focus up through the surface until the last interference fringes pass the top of the step, say at 20um. Click **Scan Top** to set the top of the scan, displayed as 20 um. The *Scan Length* will update to match the difference between this values. Note that the *Backscan* is ignored when using this feature, as the scan will always begin at the bottom value and end at the top.

After the backscan and scan length or scan top and bottom are set, select the number of scans to complete before averaging all scans into one. Increasing the number of scans will help to eliminate noise from scans, but will increase measurement time.

If the stitching upgrade has been purchased, see Setting Up a Scan Grid and Stitching Together Multiple Scans sections for additional instructions to complete a stitched scan.

4.1.2 Automatic Positioning



This option provides the user with a greater degree of control over the measurement location. When enabled, select to measure one specific stage location or multiple locations based on a comma-delimited text file of coordinates.

Measurement units used for the specific location are selected under *Motor Positioning Units* in the Edit Units dialog.

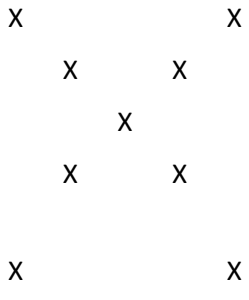
To use *Multiple locations from file*, select to start the pattern from either the (0,0) position on the stage, or the current position using the *Pattern Origin* radio buttons. Then select to **Create** a new rectangular grid pattern file or **Change** to an existing input coordinate text file. To create a custom pattern file without using an input coordinate file, see the instructions below.

Similar to the Grid function, an output folder must be selected to save the multiple image files created during the mapping process. Click **Change** to select the output folder, then use the *File* type: drop down to select the type of file to be created (.fibps, .fibcs, or .sur).

Creating a custom pattern file:

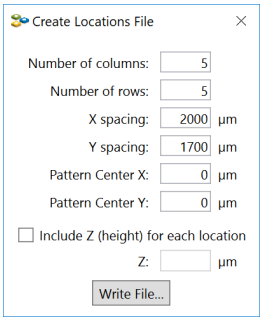
Some applications will require a pattern file in a shape other than the standard rectangular grid.

Follow the example below to create a new rectangular grid pattern file, then edit the file to create a custom pattern. For this example, we want to measure locations that are in an X-shaped pattern, covering a 10x10 mm area, using a 10x objective.



Target pattern

Step 1: Create the pattern file.



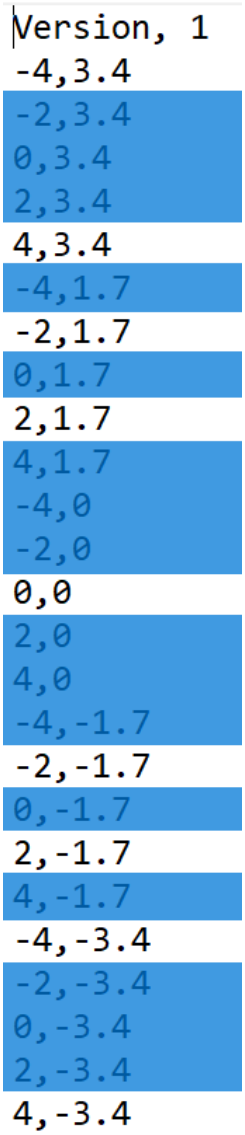
- Click **Create** to open the *Create Locations File* dialog box.
- Set *Number of columns* and *Number of rows* – For this example, set both to 5.
- Set *X spacing* and *Y spacing* – Set X spacing to 2000, and Y spacing to 1700. These values reflect the field of view for a 10x objective at 1x zoom.
- Set *Pattern Center X* and *Y* (determines the starting point for the pattern) – For this example we set both to 0.
- Set *Z (height) for each location* (this determines the focus height at each point) – For this example we will leave this field blank.
- Click **Write File**, then save the file with the name **"Custom X - 10x"**.

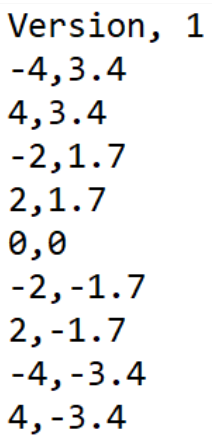
Step 2: Open the pattern file.

Using a text editor (e.g. Notepad), open the file **Custom X - 10x.txt**. (Note: if the file does not appear in the *Open* dialog box, choose **All Files(*.*)** under *File Type*.)

Step 3: Edit the pattern file.

Starting in the second row, the values shown are x-y coordinate values in mm. Note that these values are always shown in mm, regardless of the unit selected during file creation. The camera will center on, and take measurements, at these locations. To create the "X" pattern, delete the highlighted rows in the image shown below.





Custom X - 10x opened in Notepad.

Custom X - 10x after removing extra coordinates.

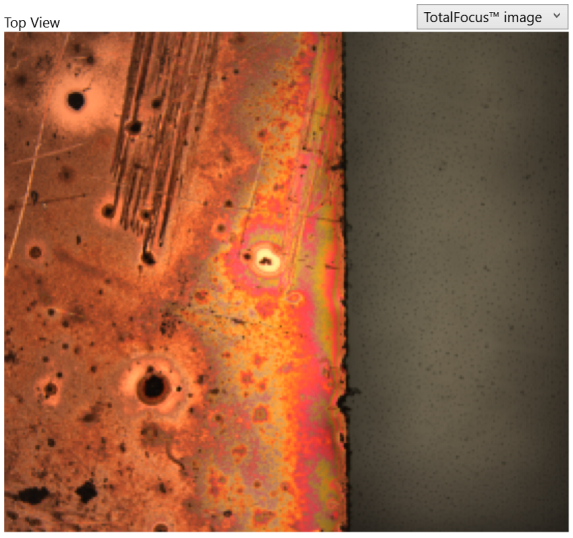
Step 4: Save the pattern file.

After modifying the text file, select **File > Save** to complete the process.

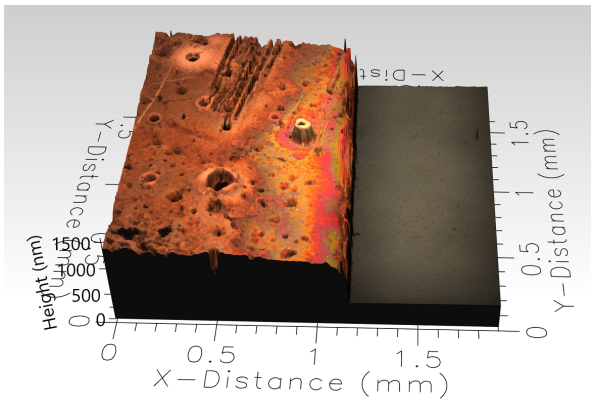
4.1.3 TotalFocus Image

TotalFocus™ Image combines measured height information with the image frames from the camera to create a large depth-of-field image of the sample.

TotalFocus™ Image is available for systems with the appropriate upgrade (UPG-TotalFocus or UPG-ColorImaging). Use the dropdown menu above the completed image *Top View* to alternate between the height view and the Total-Focus™ image.



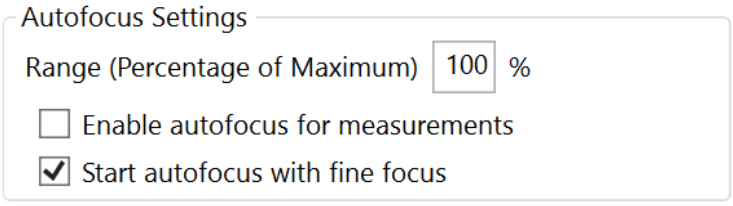
Example Top View image of a copper film on a glass slide. The rainbow colors on the copper indicate the presence of a thin film near the step edge.



The 3D View image of the same copper on glass sample.

When the color option is selected, three additional scans are taken with a red, green, and blue filter. Color and monochrome are available for Profilm3D systems with serial numbers beginning with 18K or later. Monochrome is available systems manufactured prior to 18K.

4.1.4 Autofocus Settings



The *Autofocus Settings* options allow users to set the range covered during the routine, autofocus each time a scan is taken, and to start with a fine focus search.

The *Start with fine focus search* setting is used to speed up the autofocus procedure when already close to a proper focus on your sample. Instead of going through the full range as described below, it will take a quick look over a smaller scan range to find proper image focus. If this fails, it will then attempt a coarse focus scan to find the proper focus height. When disabled, the software will always start with a coarse focus search.

The total range covered during the autofocus function is determined by the objective currently in use on the Profilm3D. Lower power objectives like the 5x will have a larger total range than higher power objectives like the 100x. Changing the value in the *Range (Percentage of Maximum)* text box will limit (or expand) the range covered by the focus process. A smaller range will speed up the autofocus process, though risks not containing the proper focus height. For this reason, it is suggested to use the coarse focus controls in the Live window to roughly orient on the sample surface before using the autofocus feature to precisely hone in on the sample.

Selecting *Enable autofocus for measurements* will cause the software to autofocus whenever the **Start** button is pushed. This feature is most useful when measuring multiple samples of similar but not identical heights, or when using the Grid or Automatic Positioning functions to measure over a large area.

4.1.5 Selecting Height Formation Settings

Height Formation Settings are used to determine how the data is processed after the scan is complete.

WLI Settings:

These options determine how the software analyzes the measured data to determine z position when taking a WLI scan. *Centroid* is the default setting and should be used first as it gives the most reliable results. *Envelope Peak* and *Envelop Center* may produce fewer invalid pixels if the optical interference signal is low due to low reflectivity or large variation in surface height within a single pixel, such as on rough or tilted surfaces. Note the latter options may give less accurate results overall.

When using the *Envelope Peak* and *Envelope Center* methods, an *Envelope Threshold* is required. This value sets a minimum amplitude variance required in order for the model to attempt to fit the data. A lower value may increase the number of measured pixels, but at a greater risk of spurious measurements.

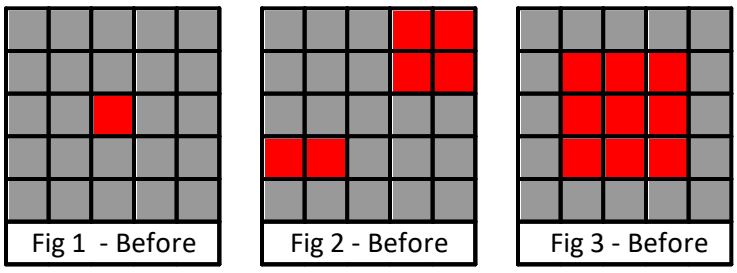
Enabling the **SNR Enhance** function allows for specifying how the software processes the signal-to-noise ratio during the scan. The default value is 2%, and can be modified as desired. A larger value may decrease the number of invalid measurement points in the resulting image, but would increase the time required for analysis.

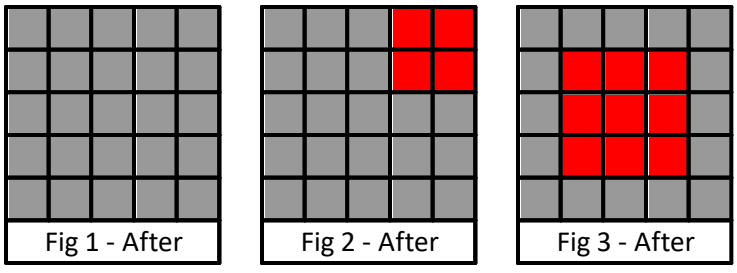
Post Processing:

Post processing accounts for points that may not have measured correctly due to vibration, excessive curvature, or other issues through two primary options. Points that are left uncorrected will present as gray pixels in the resulting image.

Invalidate clusters: In some regions where measurements are generally invalid, it may be possible that some isolated pixels and/or small clumps of pixels are not detected as invalid but actually have an arbitrary or meaningless height value due to excessive drift or noise. In such cases this feature removes these pixels from the resulting scan by marking them as invalid.

Control for this feature is set using the *Cluster area* dialog. The number entered refers to the largest contiguous grouping of pixels the software can remove. For example, the three figures below represent a 5x5 pixel grid of invalid pixels with various valid points included. By setting the maximum region size as 3, the following corrections should occur:



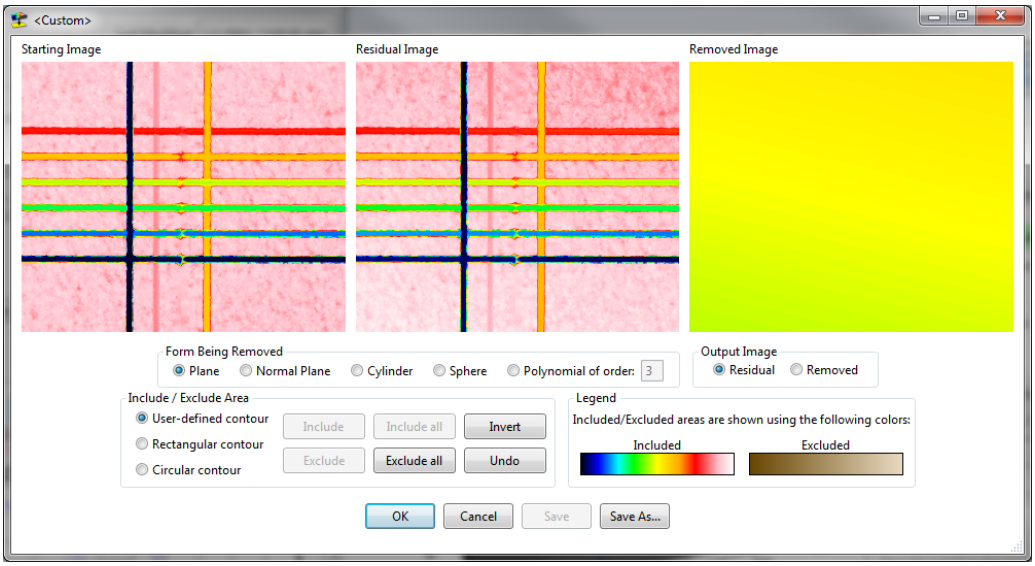


When acquisition settings have been modified to suit the application, switch to the **Analysis** tab in the **Recipe** dialog box to modify the analysis settings.

4.1.6 Data Conditioning

Data Conditioning refers to the various operators available in the software for modifying data. Any number of operations can be applied to a given image in any order. Operators can be added using the green plus buttons, and subtracted using the red x buttons.

Initially the only way to add an operator to the recipe is to complete a scan, then apply the desired operator to that scan. Once that is completed, the **Recipe** will automatically update with the *Operation* type and an *Operation Settings* value of <custom>. Clicking on the **Edit** button, opens a dialog for that given operator. In the example below, the Flatten dialog is shown.



The important addition to this version of the dialog is the **Save** and **Save As** buttons at the bottom. These can be used to save the settings selected for this operator as a new file, which will then be available in the drop-down menu for all future recipes. These custom files are saved into one of eight different file types, based upon the selected operator.

.fibos: Spatial Filter
.fiboi: Fill In Invalids
.fiboo: Remove Outliers
.fibol: Level
.fibor: Flatten
.fibof: Filter
.fibft: FFT Filter
.fiboc: Crop

When desired operators have been set, continue to the last part of the recipe; Selecting the Analysis Method.

4.1.7 Selecting the Analysis Method

The **Analysis** method refers to the type of data to be extracted from an image. As with the Data Conditioning operations predefined settings files are needed to take a scan and then to apply the desired analysis options in the Analyze ribbon for updating the recipe.

This function works differently than the **Data Condition** section in that all changes to the settings in the analysis function must be done outside of the recipe. Clicking the **Edit** button will open a notice

to exit the recipe and change the settings within the recipe controls in the main Profilm window. Once analysis options are set, click the **Save As** button to save for future use.

Example: When measuring the Sa and Sq per the ISO 25178 standard on a variety of samples, first open or complete a scan, then go through the procedures as listed in the Making Measurements - Area Roughness section. To avoid repeating the whole process for every scan, click the **Edit** button to open the **Recipe** dialog, navigate to the **Analysis** tab, then hit the **Save As** button for the *Area Roughness* settings. Give the settings file a meaningful file name (e.g., Sa + Sq - Gaussian - 80 um) and click **Save**. The saved settings will now switch from <Custom> to the new file name, and can be added to any recipe.

Only one analysis function may be active in the recipe at a given time. To apply different types of analysis to the same image, create different recipes (e.g., one for step height and another for 3D roughness) and switch between them.

4.2 Loading a Saved Recipe

Once multiple recipes have been created, select different recipes to match various applications. To load a saved recipe, go to the **Scan & Analysis Recipe** section in either the Measure or Analyze ribbon, then click on the drop-down arrow next to the currently loaded recipe. A new dialog box will open showing all available recipes. Click on the name of the recipe to use, then click the **OK** button. If an image is currently loaded, it will automatically be processed using the new recipe analysis settings.

Part V

5 Making Measurements

This section will discuss how measurements are made on the Profilm3D, split into two sections; acquiring an image, and then viewing and analyzing the data.

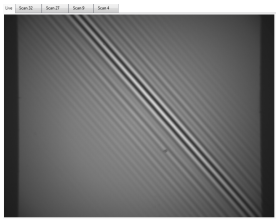
Acquiring an Image:
Focusing the Profilm3D
Adjusting Tip/Tilt
Collecting the data
Setting Up a Scan Grid
Stitching Together Multiple Images

Viewing and Analyzing an Image:
Controlling the 3D View Camera
The Slice Tool
Dimensions
Measuring Step Height
Area Roughness
Line Roughness

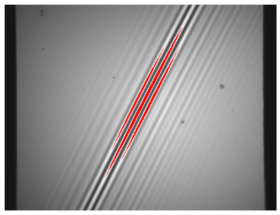
5.1 Acquiring an Image

5.1.1 Focusing the Profilm3D

The first step in measuring a sample is to load a sample onto the stage, then bring it into focus. This is accomplished using the Z motion controls under the Live tab, or by pressing the **Autofocus** button. In general, it is suggested to use the coarse focus options (double arrows) to get close to a proper focus, and then to use the autofocus routine. Alternatively, the image can be focused manually using both the coarse and fine focus (single arrow) controls. When the system is in proper focus interference patterns will be visible on the sample. See the examples below:



This image shows a sample that is in focus, but with low flatness.



This sample is currently overexposed.

When focusing through the sample, red pixels may appear in the live video image. This indicates that the spectrometer is saturating at those pixels. To account for this, either enable the **Auto Exposure** feature or lower the light source intensity using the slider.

It is generally suggested to use a fixed exposure value for best results. To fix your exposure, first focus on the sample until several interference fringes are visible in the camera field. Next, toggle on **Auto Exposure** if it is not currently active as well as **Lock Peak**. While enabled, **Lock Peak** will continuously check the video and lock in the exposure value for the highest intensity signal seen.

After that, set the light source intensity using the slider. Slowly focus up and down through the sample. Once there are no longer red pixels appearing in the image field, reset to your initial scan position based upon your back scan and scan length settings.

For samples with a wide dynamic range it may be best to leave **Auto Exposure** enabled through the scan. If **Lock Peak** was previously enabled toggle it off, and click **Reset Peak** to allow exposure settings to change during the scan.

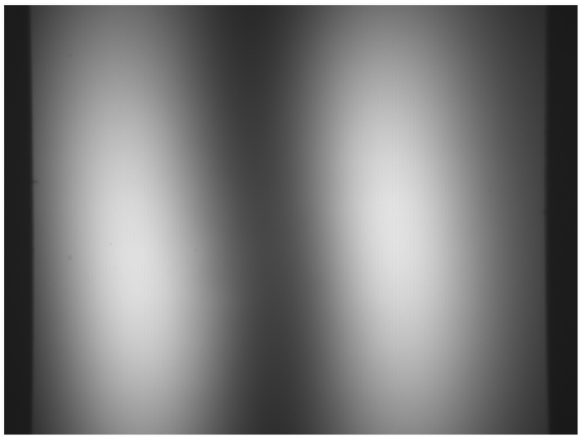
The width of the interference bands will give an indication of the current flatness the surface, with tilted surfaces having narrower bands and flat surfaces having wide interference bands. The example image above would be considered to have a high degree of tilt and should be adjusted using the tip/tilt settings on the stage to generate a better image. This process is described in the next section.

Continue to Adjusting Tip/Tilt.

5.1.2 Adjusting Tip/Tilt

With the sample loaded onto the stage and focused, next level the sample stage to make sure the surface of the sample is as flat as possible, relative to the objective. To achieve this, the Profilm3D has a tip/tilt mechanism built into the sample stage which consists of two adjustment knobs, one on the front edge of the stage, and one on the right edge. These knobs lift or lower the stage around a centralized anchor point. Rotating the adjustment knobs in a clockwise direction will lower the respective edge of the stage. Rotating them in a counter-clockwise direction will raise it. The easiest place to level a sample is the center of the stage directly over the anchor point.

Example of a highly tilted sample.



The same sample, flattened using the tip/tilt adjustment.

Making Adjustments Using the Camera Image.

Select one of the two adjustment knobs, and turn until the interference pattern is oriented in a perfectly horizontal or vertical pattern. Refocus the stage in between adjustments, as the position of the stage changes relative to the objective with each knob turn. When the knob is turned clockwise, the focus should be lower than it was previously and higher if turned counter-clockwise. The image above at left illustrates proper alignment of the front stage adjustment knob.

With the interference pattern aligned in the vertical or horizontal direction, and therefore aligned on that axis, no further adjustments with that knob are needed. Begin adjusting the other knob to maximize the width of the interference fringes. Focus through the sample using the fine focus adjustment controls, paying attention to the direction the fringes move to determine the direction to turn. For example, if while focusing down through the above left example image the interference pattern moves from the right of the image to the left, it would indicate that the right side of the stage is higher than the left. To adjust, turn the knob on the side of the stage clockwise to lower the right edge and level out the sample.

As with the first knob, refocus between adjustments using the same guide to determine focus direction (lower for clockwise, higher for counter-clockwise). The image above to the right shows an example of a sample that has been flattened to a much higher degree using the tip/tilt adjust. For smooth, flat substrates, it is possible to level the sample such that there is no full oscillation on the screen, only a minima (image is nearly black), or maxima (screen is nearly white) across the whole image field.

For non-flat samples, such as spheres, cylinders, or wavy surfaces, perfectly vertical or horizontal fringes are not possible. In these cases use the relative motion across the camera image to determine the direction to move the stage.

When the sample is flat, it is ready to scan. If scanning an area larger than the current objective can handle, Set up a Scan Grid to automatically map out the scan. Otherwise, continue to Collecting the Data

5.1.3 Collecting the Data

As an example for acquiring a scan, the following recipe will be used. This recipe is for the SHS-CrOnSi-10um step height standard, called *Step Height*. For details on how to set up custom recipes, see the Editing Recipes section.

First, select the *Step Height* recipe by clicking on the drop-down arrow next to the current recipe in the **Scan & Analysis Recipe** section. Then select *Step Height* from the list. Review and adjust the scan settings if necessary to match the example settings below, then click **OK** to apply the settings.

Measurement:	WLI
Scan Speed:	Normal
Zoom:	4x
Backscan:	15 µm
Scan Length:	30 µm
Scan Averages:	1
WLI Settings:	Centroid
Post Processing:	None

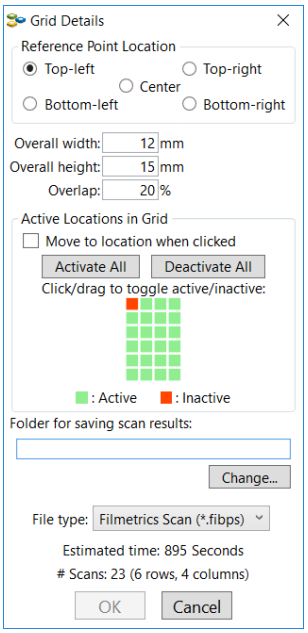
Click the **Start** button to begin collecting data. The interference fringes should move through the sample as the system completes the scan, tracked by a progress bar above the video image. When the scan is complete, a new tab will open with the resulting data.

Continue to Controlling the 3D View Camera.

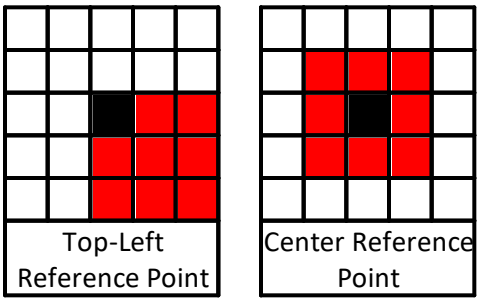
5.1.4 Setting Up a Scan Grid

Some applications may require a scan that is larger in area than the objective on the Profilm3D can image. In this case, use the **Grid** function to map a larger area to be analyzed or stitched later. Note that stitching within the Profilm software can only be completed only if the UPG-Stitching software package has been purchased.

First, define your scan area using the **Grid Details** dialog box, accessible in the **Scan & Analysis Recipe** section after clicking the **Edit** button. Next, click the checkbox next to the **Grid** option and then click on the **Details** button.



Select a *Reference Point Location* to indicate the origin point for the grid. For example if top-left is selected, the current location will correlate with the upper left corner of the resulting grid.



Next, enter values for the desired scan size and the percent overlap between individual scans. Some overlap between scans is required in order for the stitching algorithm to work. An overlap of 20% is suggested for best results. Once the values have been entered, an estimate is shown for the number of scans and associated time required to complete the full grid.

A visual representation of the grid will also appear under *Active Locations In Grid.* Click or click and drag on the colored squares to toggle them to active/inactive in the grid scan. If *Move to location when clicked* is enabled the x-y stage will also move to the corresponding location, which can be

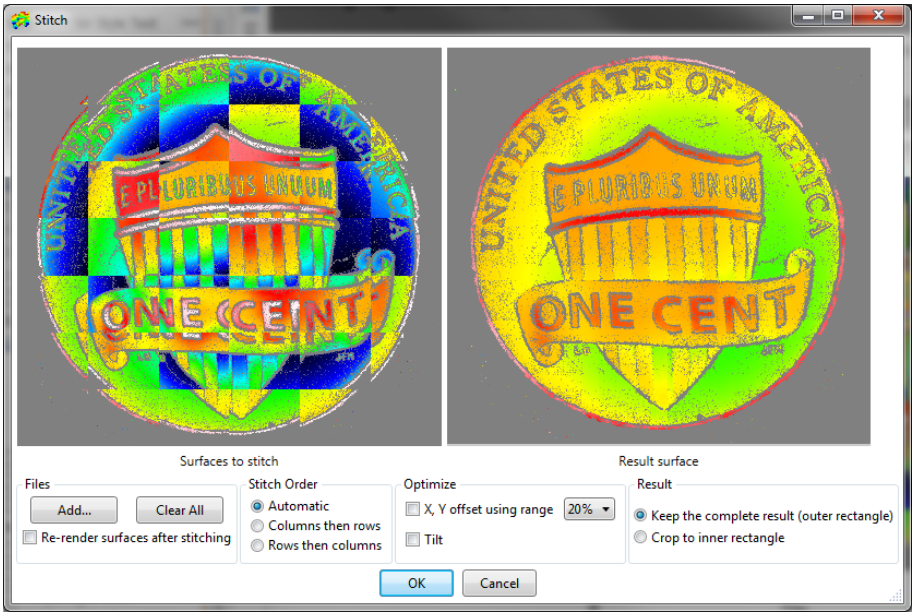
helpful to insure the desired region or feature is included in the current defined region. **Activate All** and **Deactivate All** will toggle all points on or off.

Last, select a location and file type to use for the saved scans. Files are saved to the selected location as they are completed. With the stitching upgrade from Filmetrics, any file type may be used. For use with a third party software package, like DigitalSurf's MountainsMap, use the .sur file format. Click **OK** to save the changes.

Click the **Start** button to begin the grid scan.

5.1.5 Stitching Multiple Images

To measure a sample or feature that is larger than the objective image area, Profilm has an optional upgrade (UPG-Stitching) that allows enables the stitching of multiple images into a single large image. Use the instructions below to generate a stitched image.



An example of a stitched scan

1. **Acquire scans**: Before stitching a composite scan, it is necessary to first generate the individual pieces of the image. This can be accomplished using the Grid function, or by manually moving the sample using the stage and acquiring a group of scans. When manually acquiring data, leave some overlap between scanned images (20% is recommended) for the stitching algorithm to lock onto.

2. **Open the Stitching dialog box**: In the Home menu, select **Stitch...**

3. **Select the images**: Select images by clicking on the **Add...** button. Multiple files can be selected as a group by holding down the shift key and clicking on the first and last file in a list, or by holding down the ctrl key and individually clicking on the desired scans.

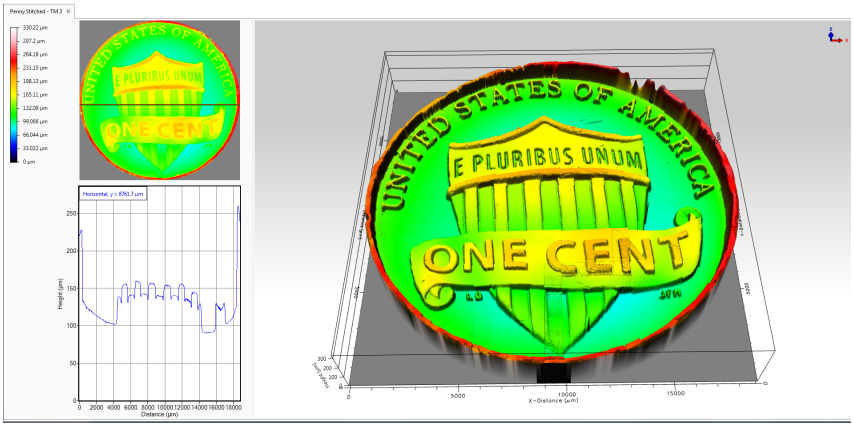
4. **Removing unwanted scans:** Some images may not be needed for the stitching process. These images can be removed from the stitching process by right-clicking on the tile in the *Surfaces to stitch* image and choosing to remove the single tile, or all tiles in the same row or column.

5. **Set the Stitch Order:** Depending on the nature of the surface, changing the order the software stitches the image can result in a better overall fit. Select *Automatic* to have the software decide which routine to use, or select to stitch by row or columns first.

5. **Enable optimizations:** Under the *Optimize* field, click the checkbox for the software to hunt for the best overlap using a value from the selected drop-down, or leave the box unchecked and it will use a standard routine. Uncorrected tilt can be optimized for in the scan as well. Click the **Update...** button to see the resulting stitched image.

6. **Set the crop:** The *Result* field offers two options for data output; complete result, represented by the *Result surface* displayed, or crop to the inner rectangle. Crop to inner rectangle will try to find the largest contiguous measure area in the stitched data, while excluding the unmeasured sections that may result from the stitching process.

7: **Output the stitched image:** Click **OK** to complete the stitching process. The stitched image display in a new tab and can be saved as a single file.



The completed stitched image

5.2 Viewing and Analyzing an Image

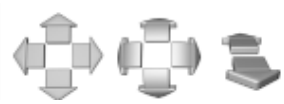
5.2.1 Controlling the 3D View Camera

Once a scan is completed, a new 3D image will be generated. Interact with the *3D View* using the controls below:

1. Left-click and drag to rotate the image. The camera will rotate around the current position of the mouse, indicated by a marker on the 3D view.
2. Right-click and drag to pan the image.

3. Turn the scroll-wheel to zoom; scroll up to zoom in, scroll down to zoom out. The zoom will center over the pixel the mouse cursor is currently hovering over.

Button control of the camera can be enabled using the 3D General Settings dialog, accessible through the Display Settings tab under the Analyze ribbon. These buttons will display in the lower left corner of the 3D scan, and can be clicked to rotate, pan, and zoom the image.



The software supports the SpaceMouse™ 3D mouse system for 3D image manipulation.

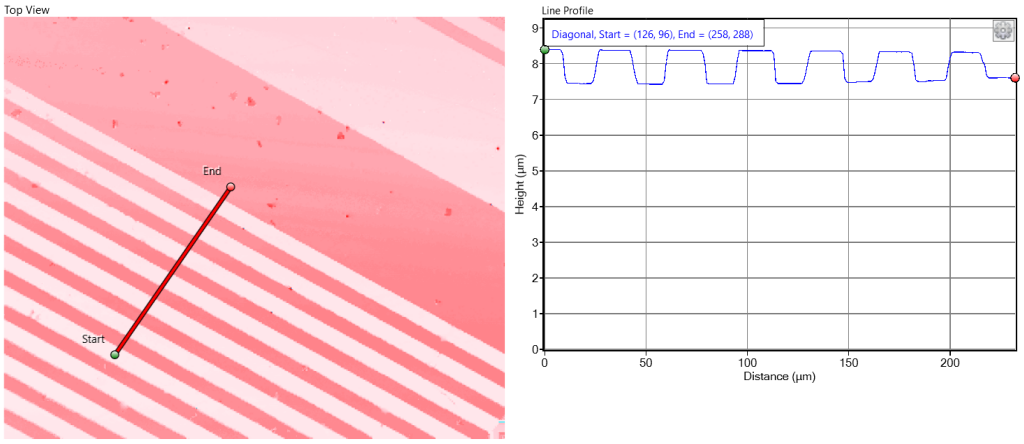
Clicking **Show Step Plane** will prompt the software to display the currently selected step height plane from the *Top View* on the 3D model, but only if the Step Height or Slice function is enabled. Clicking **Show intersection plane** will show where the step height plane meets with the measured surface.

In the upper right corner of the *3D View* will be a XYZ origin plot to maintain orientation while moving around the sample. The plot is anchored at what would be the front left corner of the sample image, which correlates with the bottom left corner of the *Top View*. To revert to this view, click the **Default Camera View** button to reset changes that have been applied to the 3D model.

The **3D Lighting Settings** dialog controls the lighting settings for the model. These values can be set to user preference, and the image will update as changes are made. To reset to the default setting (*Relative to Camera*, *Azimuth* 0, *Elevation* -45), click the **Default Light Orientation** button. Relative light intensity is controlled using the *Light Intensity* slider. Default light intensity is 100%.

5.2.2 Slice

The **Slice** tool provides a view of the line profiles without adding analysis to the data. Slice uses the same methodology as the Step Height - Line and Line Roughness functions. A horizontal line is drawn at the midpoint of the surface. Click and drag on the line, or the end points, to resize or move. Clicking on the end points switches to diagonal mode. Switch between the other modes using the drop-down arrow below the **Slice** button, or using the keyboard commands for horizontal (ctrl+H) or vertical (ctrl+V). If the slice plane passes over invalid data, that section of the line profile is shown in gray.



Example Slice line segment using the diagonal mode.
Note that the x-axis scale will automatically adjust to match the length of the segment in diagonal mode.

5.2.3 Dimensions

Measuring Line Segments

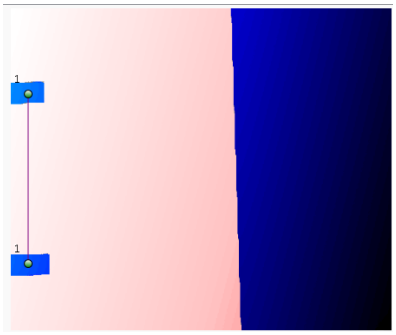
First, click on the **Dimension** button under the Analysis tab. A new side panel will open. Click the **Add Line** button and a line segment will be drawn on the *Top View* window, as shown in *Example 1* below.

To measure more than one feature at a time, click **Add Line** again to draw a new line segment, as shown in *Example 2*.

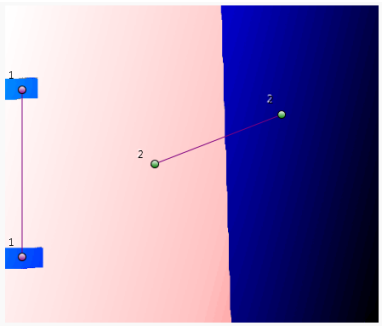
To move the line segment, click on either of the end points indicated by the matching number above the dots and drag it to the location of interest. The currently active line segment is indicated by the endpoints being highlighted in green. Unselected line segments will be shown as purple.

With line segments placed, results of the linear measurements will be visible in the *Dimension Measurements* in the side panel. Results for the change in x, y, and z positions, as well as total line segment length will be displayed in the units previously selected in Preferences.

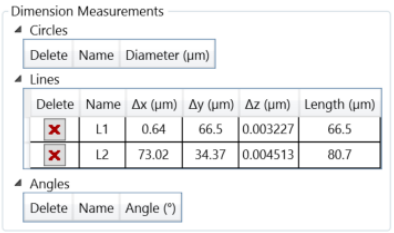
Hide or show the results by clicking the directional arrows next to *Circles, Lines,* or *Angles* in the *Dimension Measurements* window. To remove a line segment select it and click **Delete Active Line**, or hit the red X next to the segment name under the *Dimension Measurements*.



Example 1: Line segment in Top View



Example 2: Multiple line segments can be displayed simultaneously



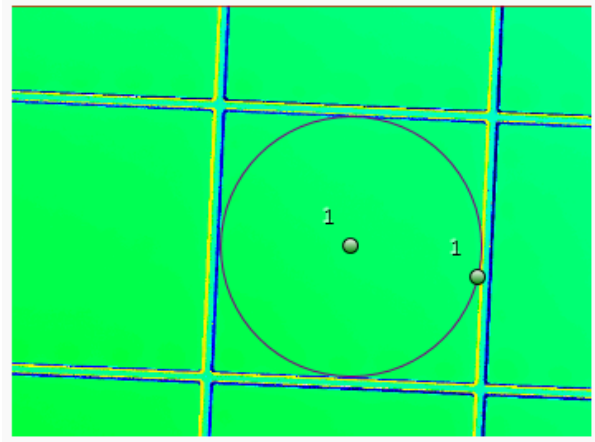
*Example 3: The **Dimension Measurements** table*

Circular Measurements

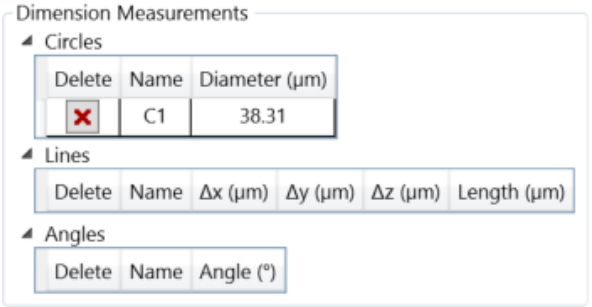
Click on the **Dimension** button to open the dimension side panel, then click **Add Circle**. This will generate a circle in the center of the *Top View*. Move the circle by clicking and dragging the center point, or resize the circle by clicking and dragging the outer point.

Multiple circles or line segments can be shown simultaneously by repeatedly clicking **Add Line** or **Add Circle**. To remove unwanted data select the segment or circle to be removed, then select **Delete Active Line** or **Delete Active Circle,** or click the red X next to the segment name under *Dimension Measurements*. The selection points that are highlighted in green will indicate the currently active line and circle.

The diameter of the circle will be shown in the *Dimension Measurements* table.



Example 1: Circle measurement in the Top View



*Example 2: The **Dimension Measurements** table*

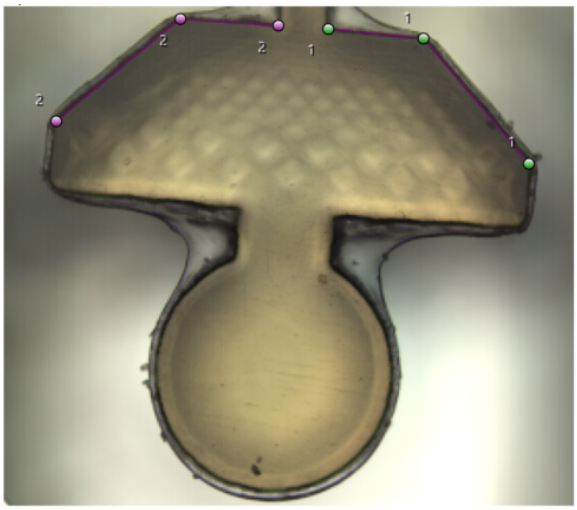
Measuring Angles

Click on the **Dimension** button under the Analysis tab. A new side panel will open. Click the **Add Angle** button and an angle will be drawn on the *Top View* window, as shown in *Example 1* below. To measure more than one feature at a time, click **Add Angle** again to draw a new angle.

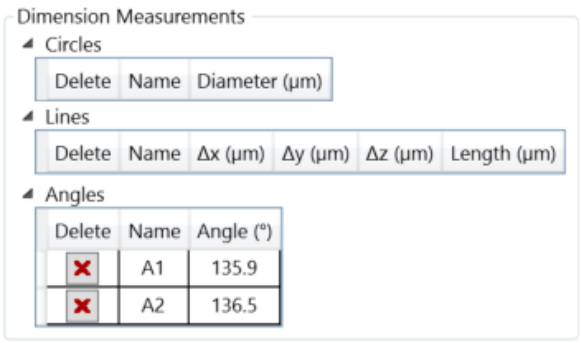
To move the line segment, click on any points indicated by the matching number above the dots and drag it to the location of interest. The center point will act as the vertex of the angle. The currently active line segment is indicated by the points being highlighted in green. Unselected line segments will be shown as purple.

With line segments placed, results of the angular measurements will be visible in the *Dimension Measurements* in the side panel.

Hide or show the results by clicking the directional arrows next to *Circles, Lines,* or *Angles* in the *Dimension Measurements* window. To remove an angle select it and click **Delete Active Line**, or hit the red X next to the segment name under the *Dimension Measurements*.



Example 1: Example angle in Top View



*Example 2: The **Dimension Measurements** table*

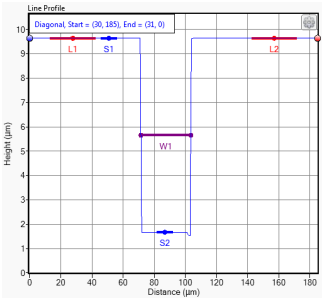
5.2.4 Step Height

Line Measurement

Measuring step height along a line in the sample is a three-step process:

1. Identifying the location on the sample to measure.
2. Selecting the leveling points.
3. Identify the step location.

To enable step height measurements, click the **Step Height** button under the Analysis tab of the Analyze ribbon. A side panel opens showing three tabs: *Line*, *Rectangle*, and *Histogram*. Select the *Line* tab.



Example linear plot with level points, step height locations, and a trench feature.

The currently selected measurement location is indicated by a red line across the sample. By default, this line starts at the center on the *Top View* window and measures horizontally across the sample. One end of the line is highlighted by a green point, the other is red. To move the measurement line, click on different locations in the top-view, or click and drag the red bar.

Click on either of the end points to switch to diagonal mode, explained in the next paragraph. When moving the measurement location by clicking on the *Top View* image, the linear plot for that location will be shown in the graph window.

Change the direction to vertical using the **Line Profile Orientation** dialog, or by pressing Ctrl+V on the keyboard. In diagonal mode, click and drag on either the **Start** or **End** points to set a self-defined path where step-height measurements are aligned to match features that don't align in the horizontal or vertical direction. The graph updates to scale based on the length of the line. Click **Go To Step 2** when the line is placed in the desired location.

If the data has already been leveled using the Level or Flatten functions in the Add Operator to Recipe tab, click **Skip Leveling & Go to Step 3**. If not, use the following instructions to level the surface.

Level points are set by placing them on the *Line Profile*, displayed in the lower left corner of the screen. There are two points, L1 and L2, indicated by vertical red lines surrounded by pink rectangles. The red line indicates the location of the leveling point. The rectangles show the area around the level point that is used to determine average height.

To determine rectangle width, click on the **Options** button under step 2 in the side panel. A wider range is generally more useful on rougher surfaces. Move the level points by clicking and dragging the red line. Place the level points so that they are both on the same plane of the surface with as much space as possible between them. Once suitable locations are selected, click **Apply Leveling & Go to Step 3** to continue.

Finally, set the step-height locations. This function works similarly to the level function, but instead of placing both points on the same plane of the sample, one point is placed at the bottom of the step and one at the top. Click the **Options** button to determine the width of the step-height points. Widths can be set independently, for example on a surface with a combination of smooth and rough surfaces.

Measure Feature Width searches for certain types of features (ridges and trenches) to determine their width. Use average feature width if there is more than one feature in the area of interest. Change the threshold percentage to change where the width analysis is performed (0 being the bottom, 100 being the top).

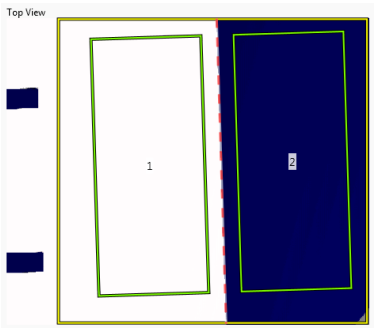
Materials enables the dissimilar materials function, used to account for errors in step-height measurement due to phase shifts in the reflection from different materials. For example, you may have a glass substrate coated with a metal film, aluminum for example. Due to the phase shift caused by the differing reflectance properties of the surfaces, the software may incorrectly locate the top of the step relative to the substrate. Enabling *Dissimilar Materials* allows the software to correct this problem in order to calculate an accurate step-height.

When using this feature, make sure to match the step-height point with the material. For example, in the hypothetical above S1 should be aluminum for the top of the step, and S2 should be dielectric for glass).

Once both points are in the desired locations, click **Finish** to show the results for step-height and trench width, if enabled.

Rectangle Measurement

The rectangle tool is used to measure the average height of the step across a user-defined region. Click the **Step Height** button to open the side panel, then click the *Rectangles* tab.



Select the *Automatic* radio button to have the software automatically locate the step edge (indicated by a dashed white and red line drawn through the 2D scan) or the *Manual* radio button to manually size and position a line segment so that it overlays the step edge to be measured.

For *Automatic* location, if the line is not correctly detected click and drag the outlines of the yellow box to change the analyzed area.

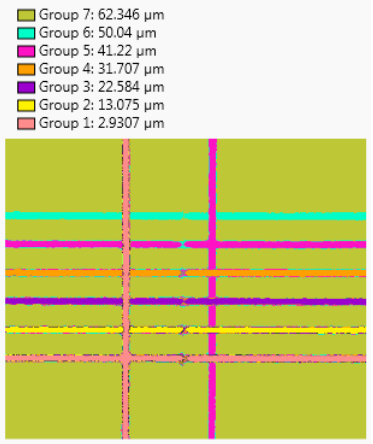
Click the **Next** button, and the software populates a pair of boxes labeled 1 and 2 on opposite sides of the step edge. Move and resize the boxes as desired. The boxes are mirrored – a change made to one is immediately reflected in the other.

The image can also be leveled using the rectangles area. Click the **Options** button to open the *Rectangle Step-Height Options* dialog box. Click the check box to enable surface leveling using rectangle one as the reference surface. Dissimilar materials is also available in this dialog.

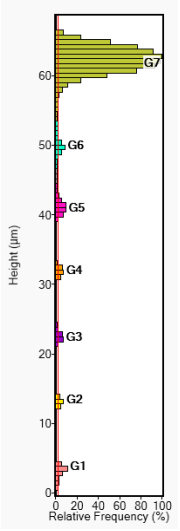
Click **Finish** to show the average measured step height under *Step-Height Result*.

Histogram Measurement

The histogram step height mode can be used to measure a variety of different samples, such as those with multiple step heights, or samples that may not have a well-defined step edge. While it is a very versatile tool, it is also the most complex of the three step height methods.



To enable the histogram mode, click on the **Step Height** button underneath the **Analysis** section of the **Analyze** ribbon to open the step height side panel. Click on the *Histogram* to get started.



Once selected, the software will present the histogram for the currently loaded scan. This graph shows the distribution of points in the data by height. The example to the left is a user calibrating their dicing sawThe majority of the points fall near the mid 60 micron range (the top of the sample) with small groupings roughly every 10 microns as they cut deeper and deeper into the sample. The parsing is based upon the settings below.

The first of these settings is the *Minimum relative frequency* which is indicated by the red vertical bar in the data. In order for a height value to be identified as a histogram region, it must reach this minimum value. Set this value by typing a number into the text box, or by dragging the red bar.

Next is the *Number of bins* which defines how many histogram groupings are on the graph. A higher number can be used to more finely key in on sections with more rapidly changing heights, while a lower number can be used to hide noise from rougher surfaces. Any value over 160 will revert to continuous mode, which is the maximum resolution option.

Minimum Height Variation refers to the width of the histogram groupings. The higher this number is, the wider a histogram region has to be to become a group. Using the example on the left, a variation of 1% gives the results shown in the image. Increasing that value to 5% would exclude every group except for G1, G5, and G7, and 10% would only leave G7.

While separating and defining regions, they will each be labeled in different colors on both the histogram and the 2D plot inside the histogram. This makes it easier to see changes as they are applied.

With the regions defined, there will be a variety of different data points available. The first will be a listing of the average heights of each level relative to the currently selected Origin point. Below the 2D plot will be a pair of tables.

The first table lists the average heights for each grouping, relative to all other available groupings. The second table will list some basic information about each layer: average, min and max height of the surface, azimuth, and inclination of the surface plane. Data from these tables can be copied out to other software programs by right clicking on the table.

5.2.5 Area Roughness

To enable, click on the **2D Area Roughness** button in the **Analysis** section. This opens the *Area Roughness* side panel, which contains the controls and results for this function.

First, set the *Area of Interest* for 3D parameter calculation. There are two options: *Entire image* or *Restricted area*. When *Restricted area* is selected, a yellow box on the *Top View* indicates the area currently being analyzed. This box can be moved by clicking inside the box and dragging it, or resized by clicking and dragging on one of the edges or corners. The exact length and width of the *Restricted area* may also be defined by typing values in the width and height boxes.

Once an area has been defined, the software will display the values for the parameters listed below. For details on each parameter, including the equations used to determine the values, see the Area Roughness Parameters section.

For best results when measuring roughness, level the surface using the Level or Flatten tool, apply a Filter, then apply the surface analysis.

Select which parameters will be shown by clicking the **Settings** button in the side panel. In the **Area Roughness Settings** dialog box click the checkbox to enable or disable the desired parameters. To disable or enable an entire section of parameters, click the checkbox next to *All Parameters* for that section.

Some parameters have additional configuration options, indicated by a gear button to the right of the parameter description in the *Area Roughness Settings* dialog box. Click the gear button to open a dialog box for additional parameter configuration. For more details, see the Area Roughness Parameters section.

The measured data can be copied from the software by selecting specific rows or the entire table. To select multiple rows, do one of the following: click and drag over the desired range, hold down the Ctrl key to select different individual rows, or hold down the Shift key to select all rows between the first row and last row selected. Right-click anywhere in the Area Roughness table to open a drop-down menu and select the desired option.

General Values			
Average	0.2658	µm	Average height in the designated area
Maximum	0.8190	µm	Highest height in the designated area
Minimum	-0.2680	µm	Lowest depth in the designated area.
Range	6.2349	µm	Total range between the highest and lowest point in the designated area.
Sdar	1.294	mm²	Developed surface area
Spar	0.9472	mm²	Projected surface area

Parameter			ISO 25178 Height	EUR 1578N Amplitude	ASME B46.1 3D
Sp	0.5532	µm	Maximum peak height	Maximum peak height	Peak height
Sv	0.5338	µm	Maximum valley depth	Maximum pit height	Valley depth
St	38.3	µm	Maximum Height (*Sz)	Maximum Height	Maximum peak to valley height
Sa	0.1340	µm	Arithmetic mean height	Arithmetic mean height	Arithmetic mean height
Sq	0.1414	µm	Root mean square height	Root mean square height	Root mean square height
Ssk	0.2394		Skewness	Skewness	Skewness
Sku	1.3834		Kurtosis	Kurtosis	Kurtosis

Parameter			EUR 1578N Hybrid	ISO 25178 Hybrid
Sdq	1.088		Root mean square gradient	Root mean square gradient
Sdr	36.63	%	Developed interfacial area ratio %	Developed interfacial area ratio %

Parameter			ISO 25178 Spatial
Sal	2.22	µm	Autocorrelation length

Parameter			ISO 25178 Functions
Smc	18.82	µm	Areal material ratio
Smr	28.93	%	Inverse areal material ratio
			Core height
			Material Ratio Peak
			Material Ratio Date
			Reduced Peak Height
			Reduced Dale Height

5.2.6 Line Roughness

Similar to the 2D parameters, this feature must first be enabled by clicking on the **Line Roughness** button under the **Analysis** section of the **Analyze** ribbon. The *Line Roughness* side panel will then open.

The software will draw a line across the *Top View* image, indicating the location on the scan data is currently being calculated from. By default this line will be centered horizontally on the scan. To alternate between horizontal, vertical, or diagonal line segments, use the radio buttons under

Profile Orientation in the side panel, or switch between horizontal and vertical using the keyboard commands ctrl+H and ctrl+V respectively.

Clicking and dragging on the center of the line will move the line in the x or y direction as needed, while clicking and dragging on the end points will switch into diagonal mode.

The **Settings** button works similar to the one in the Area Roughness tool, with check boxes being used to select which parameters to display. In addition, there are also the *Profile Parameter Settings* which will affect how the resulting values are calculated.

The first selection is the *Filter* method to be used; Gaussian, Spline (β = 0.625242), and Spline (β = 0). For a more in-depth explanation of the difference between these filtering methods, see the Filter article in the **Software Overview** section.

Next, set a cutoff wavelength to use for the filter. The values available will be calculated based on the loaded scan. The correct value for the application will be dependent on its individual needs, however the table excerpted from ASME B46.1 2D below can give a rough guideline depending on expected roughness of the surface.

Roughness				Cutoff
Rz (µm)	Ra (µm)	Rz (µin)	Ra (µin)	λ_c (µm)
< 0.025 - 0.1	> 0.006 - 0.02	>1 - 3.9	>0.24 - 0.79	80
0.1 - 0.5	0.02 - .1	3.9 - 19.7	0.79 - 3.9	250
.5 - 10	.1 - 2	19.7 - 393.7	3.9 - 78.7	800
10 - 50	2 - 10	393.7 - 1969	78.7 - 393.7	2500
50-200	10-80	1968.5 - 7874	393.7 - 3149.6	8000

Last, set the *Number of Samples* to use. By default, this value will be set to *Max Samples,* but can be changed to a user defined value. The number used refers to the number of segments the selected line will be broken into, based on the cutoff length selected. If the *User Specified* option is used, it will use the lesser of the entered number or the maximum available number of samples.

Data can be copied out of the software either by selecting individual rows, groups of rows, or the whole table at once by right clicking on the table. The table below provides a quick reference for each parameter.

Parameter	ASME B46.1 2D
Rp	: Maximum peak height+‡
Rv	: Maximum valley depth+‡
Rz	: Maximum height*†
Rt	: Maximum peak to valley height+‡
Rpm	: Maximum peak height*†
Rvm	: Maximum valley depth*†
Rtm	: Maximum height*†
Rmax	: Maximum sample peak to valley height†
Ra	: Arithmetic mean deviation+‡
Rq	: Root mean square deviation+‡
Rsk	: Skewness+‡
Rku	: Kurtosis+‡

Parameter	ISO 4287 Amplitude
Rp	: Maximum peak height*†
Rv	: Maximum valley depth*†
Rz	: Maximum height*†
Rt	: Total height+‡
Ra	: Arithmetic mean deviation*†
Rq	: Root mean square deviation*†
Rsk	: Skewness*†
Rku	: Kurtosis*†
Rp1max	: Maximum sample peak height†
Rv1max	: Maximum sample valley depth†
Rt1max	: Maximum sample peak to valley height†
Rc	: Average peak to valley height†

Parameter	ISO 4287 Curves
Rmr	: Profile material ratio ‡*

* Averaged over sample lengths.

+ Calculated over evaluation length.

‡ Uses currently selected filter and cutoff wavelength

† Uses currently selected filter, cutoff wavelength, and number of samples.

5.2.7 Bearing Ratio

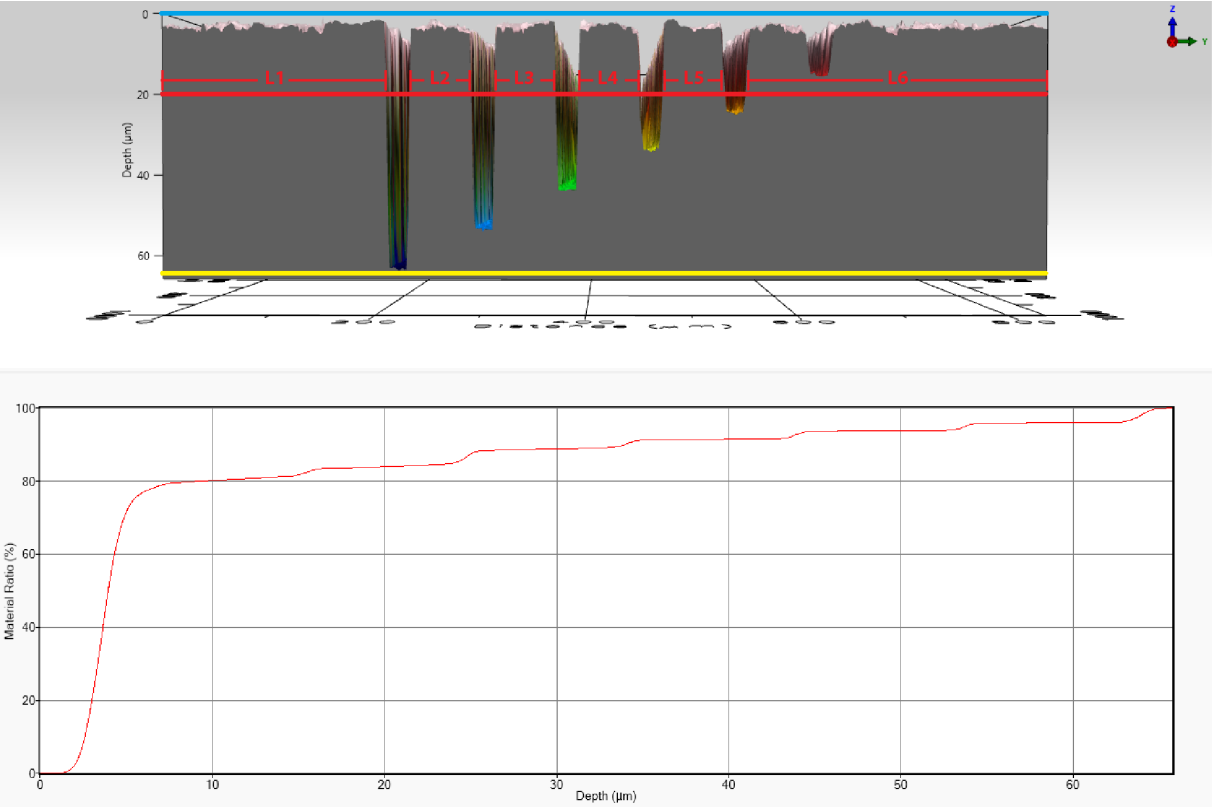
The Bearing Ratio Curve is the integral of an Amplitude Distribution Function (ADF) used to quantify the ratio of material to air in a scanned surface. This property is known by many names including:

Abbot-Firestone curve

Abbot-curve
Bearing area curve
Material ratio curve

All these names refer to the same value.

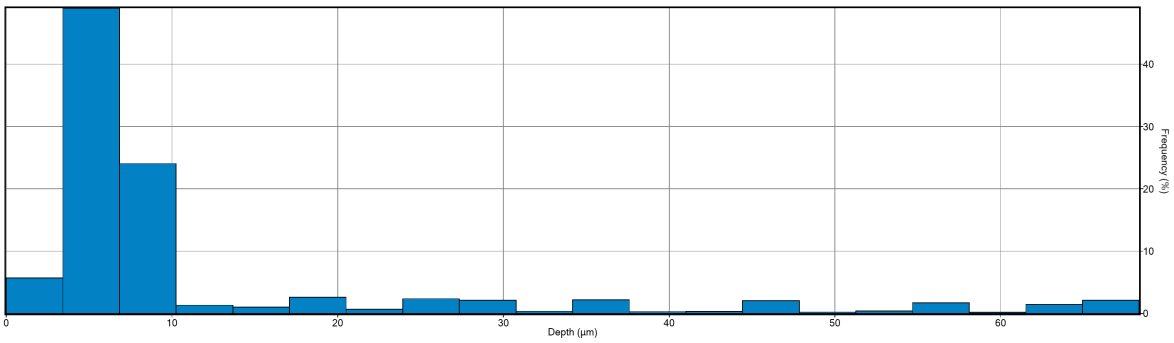
Imagine it as taking a horizontal slice through the scan at a given depth. What would the ratio of material to air be? For example, in the image below the blue line indicates a depth plane of 0 µm. A plane drawn at that height would be 100 percent air, as there is no material at that height.



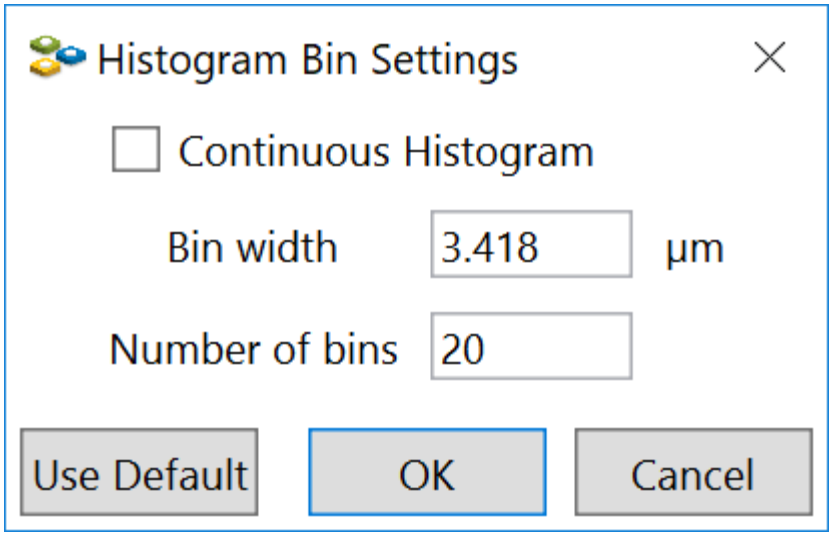
Example Bearing Ratio Curve: Dicing Saw Qualification

However, a plane drawn at a 20 µm depth, illustrated by the red line, has a mix of material and air present. To calculate that ratio, take the sum of the lengths of material (L1, L2, L3...) and divide it by the total length of the scan. At this depth, it works out to a value of roughly 84% surface to air. The further down in the sample, the more the ratio tips towards 100%, which in this case occurs just past the bottom of the lowest dice channel at roughly 66 µm (yellow line).

In addition to showing the bearing ratio curve, this feature also can show a histogram function. An example histogram from the same scan above is shown below.



The histogram shows, as a function of the currently selected z-scale, how many points fall in a given thickness range. The width of these ranges, or bins, can be controlled by clicking on the drop-down on the **Bearing Ratio** button and selecting **Settings**.

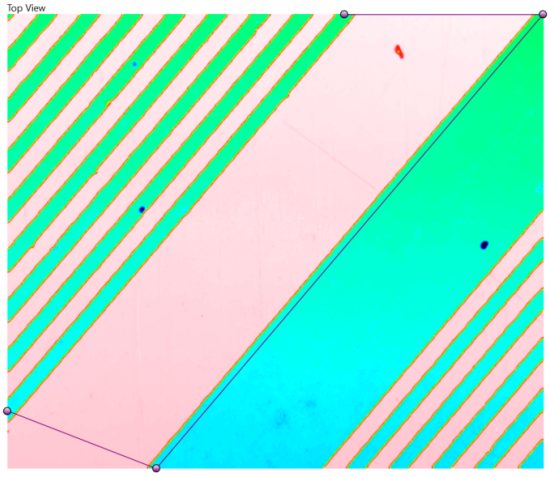


Values can be entered by a set bin width or by the numbers of bins desired. A change in either value will update the other to match (e.g., for a total range of 10 um, changing the number of bins from 2 to 5 will change the bin width 5 µm to 2 µm). Selecting **Use Default** will set the number of bins at 20. Selecting to use a continuous histogram will fix the number of bins at 512.

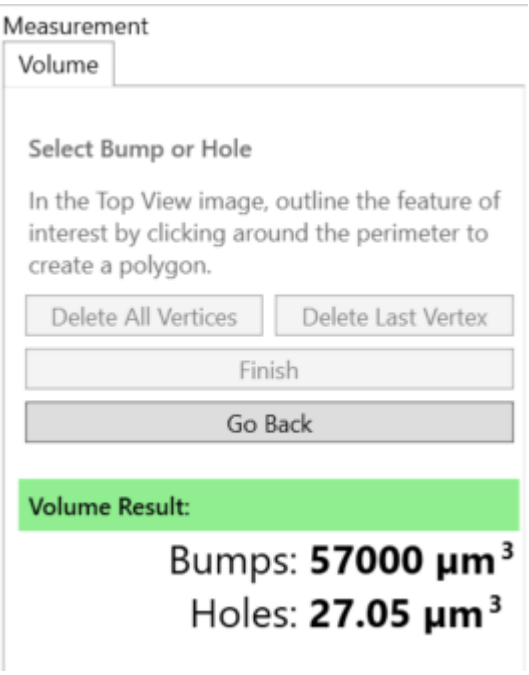
The bins can be oriented horizontally (height values on the y-axis, frequency on the x-axis), or vertically (as shown above with height on the x-axis, frequency on the y-axis). To switch between the two, right click on the graph window and select from the resulting drop-down menu

5.2.8 Volume

In Profilm, Volume is the measurement of material above the best fit surface plane (bumps), and below the best fit surface plane (holes), within a defined polygon. Click on the *Top View* image and add a minimum of 3 vertices to create a new polygon. Click **Delete All Vertices** to remove all vertices, or **Delete Last Vertex** to remove the most recently added vertex. Click **Finish** to draw the surface plane. The total volume of all bumps and holes relative to the surface plane are displayed.



Vertices placed on the Top View image

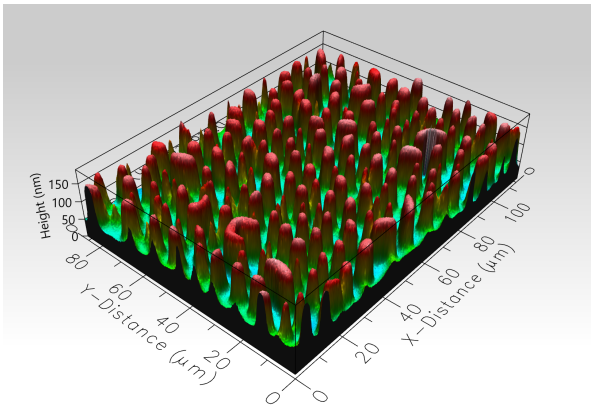


Resulting volume measurement

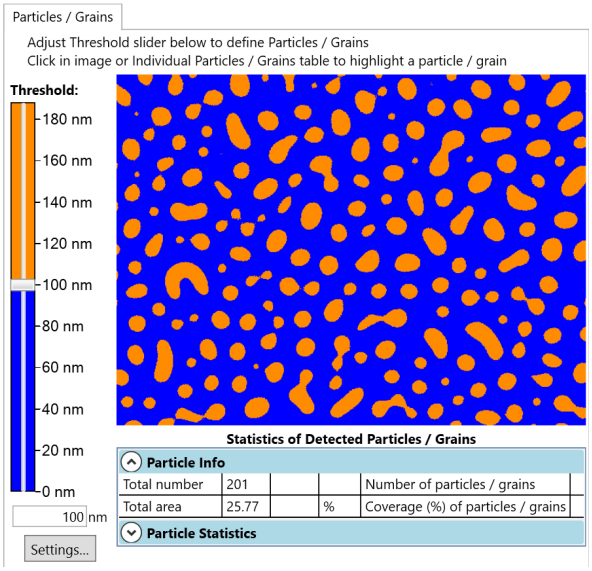
Click and drag any vertex to change the shape of the polygon, or click **Go Back** to add or delete additional vertices.

5.2.9 Particles/Grains

Identifies and characterizes features above or below a set height plane. Select **Particles/Grains** from the *Analysis* ribbon to open the side panel with controls.

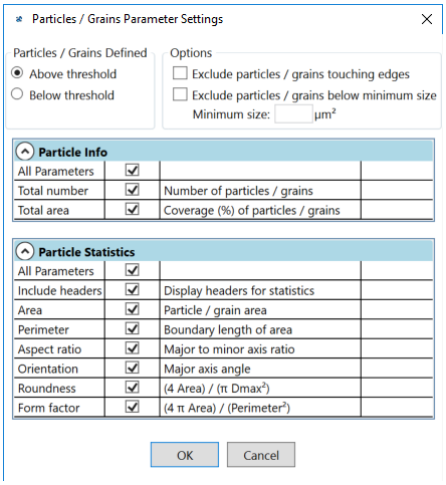


Example image of an immiscible polymer blend



Example Particles/Grains plot for the image at left, 100nm threshold.

The side panel displays a two dimensional version of the image, colored in orange and blue. The region color is determined by whether the region is found above (orange) or below (blue) the height plane set by the Threshold value. Adjust this value by clicking and dragging the slider bar, or by entering a number in the text box. The image updates as changes are made.



The Settings button opens the *Particles / Grains Parameter Settings* dialog box.

Select to analyze features above or below the threshold value, and whether particles/grains touching the edge of the image should be included for analysis. Set a minimum particle size to exclude smaller features or outlier points. Select which parameters, if any, are displayed in the *Particle Info* and *Particle Statistics* tables.

Below the 2D image in the dialog box are three tables: *Particle Info*, *Particle Statistics*, and *Individual Particles/Grains*. Info and statistics provide collective values for every feature analyzed based upon the settings selected, while individual values refer to specific features.

For example, the table below shows the values for the 3D image shown earlier. *Particle Info* shows values for the plane drawn at 100 nm (the threshold value). There are 201 particles accounting for 25.77% of the total area of the surface. *Particle Statistics* breaks this down even further, calculating values for *Area*, *Perimeter*, *Aspect Ratio*, *Orientation*, *Roundness*, and *Form Factor* of those particles. Definitions of these values are also included.

Particle Info

Total number	201			Number of particles / grains	
Total area	25.77		%	Coverage (%) of particles / grains	

Particle Statistics

Parameter	Mean	Stdev	Units	Description	
Area	12.45	9.663	μm^2	Particle / grain area	
Perimeter	13.04	7.06	µm	Boundary length of area	
Aspect ratio	1.539	0.7769		Major to minor axis ratio	
Orientation	11.67	47.64	°	Major axis angle	
Roundness	0.6467	0.1694		(4 Area) / (π $Dmax^2$)	
Form factor	0.9305	0.5629		(4 π Area) / ($Perimeter^2$)	

Particle Info and Statistics for the immiscible polymer blend.

Individual Particles/Grains refers to the values for each unique feature in the image. Click on a particle in the 2D image to highlight it in the image (shown in teal) and in the table (shown in yellow), or vice versa. Select multiple features by holding down the Ctrl key while clicking each feature. Right-click on a table to copy data to the clipboard. A dropdown prompts to copy the selected data, or the entire table.

Individual Particles / Grains

ID	Area (μm^2)	Perimeter (µm)	Aspect ratio	Orientation (°)	Roundness	Form fa
1	7.614	11.32	1.351	-7.59	0.6432	0.7467
2	14.82	15.81	1.434	82.08	0.6454	0.7455
3	1.336	4.6	2.102	2.308	0.438	0.795
4	22.28	17.9	1.173	-72.98	0.72	0.8741
5	2.362	5.89	1.087	84.09	0.7115	0.8569

First five results of Individual Particles / Grains from the immiscible polymer blend.

Part VI

6 Analysis Parameter Definitions

6.1 Area Roughness Parameters

Profilm is capable of measuring multiple area roughness parameters, each of which is described below in greater detail.

6.1.1 Average

The **Average** is defined as the average absolute height of the sample surface in the area being analyzed.

6.1.2 Maximum

The **Maximum** is defined as the highest absolute height in the selected area. This varies from Sp (Peak Height), which measures the highest point from mean plane of the sampling area.

6.1.3 Minimum

The **Minimum** is defined as the lowest absolute depth in the selected area. This varies from Sv (Valley Depth), which measures the lowest depth from the mean plane of the sampling area.

6.1.4 Range

The **Range** is defined as the difference between the **Maximum** and **Minimum** values.

6.1.5 Sdar – Developed Surface Area

The **Developed Area, Sdar**, is defined as the total surface area, taking in account the entire topography of the sampling area. The ratio of **Sdar** to **Spar** is linked to the volume of the surface peaks and valleys.

6.1.6 Spar – Projected Surface Area

The **Projected Area, Spar**, is defined as the surface area of a flat x,y plane of the sampling area. The ratio of **Sdar** to **Spar** is linked to the volume of the surface peaks and valleys.

6.1.7 Sp - Peak Height

The **Peak Height**, **Sp**, is defined as the largest peak height value from the mean plane within the sampling area.

$$S_p = MAX(z(x_i, y_j))$$

6.1.8 Sv - Valley Depth

The **Valley Depth**, **Sv**, is defined as the largest valley depth value from the mean plane within the sampling area.

$$S_v = MIN(z(x_i, y_j))$$

6.1.9 St – Maximum Peak to Valley Height

The **Maximum Peak to Valley Height, St**, is defined as the height difference between the highest peak and the deepest valley.

St = Sp + Sv

6.1.10 Sz - Ten-Point Height

Mean of distance between the 5 highest peaks and the 5 deepest valleys. A region of 3x3 samples is taken into account to determine the peaks and the valleys. The digital equation that represents this algorithm is displayed below.

$$S_z = \frac{\sum_{i=1}^{5} | z_{pi} | + \sum_{i=1}^{5} | z_{vi} |}{5}$$

where Z_{pi} and Z_{vi} (i = 1,2,3,4,5) are the five highest peaks and five lowest valleys respectively.

6.1.11 Sa - Average Roughness

The average roughness parameter, **Sa**, is the most used surface roughness parameter. It is the arithmetic mean or average of the absolute distances of the surface points from the mean plane. The digital equation that represents this algorithm is displayed below.

$$S_a = \frac{1}{MN} \sum_{j=1}^{N} \sum_{i=1}^{M} | z | (x_i, y_j)$$

where M is the number of columns in the surface and N is the number of rows in the surface.

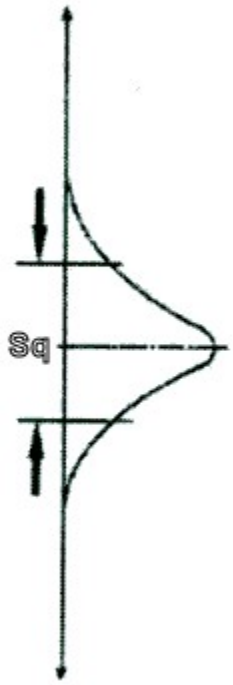
6.1.12 Sq - RMS Roughness

The **Root Mean Square (RMS) Roughness parameter**, **Sq**, is the root mean square of the surface departures from the mean plane within the sampling area. The digital equation that represents this algorithm is displayed below.

$$S_q = \sqrt{\frac{1}{MN}\sum_{j=1}^{N}\sum_{i=1}^{M} z^2(x_i, y_j)}$$

where M is the number of columns in the surface and N is the number of rows in the surface.

Sq is very general and is an often used parameter. In the field of statistics, it is the sample standard deviation.

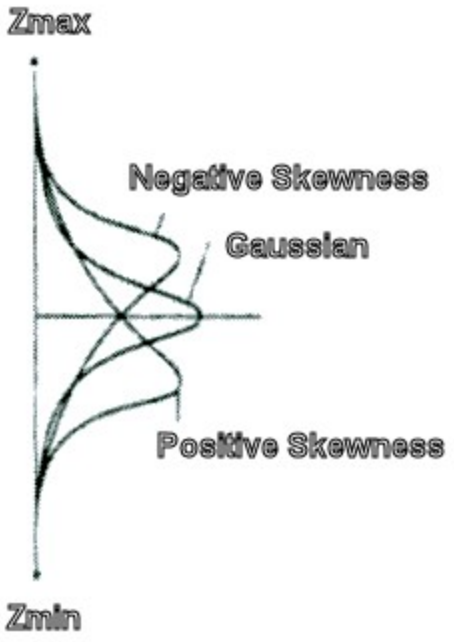


6.1.13 Ssk - Skewness

Skewness measures the symmetry of the variation of a surface about its mean plane. A Gaussian surface, having a symmetrical shape for the height distribution, has a skewness of zero. A plateau honed surface with predominant plateau and deep valleys will tend to have a negative skew, whereas a surface comprised of disproportionate number of peaks will have positive skew. The digital representation of this parameter is displayed below.

$$S_{sk} = \frac{1}{MNS_q^3} \sum_{j=1}^{N} \sum_{i=1}^{M} z^3(x_i, y_j)$$

where M is the number of columns in the surface and N is the number of rows in the surface.

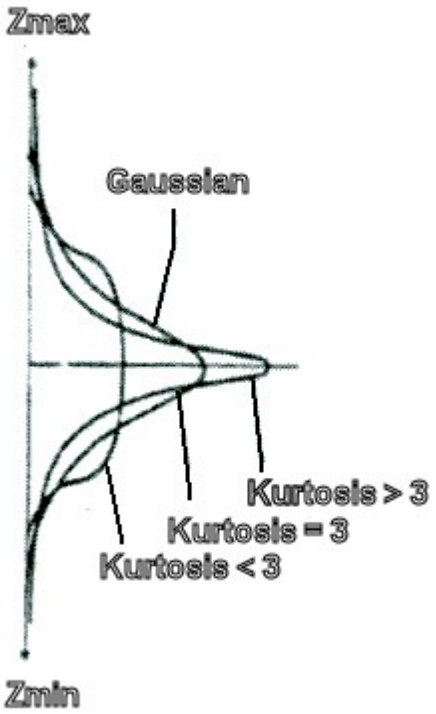


6.1.14 Sku - Kurtosis

Kurtosis is a measure of the peakedness or sharpness of the surface. The digital representation of this parameter is displayed below.

$$S_{ku} = \frac{1}{MNS_q^4} \sum_{j=1}^{N} \sum_{i=1}^{M} z^4(x_i, y_j)$$

A Gaussian surface has kurtosis value of 3. A surface that is centrally distributed has a kurtosis value greater than 3. A surface that has a well spread out distribution has a kurtosis value of less than 3. By using a combination of the skewness and kurtosis values, it is possible to identify plateau honed surfaces that have relatively flat top, but contains deep valleys.



6.1.15 Sal - Autocorrelation Length

Autocorrelation Length is a measure that helps to describe the autocorrelation character of a surface. It is described as the shortest horizontal distance in which the autocorrelation function decays to 0.2 in any direction.

$$S_{al} = \min\left(\sqrt{\tau_x^2 + \tau_y^2}\right),$$

$$R = \{(\tau_x, \tau_y) : ACF(\tau_x, \tau_y) \leq 0.2\}$$

For surfaces with predominate lay, Sal is found in a direction perpendicular to the lay of the surface. If a surface is dominated by low frequency components (long wavelength components), Sal will be large. If a surface is dominated by high frequency components (short wavelength components), Sal will be small.

6.1.16 Sdq – Root Mean Square Gradient

The **Root Mean Square Gradient, Sdq**, is defined as the root mean square slope of the surface departures from the mean plane within the sampling area.

6.1.17 Sdr% – Developed Interfacial Area Ratio

The **Developed Interfacial Area Ratio**, **Sdr**, is defined as the percentage of additional surface area contributed by topography within the sampling area relative to the surface area of the sampling area as a flat x, y plane.

6.1.18 Smr(c) - Areal Material Ratio

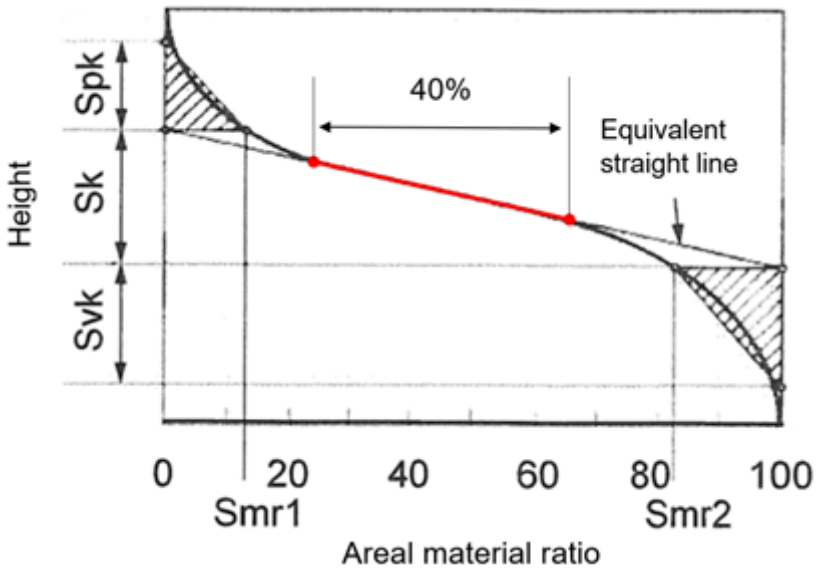
Area of the material at or above the specified height c expressed as a percentage of the total evaluation area. The height c is defined as the distance above or below the reference plane, where the reference plane can be configured as the maximum height, mean height, minimum height, or the height where the material ratio is equal to a specified percentage by clicking the cog icon next to Smr in the Area Roughness Settings dialog.

6.1.19 Smc(mr) – Inverse Areal Material Ratio

Inverse areal material ratio – The height c at which the Areal material ratio (mr) is equal to the specified percentage set by clicking the cog icon next to Smc in the Area Roughness Settings dialog. The returned height c is measured using the mean plane as the origin.

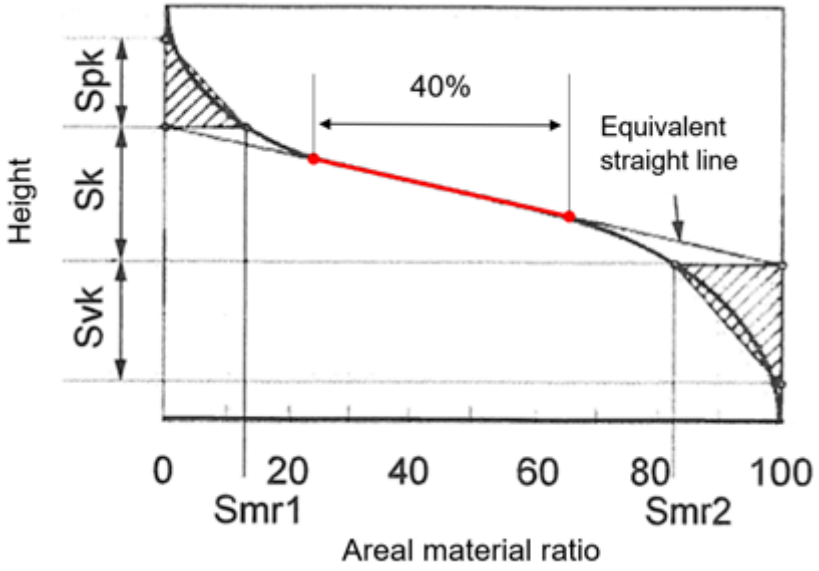
6.1.20 Sk - Core Height

The core height is defined by first calculating the equivalent straight line. The equivalent straight line is a 40% wide, best fit to the central region of the material ratio curve positioned to give the lowest slope. Once the equivalent straight line is determined, the intersection of this line with material ratio = 0% defines the top of the core surface region, and the intersection of this line with material ratio = 100% defines the bottom of the core surface region. Sk is the distance between the top and bottom core heights.



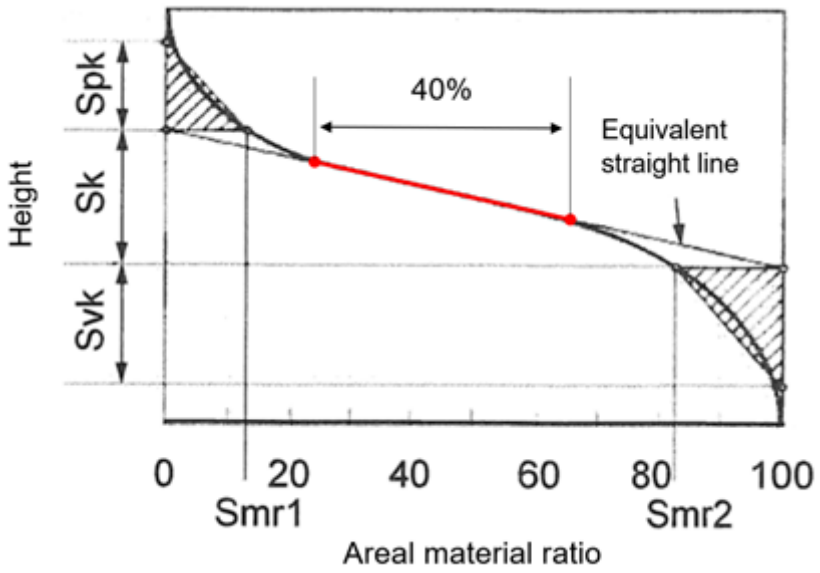
6.1.21 Smr1 - Material Ratio Peak

The ratio of the area of the material at the intersection line which separates the protruding hills from the core surface to the evaluation area.



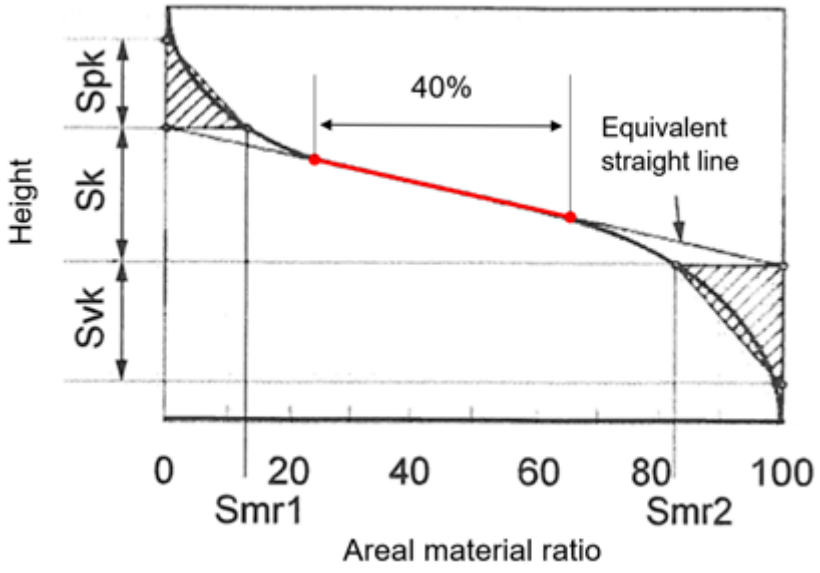
6.1.22 Smr2 - Material Ratio Dale

The ratio of the area of the material at the intersection line which separates the protruding dales from the core surface to the evaluation area.



6.1.23 Spk - Reduced Peak Height

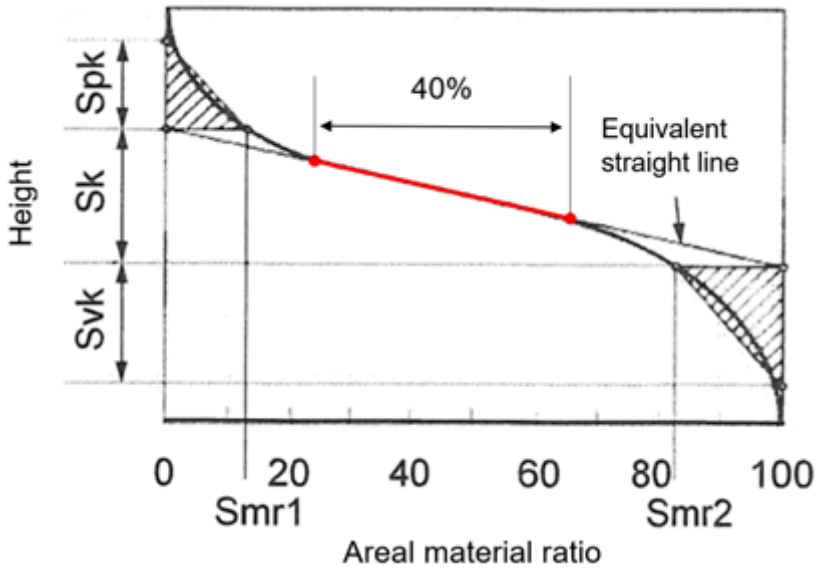
Average height of the protruding peaks above the core surface. First, the area of the peak region between the material ratio curve and the top of the core region is calculated. Spk is defined as the height of the triangle with base = Smr1 and the same area as the peak region (see left most cross hatched region in figure).



6.1.24 Svk - Reduced Dale Height

Average height of the protruding dales below the core surface. First, the area of the dale region between the bottom of the core region and the material ratio curve is calculated. Svk is defined as

the height of the triangle with base = 100 – Smr2 and the same area as the dale region (see right most cross hatched region in figure).



Part VII

7 Appendices

7.1 Software Automation

Measurements can be automated, and additional data analysis can be performed by user-supplied software. This is done using the FIRemote class, which is part of Profilm.exe. An example client program and source code are included as part of the Profilm software package.

To run the example client program, execute FIRemoteTestProfilm.exe, which is in %programfiles% \Filmetrics\Profilm\. Complete documentation of the FIRemote class, together with the source code for FIRemoteTestProfilm.exe is in %programfiles%\Filmetrics\Profilm\SourceCode\.

See “Profilm Software Automation.pdf” for detailed instructions on writing your own client software.

7.2 Supported File Types

Filmetrics File Types
.fibps
.fibcs

General use files
.sur
.txt

Alicona
.al3d

Asylum Reseach
.ibw

Bruker
.opdx

FRT MicroProf
.frt

Gwyddion
.gwy

ImageMetrology
.bcr
.bcrf

Keyence Profilometry
.vk4

Lext
.lext

MetroPro
.dat

NT MDT SPM
.mdt

OpenGps
.x3p

Park Systems
.tif*

Sdf
.sdf*

Seiko
.xq*

Sensofar
.plu
.plux

SpipAscii
.asc

Veeco Dimension and **Veeco Nanoscope**
.*

WITec
.dat
.wit

Wsf Ascii
.wsf

WycoVeeco
.opd
.ascii

Zeta3D
.zmg

7.3 WLI

White Light Interferometry (WLI) is a measurement technique that uses white light in conjunction with movement perpendicular to the sample to accurately determine sample position.

Light travels from the light source down through a special interferometric objective (Mirau or Michelson in the Profilm3D) which splits the light, sending half to the sample being measured, and the other half to a reference mirror. As this light reflects back from both the sample and the mirror, it will be recombined on its path back to the detector, though with a difference in phase having occurred. This phase shift will cause the light to interfere with itself, giving the characteristic light and dark fringes (interference pattern) in the image.

With careful design of the interferometer, the point of greatest contrast in these fringes will indicate a perfect focus condition. By scanning the objective up and down relative to the sample the interference patterns will move in concert with the focal plane, showing the change in distance from the lens as it moves. By coupling this motion to a high precision z movement system, an imaging sensor, and our advanced software package we are able to generate the 3-Dimensional scans for the Profilm3D.

7.4 PSI

Phase Shifting Interferometry (PSI) is similar to white light interferometry, though with several key differences that allows it to make higher precision measurements.

The first important difference between PSI and WLI measurements, is that the PSI measurement is taken over a much tighter wavelength range as opposed to the white light measurements of WLI. This is achieved in the Profilm3D with the use of an optical filter that can be placed in the light path. Doing this expands the envelope of the correlogram that the system detects from the interference patterns.

The next important difference is that PSI measurements require a much higher degree of flatness in the sample. This has to do with how the data is acquired. When scan is made, the software is imaging at some fixed interval as it moves through the scan, generating height data for each pixel in a series of frames. When performing a WLI analysis, we are able to select the best intensity values from each pixel at varying frames, so that one pixel may be using frames from early in the scan while another is using those from the middle. A PSI analysis will instead complete a scan, and then pick the same finite range of frames for each pixel. This limits uncertainties that may appear as a result of stringing together different frame groupings that may occur in WLI mode, and increases the precision of the measurement system.

The trade-off for this higher degree of precision is that the efficacy fails at larger thickness ranges, so the PSI works best for films with less substantial changes in surface height. It is also considerably more sensitive to sample tip/tilt, so extra care must be taken when aligning the sample on the stage.

7.5 Composite WLI/PSI

The Composite WLI/PSI scan is used to expand the functional range of the PSI scan from an upper limit of roughly three microns up to ten microns. This is accomplished by taking matching WLI and PSI scans from the same location using the same settings. The PSI scan is then unwrapped using the WLI scan height data as a guide.

7.6 SNR Enhance

The **SNR Enhance** function uses noise reduction techniques to remove background noise from the measured correlogram data prior to analysis. This can be used to improve results on less ideal surfaces, though at a trade-off of increased analysis time.

Interaction with the **SNR Enhance** function is controlled by the **Threshold** value. This value refers to the detected correlogram envelope amplitude as a percent of the total measurement range. If the detected amplitude in the correlogram is less than the selected value, the height data for that pixel will be marked as invalid.

7.7 System Specifications

7.7.1 Performance Specifications

	WLI	PSI
Thickness Range	50 nm - 10 mm	0 - 3 µm
RMS Repeatability[1]	1.0 nm	0.1 nm
Step Height Accuracy [2]	0.7%	
Step Height Repeatability [3]	0.1%	
Sample Reflectance Range	0.05% - 100%	
ISO 25178 Compliant	Yes	

[1] Typical value
[2] 8 µm step, 1 sigma
[3] 8 µm step, 100 successive measurements, 1 sigma

7.7.2 Objectives Specifications

Objectives [1] (Nikon CF IC Epi Plan)

Magnification	2.5X	5X	10X	20X	50X	100X
Field of View at 1X Zoom	8.0 x 6.8 mm	4.0 x 3.4 mm	2.0 x 1.7 mm	1.0 x 0.85 mm	0.4 x 0.34 mm	0.2 x 0.17 mm
Numeric Aperture	0.075	0.13	0.3	0.4	0.55	0.7
Working Distance	10.3 mm	9.3 mm	7.4 mm	4.7 mm	3.4 mm	2.2 mm
Spatial Sampling at 4X Zoom [2]	3.52 µm	1.76 µm	0.88 µm	0.44 µm	0.176 µm	0.088 µm
Resolving Power of Lens	3.7 µm	2.1 µm	0.92 µm	0.69 µm	0.5 µm	0.4 µm
Maximum Sample Slope [3]	3°	8.5°	14°	21°	25°	42°

[1] Sold separately
[2] Pixel size projected on sample
[3] Greater (better) for rough surfaces

7.7.3 Mechanical Specifications

Z Range	100 mm
Piezo Range	500 µm
Scan Speed, Vertical	12 µm
XY Stage Type	Automated
XY Stage Range	100 mm x 100 mm
Tip/Tilt Stage	± 5°, Manual
Camera	2592 x 1944 (5 megapixels)
Camera Zoom [1]	1X, 2X, 4X
System Size, W x D x H	300 mm x 300 mm x 550 mm
System Weight	15 kg

[1] Digitally realized. Number of effective pixels after binning is 648x484

7.7.4 Facilities Requirements

Electrical Requirements

Input:100-240 VAC, 50/60 Hz, 1.4A
Output: Systems shipped prior to 2019: 5V ⎓ , 6A Systems shipped 2019 and later: 24V ⎓ , 3.7A
System Fuse: 6A, 250 VAC, Fast Acting, Sz 3AG (Dims: 0.25"x1.25"; Cylinder)

Enviromental Operation Condition	Acceptable Range (Use)	Acceptable Range (Storage)
Temperature:	15°C to 25°C	-30°C to 60°C
Relative Humidity:	10% to 90%, Non-condensing	5% to 95%, Non-condensing
Barometric Pressure:	480 mmHg to 1000 mmHg	400 mmHg to 1100 mmHg
Altitude:	-500 m to 3000 m	-500 to 5000 m

7.8 Replacing the Fuse

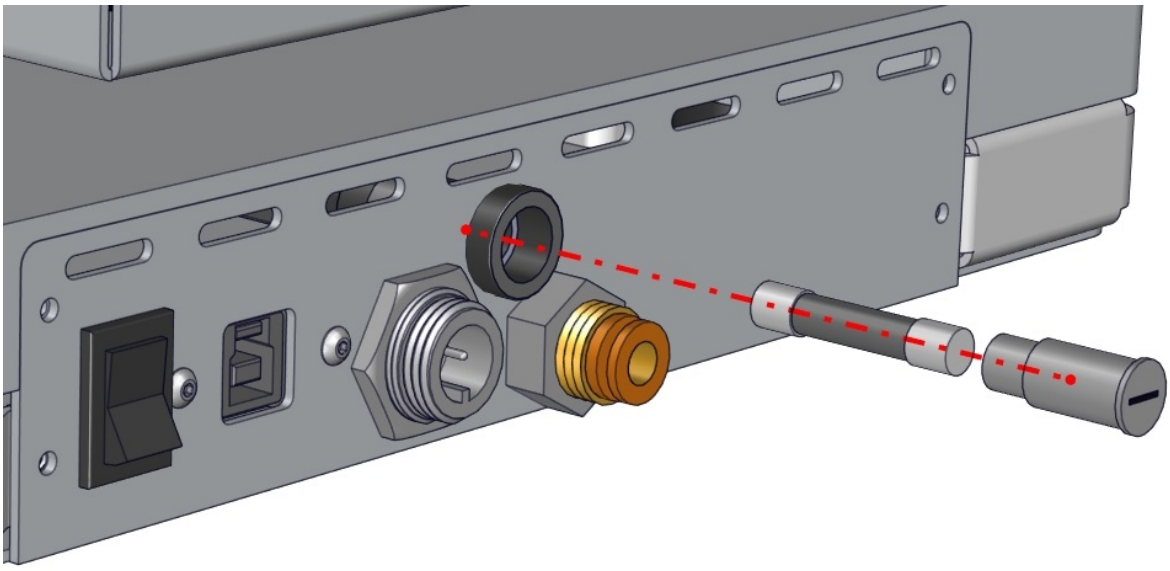
For systems manufactured prior to 2019, the following instructions can be used to replace a blown fuse.

1. Exit the Profilm software, and power down the instrument.

2. Disconnect the power from the Profilm3D.

3. Disconnect the USB 3.0 cable from the Profilm3D.

4. Use a flat head screwdriver to unlock the fuse holder by turning counter-clockwise. The fuse holder location is show in the drawing below.



5. Pull the fuse holder out of the Profilm3D.

6. Remove the fuse from the fuse holder.

7. Place the new fuse into the fuse holder. For specifications on the type of fuse to use, see the Facilities Requirements.

8. Replace the fuse holder in the back of the Profilm3D. You may have to rotate the fuse holder in order for it to insert properly.

9. Using the flat head screwdriver, push in on the fuse holder while turning it clockwise to tighten.

7.9 Changing Objectives

If more than one objective has been purchased, you can use the following instructions to change between lenses.

1. Remove any samples from the stage.
2. Set the height value to at least 50 mm (50000 microns).
3. Gently turn the current objective clockwise. Be careful not to twist too hard, as torqueing the piezo assembly can potentially cause damage to the instrument.
4. Return the objective to its holding case.
5. Remove the new objective to be attached from its case.
6. Align the threads on the objective with the threads of the mounting ring, and carefully begin to turn the objective counter-clockwise until it catches. Be careful not to cross-thread the objective.
7. Continue turning the objective until snug to the mounting ring, roughly finger tight.
8. In the Profilm software, select the new objective from the drop-down menu in the **Objectives** section of the Live window.
9. Check the operating voltages of the piezo assembly, which can be made visible in the **Live** window by clicking on the Help ribbon, and selecting Hardware. Voltages should be at 30V ± 3V, though the system will still operate effectively at other values. If voltages drop to 0, or are showing 80 or above, contact Filmetrics for further assistance.

If switching between two different Mirau objectives, the objective changing process is now complete. When switching from a Mirau to a Michelson objective or between the two Michelson options, continue on to the following steps:

1. Using the **Z Position** controls, move the stage head down until there is an image focus of the sample stage.
2. Click the **Set Zero** button.

Index

- A -

- S -