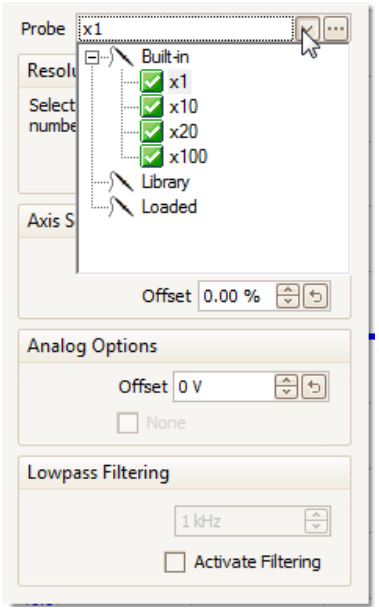
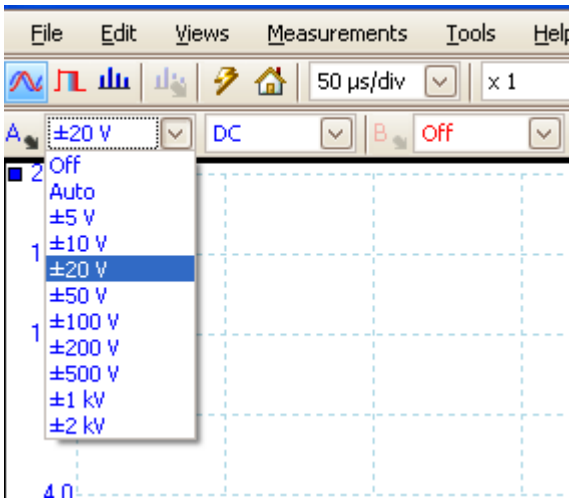
1 Custom probes

PicoScope 6 allows you to adjust the scaling of your input signal using custom probes. There are a number of predefined probes. By default PicoScope 6 has probes set up for x10, x20, and x100. The automotive software has an additional set of predefined probes (please refer to the **Automotive** section for more information).

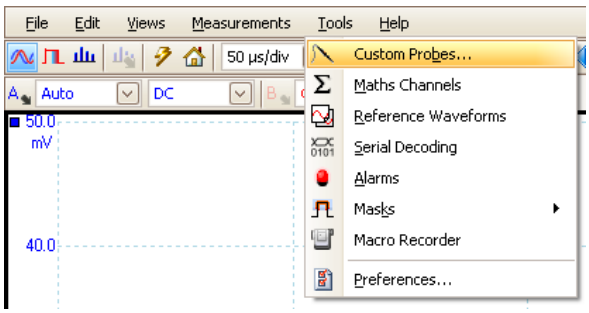
To access custom probes, click **Channel Options** and then click **Probe**:



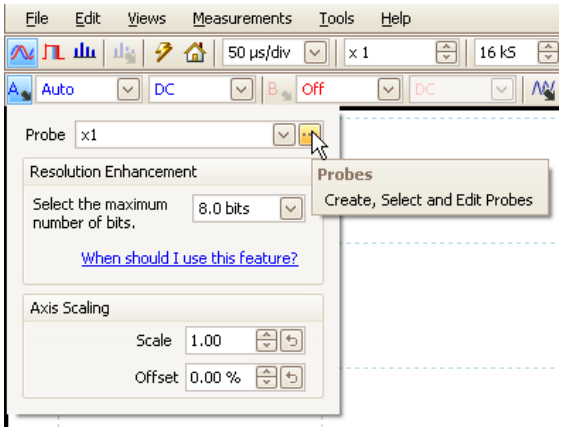
Here are the voltage ranges when a x100 probe has been selected:



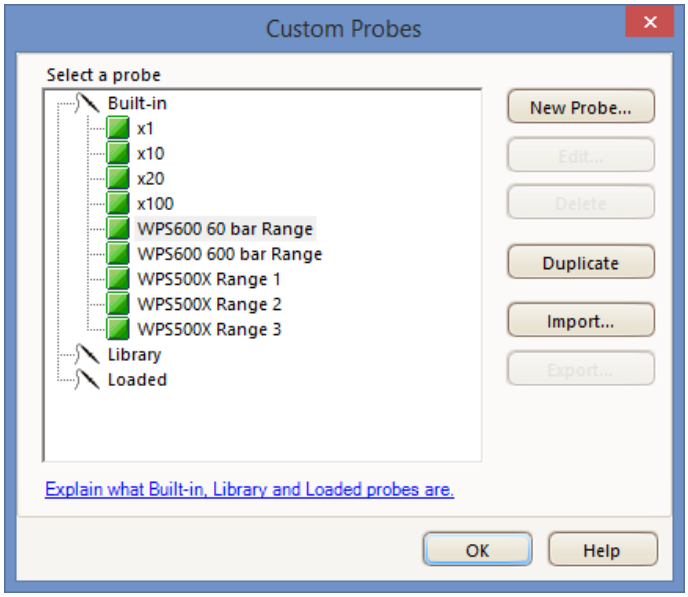
You also have the flexibility to create your own custom probes with their own units of measurement. You can achieve this using the **Custom Probe** wizard. This can be accessed by selecting **Tools > Custom Probes** from the menu:



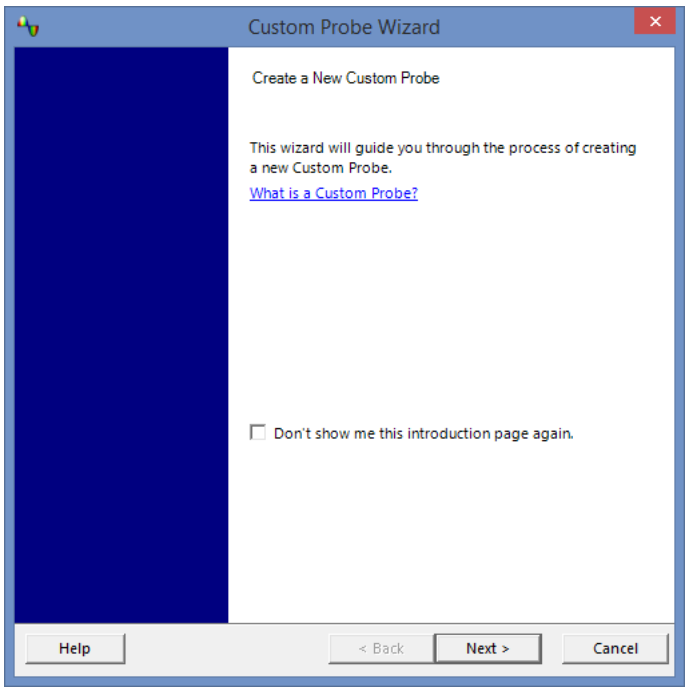
Alternatively, you can click **Channel Options** and then the ⋯ button:



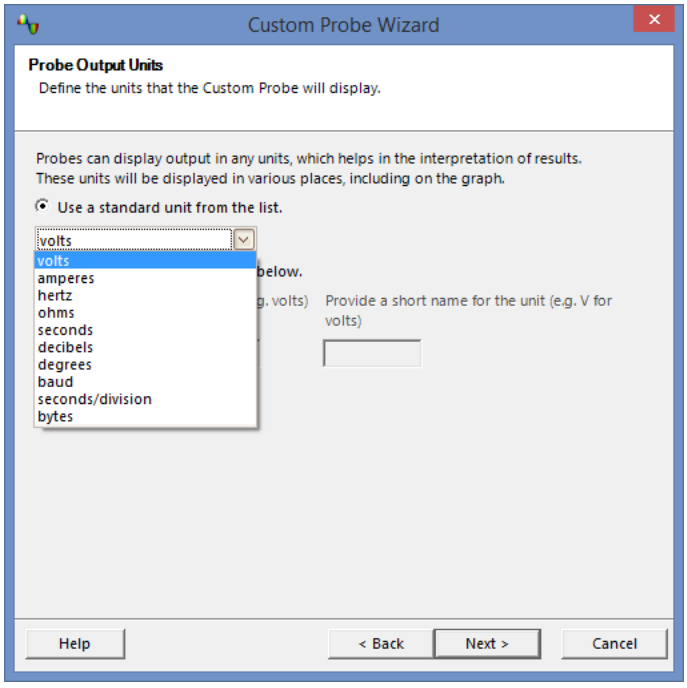
The **Custom Probes** window appears with a list of built-in probes. You can create new probes from scratch, copy and edit existing probes, or import and export probes:

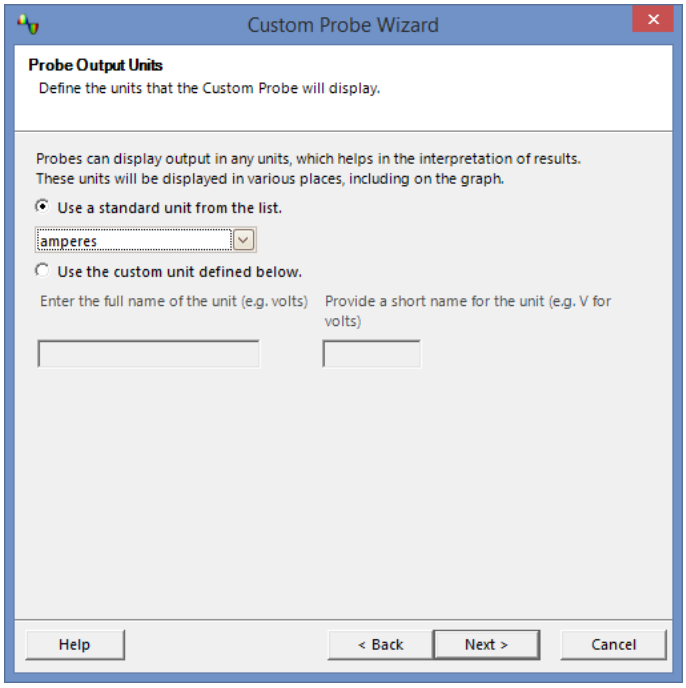


When you click **New Probe**, the **Custom Probe** wizard runs through the steps to create a custom probe. To give an example we will create a DC current clamp that takes 10 mV per amp and can measure a maximum current of 100 A.

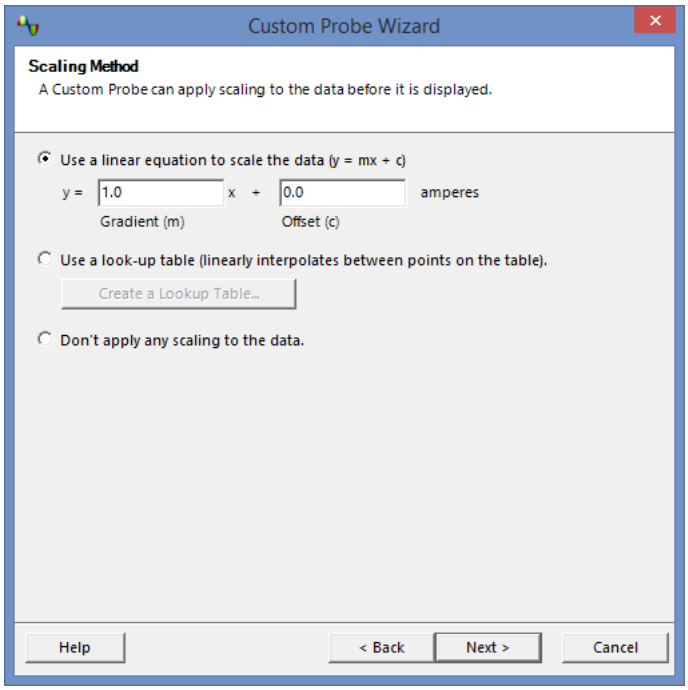


Click **Next** to move on to the next window, which allows you to either select standard units from the list or create your own. In this case, amperes can be chosen from the list as it is already defined:





The next step defines the scaling method. Either an equation, or a lookup table, or no scaling can be applied. No scaling is useful if just the units need changing or for limiting the view of a range e.g. showing only positive voltages. More information on this is available in the **Advanced** section.



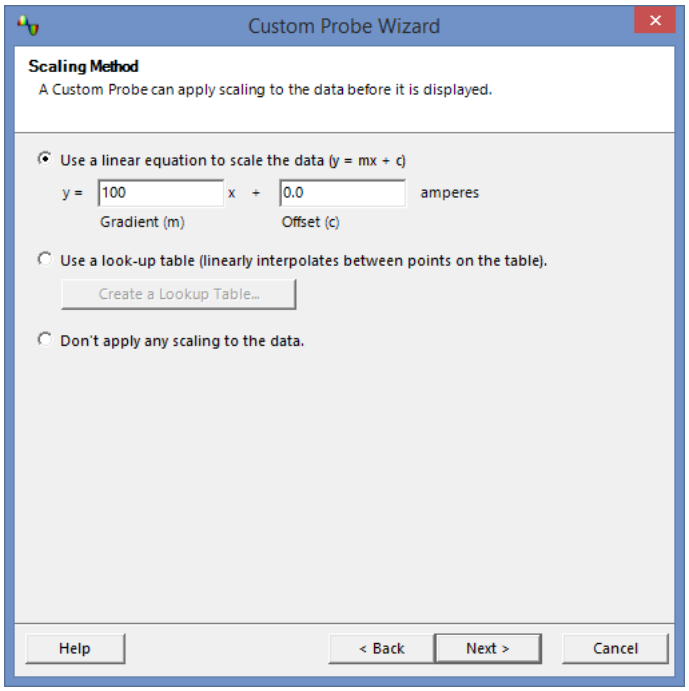
Both the linear equation and lookup table methods are covered in this section to highlight the differences.

1.1 Linear equation

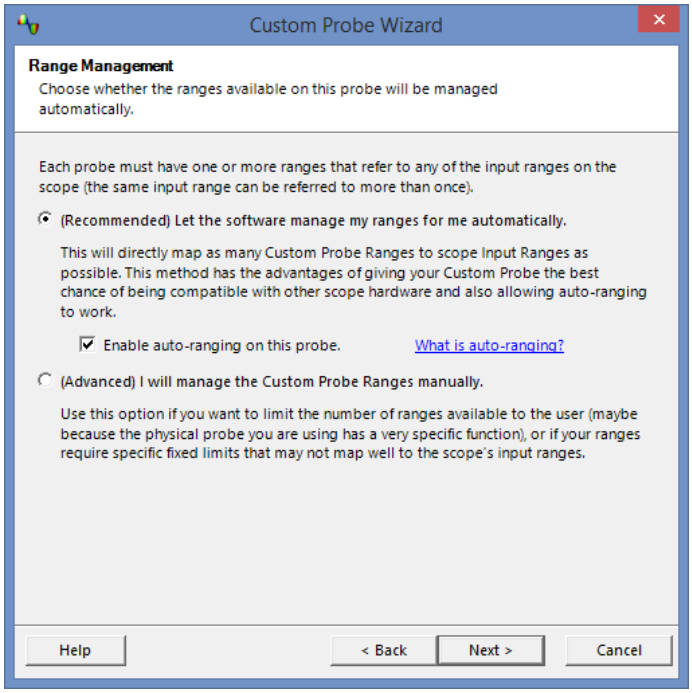
The image below shows the equation y = mx + c, where m is the gradient and c is the offset.

The gradient is the multiplication factor and in this case is 100, since the probe is 10 mV per 1 A.

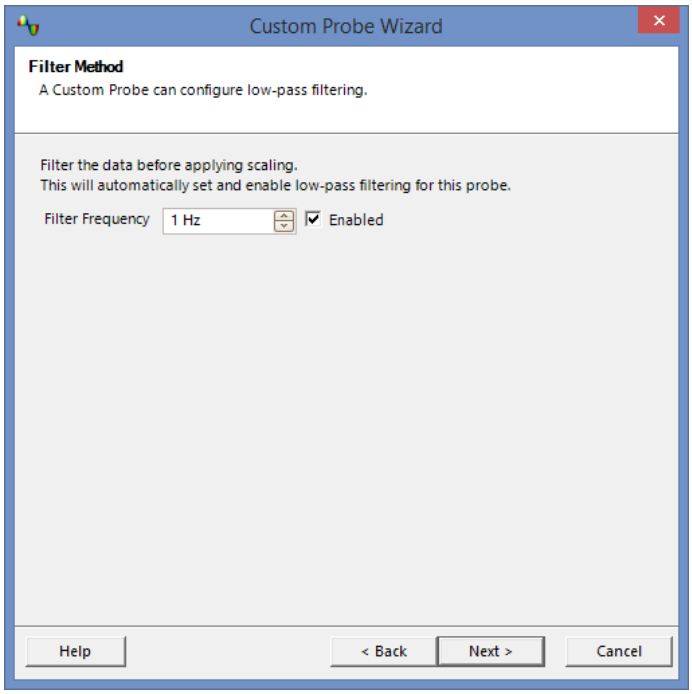
The offset is used to zero the effects of a DC offset on a signal. In this case it is 0.



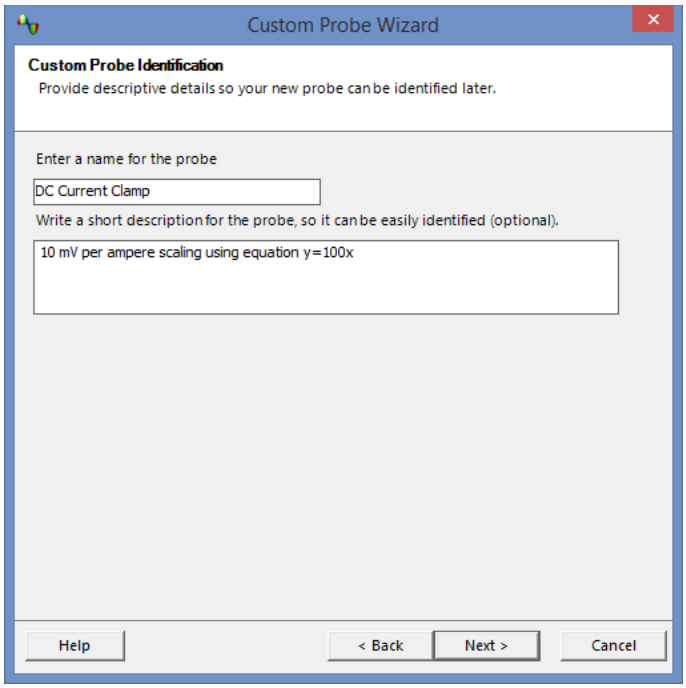
The next screen defines how the ranges will be managed. The **Recommended** option lets the software manage these ranges automatically. When **Enable auto-ranging on this probe** is selected, PicoScope continually monitors the input signal and adjusts the range when necessary to allow it to display the signal with maximum resolution. The **Advanced** option allows you to customise the ranges. This is covered in the **Advanced Custom Probes** section. In this example **Recommended** is selected.



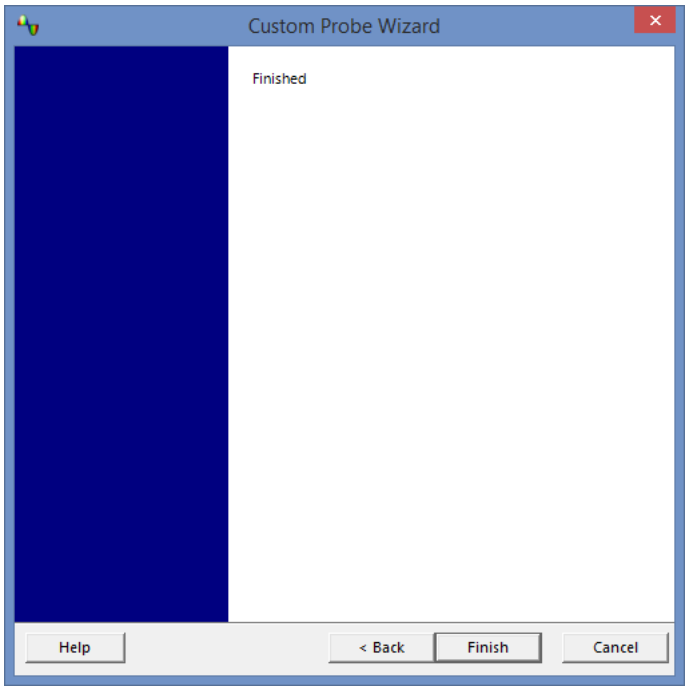
The next screen shows the **Filter Method**, which allows you to enable filtering. This is a software feature that applies a low pass filter. Setting the filter frequency only allows frequencies up to this value to be viewed, while any higher frequency components are removed. This is useful for removing high frequency noise or interference on a signal.



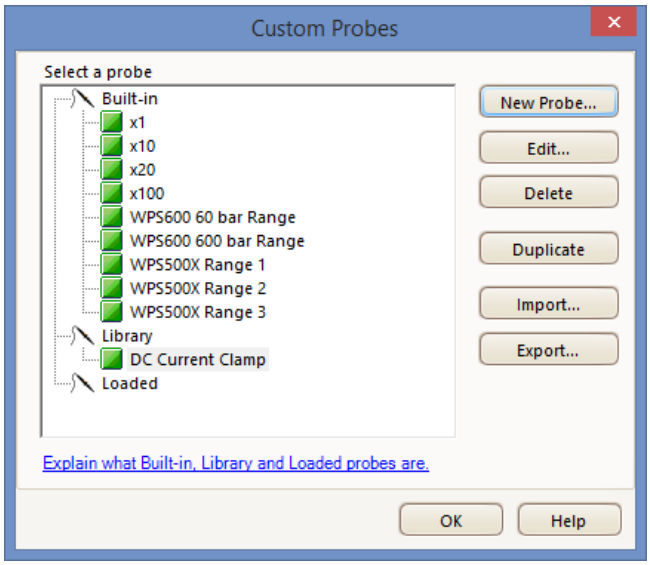
The next screen is where you can define the name of the custom probe, with the option to also add extra information:



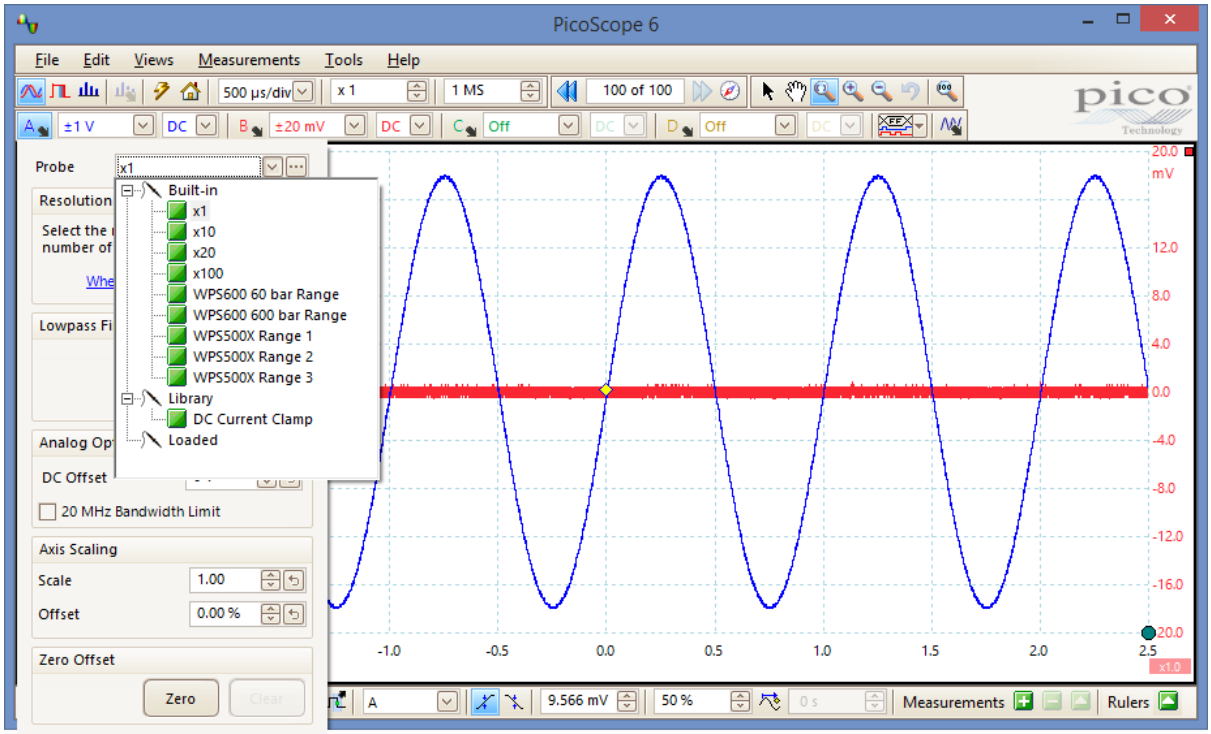
On the final screen in the **Custom Probe** wizard, click **Finish** to confirm your changes. Click **Cancel** to erase any changes made, or **Back** to make modifications:



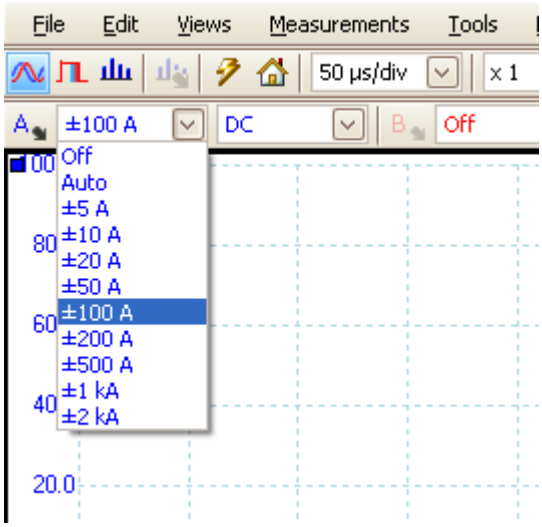
The **Custom Probes** list now shows the newly created probe under the *Library* section:



The newly created probe can now be accessed from the **Channel Options** button under **Probe** on any channel. The probe will appear under *Library*:



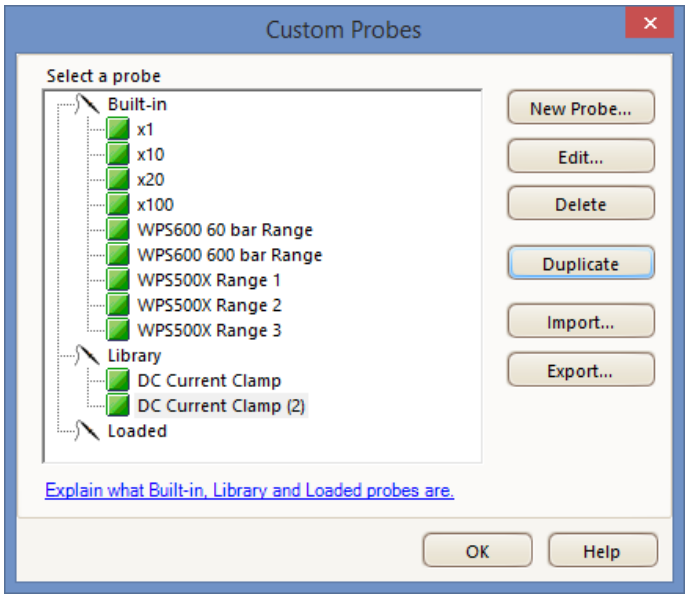
Selecting this probe reveals a new set of ranges:



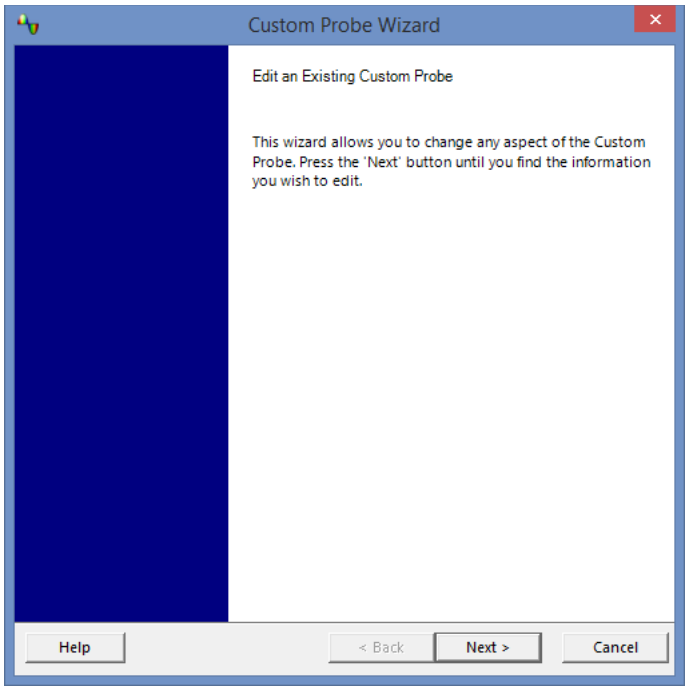
Lots of ranges appear and half of them will be of no use as the probe only goes up to 100 A. Also, the ranges that are useful show negative and positive values, but for a DC current clamp only the positive values are required.

1.2 Duplicating and editing a probe

The previous section covered the DC current clamp using the equation method. To highlight the differences we will create another probe. The first probe is duplicated and edited to use a lookup table instead of an equation. To do this, open the **Custom Probes** window, highlight the newly created probe, and then click **Duplicate**:



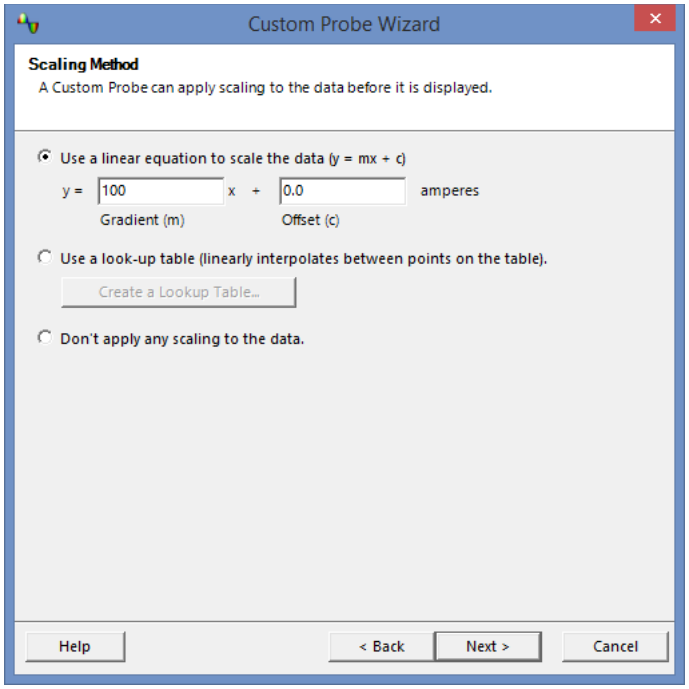
This creates an identical probe with a number (2) at the end of the name to indicate the copy number. To edit the copied probe, highlight it and click **Edit**:



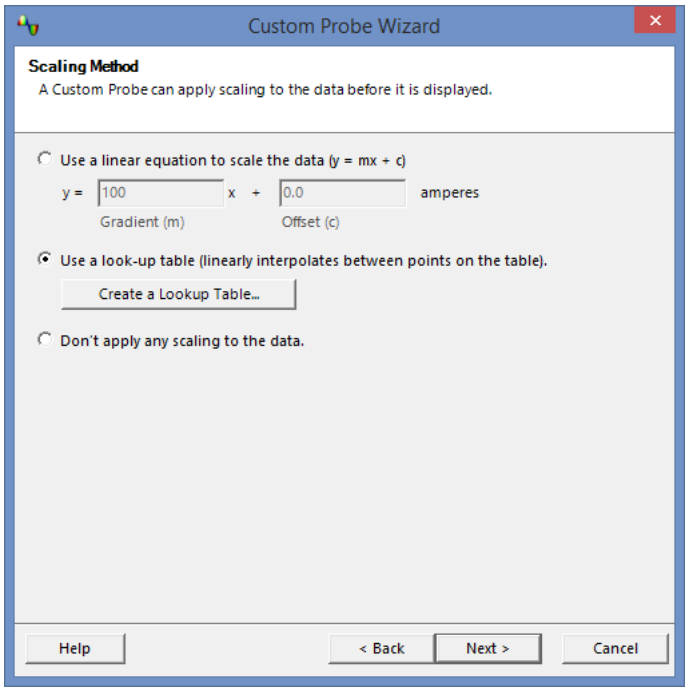
The **Custom Probe** wizard appears with the same options as a newly created probe. The only difference is that the settings for the copied probe will appear.

1.3 Look-up table

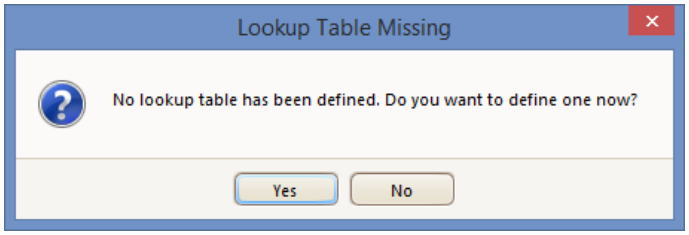
The image below shows the previously created linear equation.



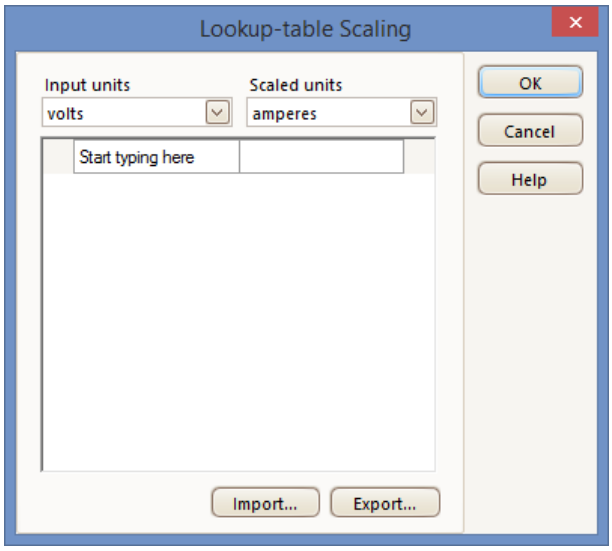
Select **Use a look-up table** to enable the **Create a Lookup Table…** button:



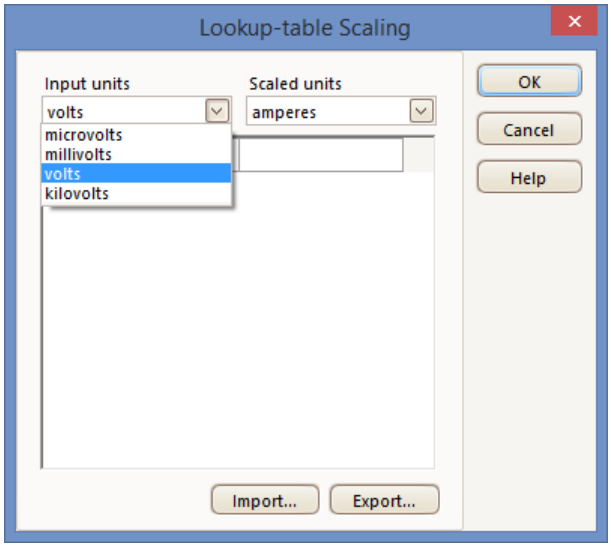
If you click **Next** without selecting **Create a look-up table…** you will be prompted to do so:



Select **Yes** or **Create a Lookup Table** to open a blank table, allowing you to create a customised lookup table. Click **Import** to save the scaling file as a CSV or text file, or **Export** to export preconfigured CSV or text files:

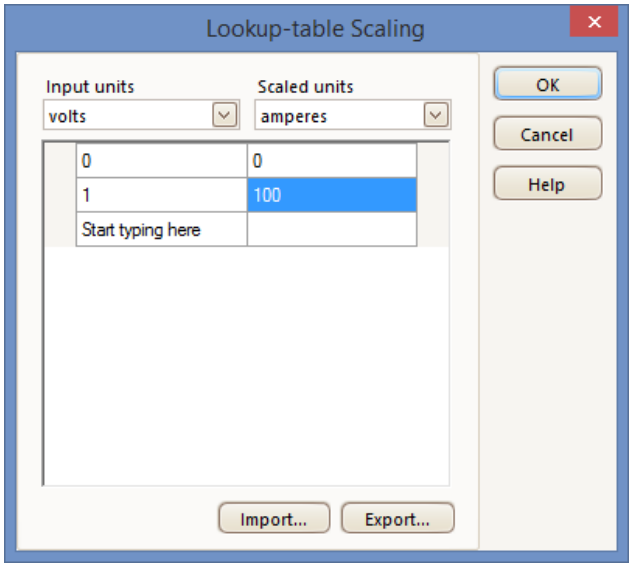


Alternatively the probe can be defined within the table provided. This can be populated with the raw input readings and then scaled readings as the table below shows. The first things that have to be selected are the units for the *Input* and *Scaled* readings. A range of input and scaled units is available:

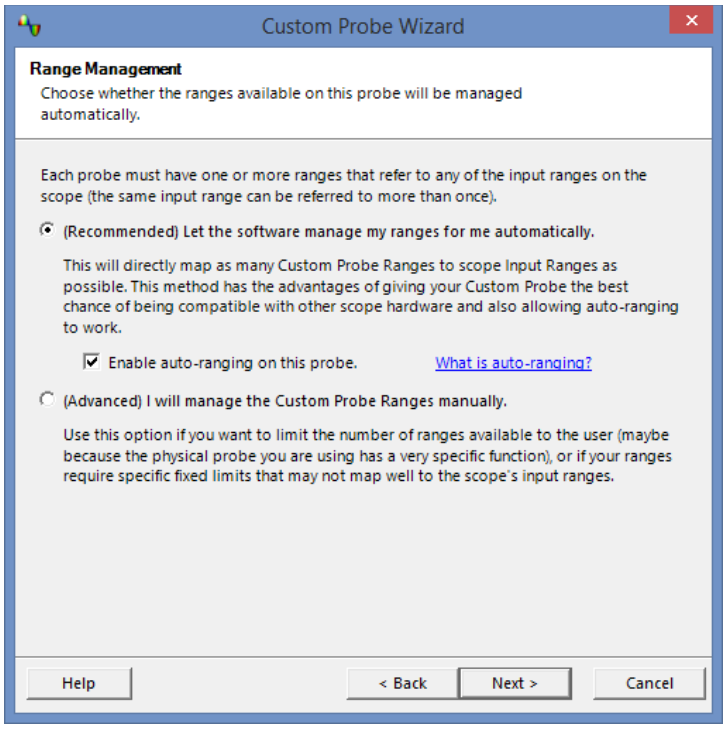


In the DC current clamp example, the maximum current to be measured is 100 A and at 10 mv/A this equates to 1 V maximum. Therefore the input units are in volts. The scaled units will remain in amperes.

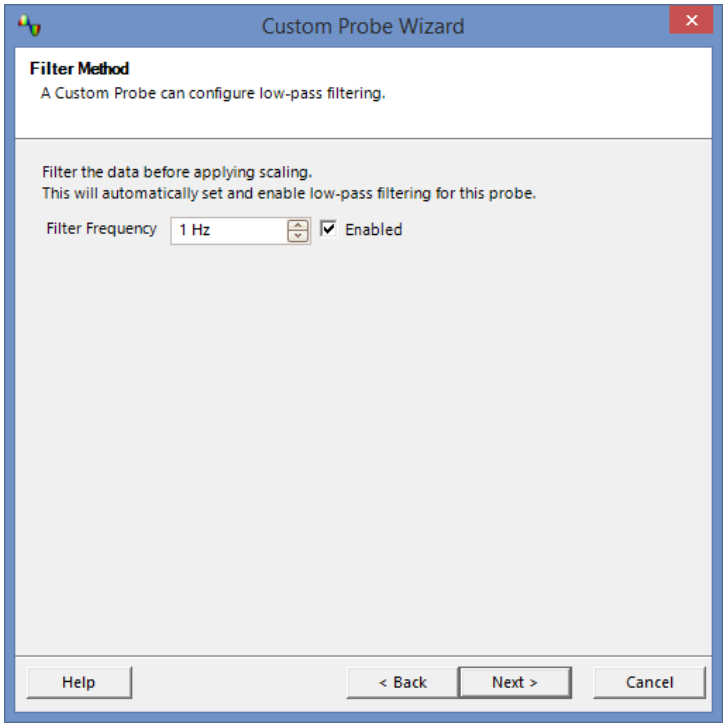
At least 2 pairs are needed to produce a scaling file assuming the relationship is linear. If the relationship is not linear then multiple points are required. In this case it is a linear relationship and 2 points are chosen. The first set of values is 0 V = 0 A and the next will be 1 V = 100 A:



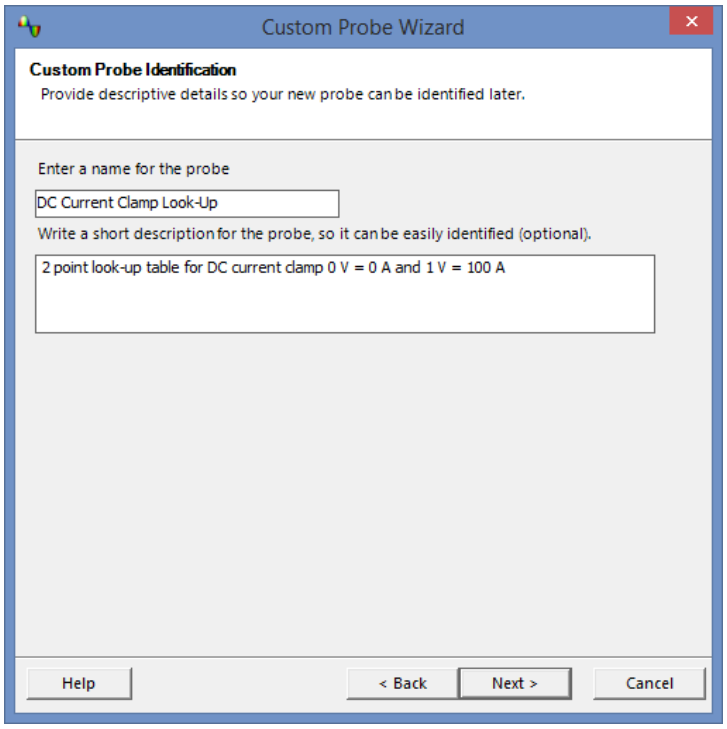
Once the scaling is set, you can adjust the ranges in the **Range Management** screen. As before, the **Recommended** option is selected:



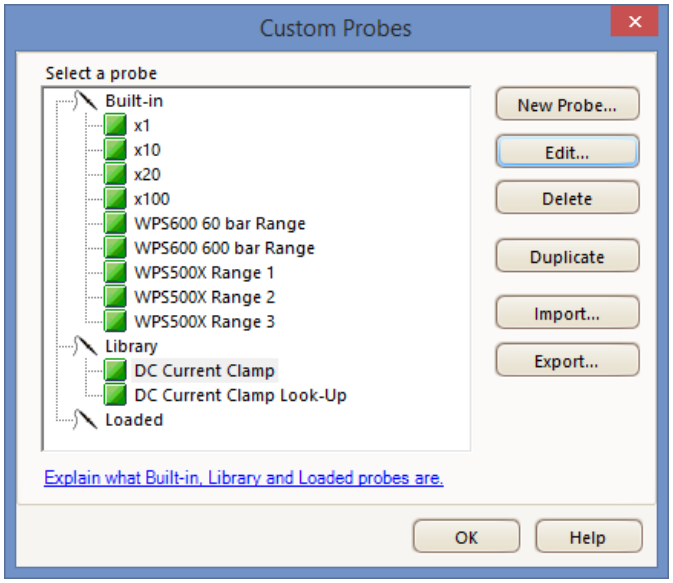
Filtering can also be enabled:



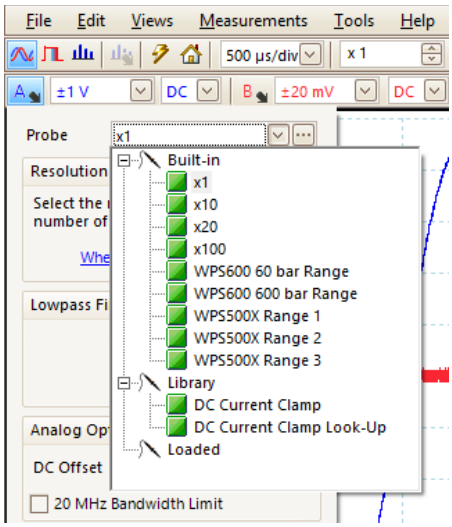
The name and description can now be edited:



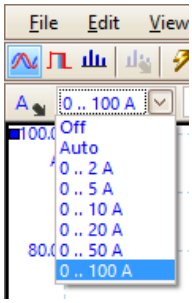
Once completed, the modifications made to the probe will appear in the library list:



Accessing the probe from the **Channel Options** button shows the newly added probe in the list:

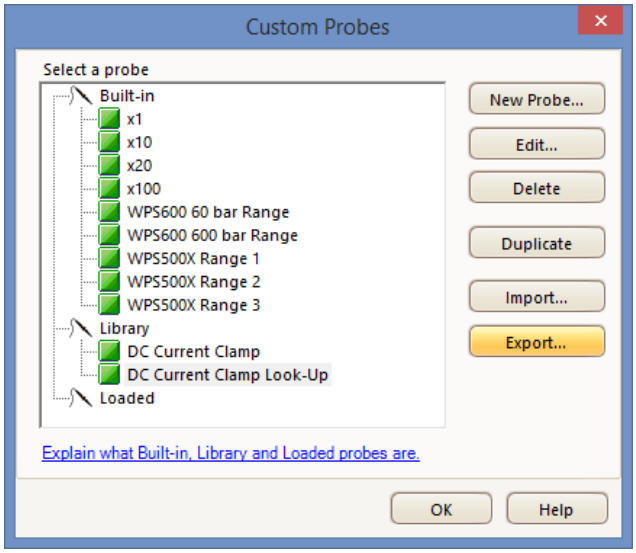


Select the new probe. The list of ranges now has a maximum current of 100 A and only shows positive values:

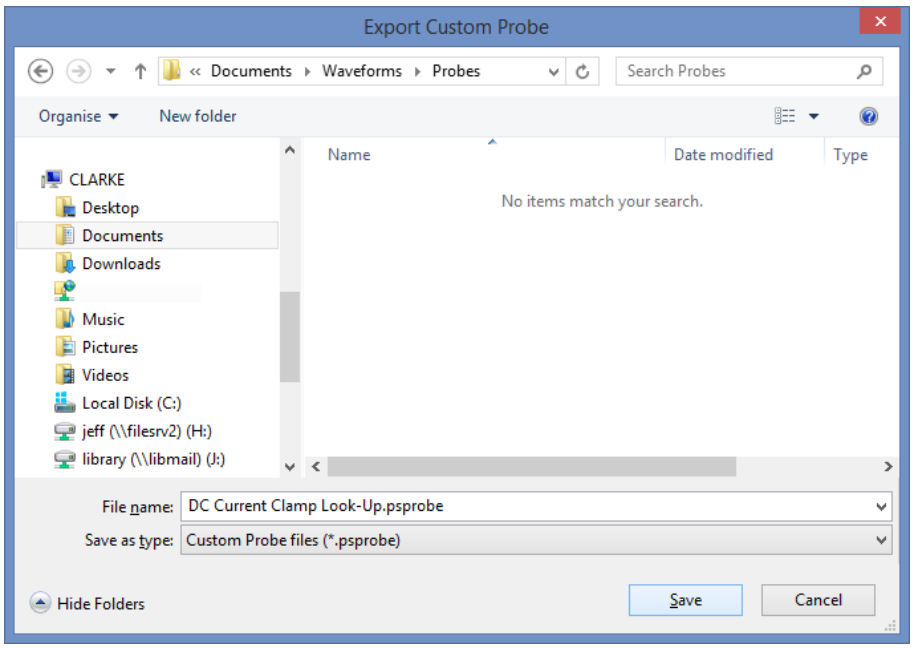


1.4 Importing and exporting a probe

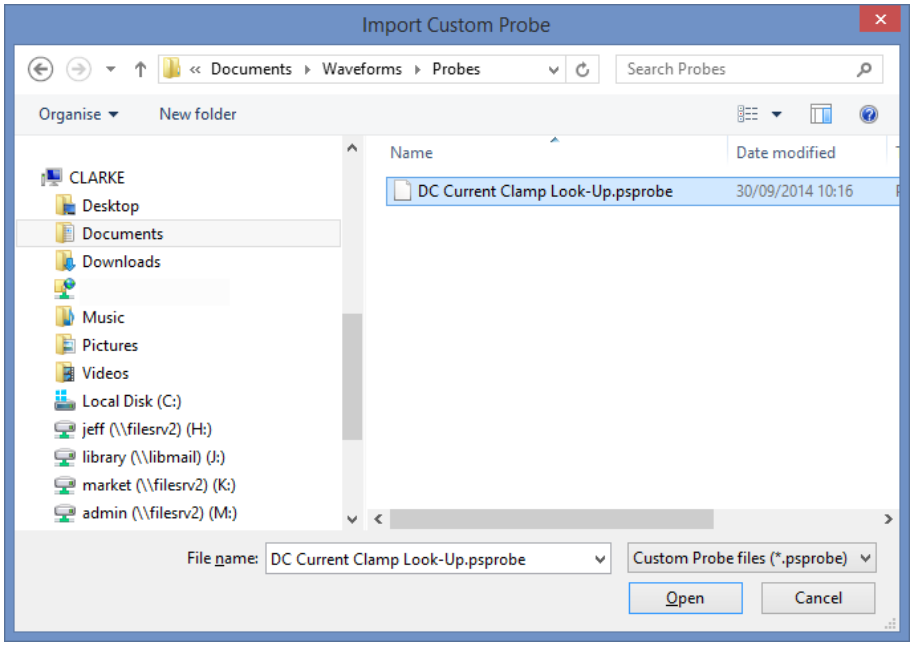
To save a probe, highlight the required probe and then click **Export:**



The default location is under the **Probes** folder which is located in the **Waveforms** folder under **My Documents**, but the file can be saved anywhere. The file extension of the probe is *.psprobe*:

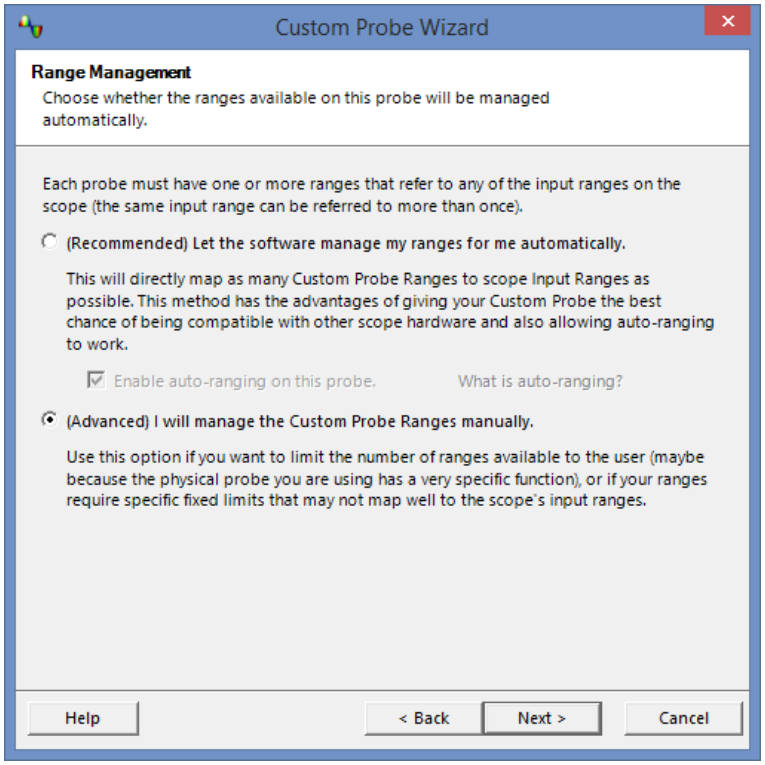


To import a probe, simply click **Import** and locate the destination folder. Any probes with the *.psprobe* extension will appear in the list:

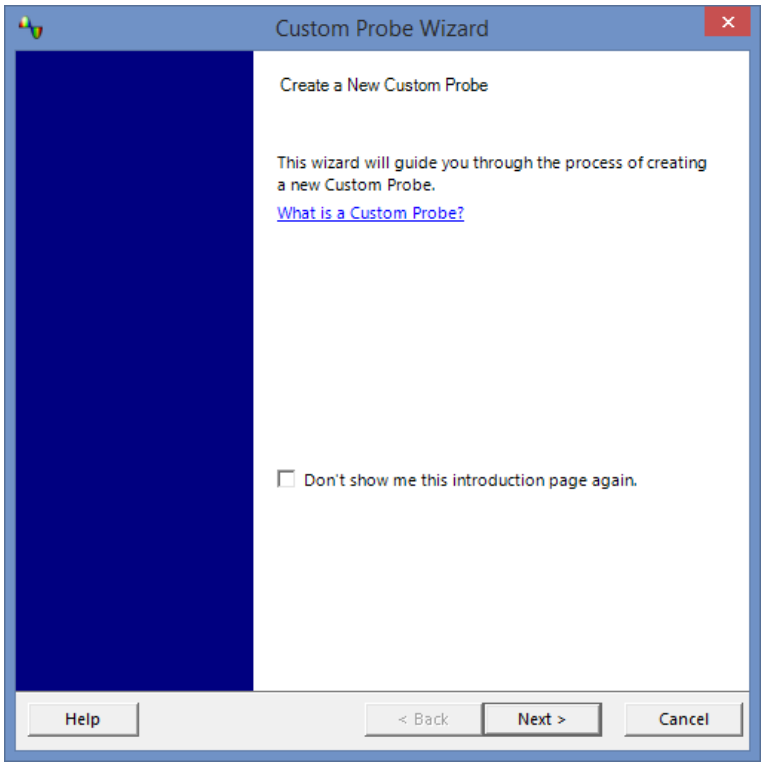


2 Advanced custom probes

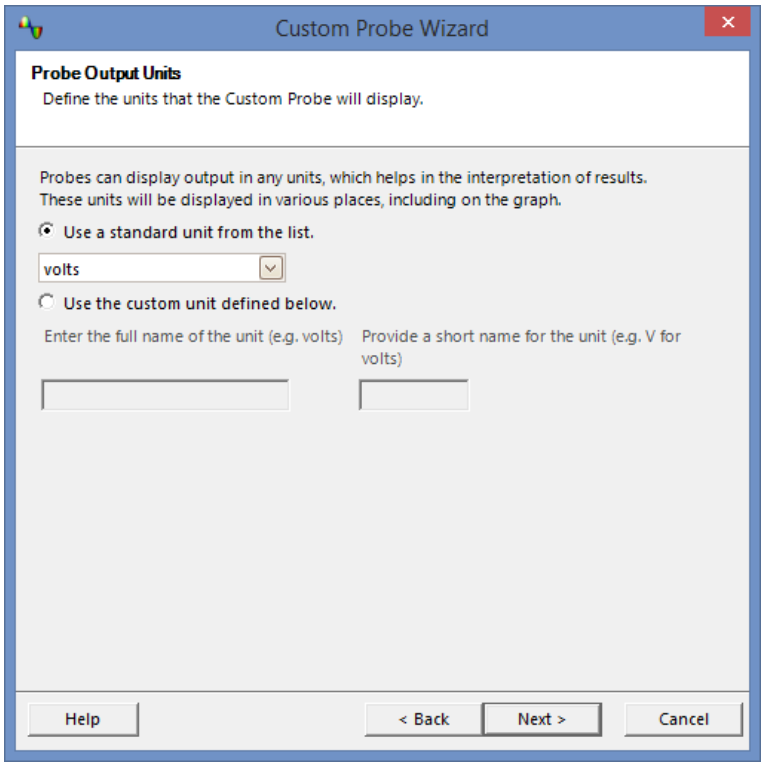
Previously when the **Range Management** section was encountered the automatic settings were chosen. This section covers the manual customisation of probe ranging. To describe how this works a custom probe will be created that only shows positive voltages in ranges from 0 V–1 V to 0 V–20 V.



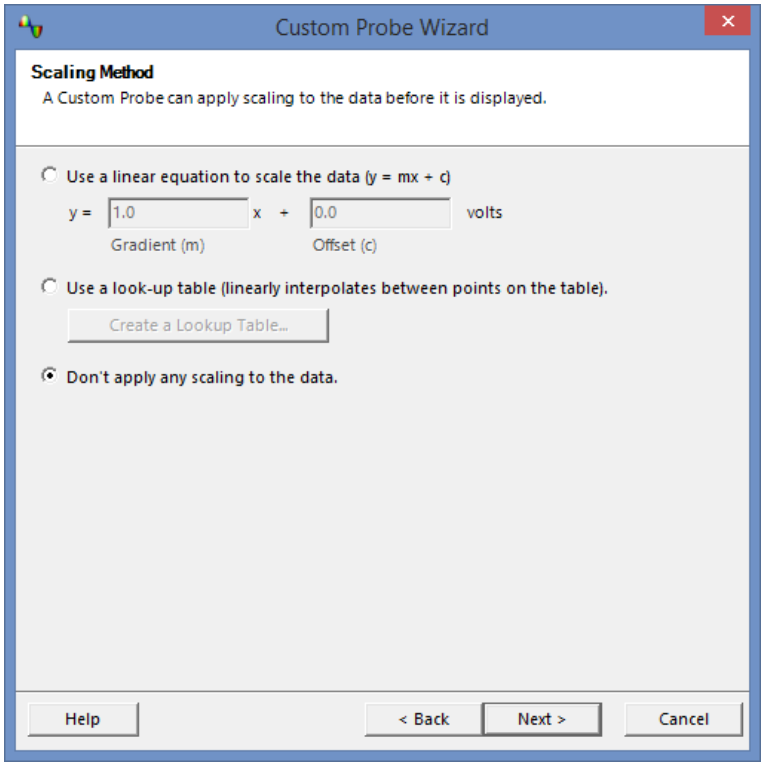
Going through the process of a custom probe:



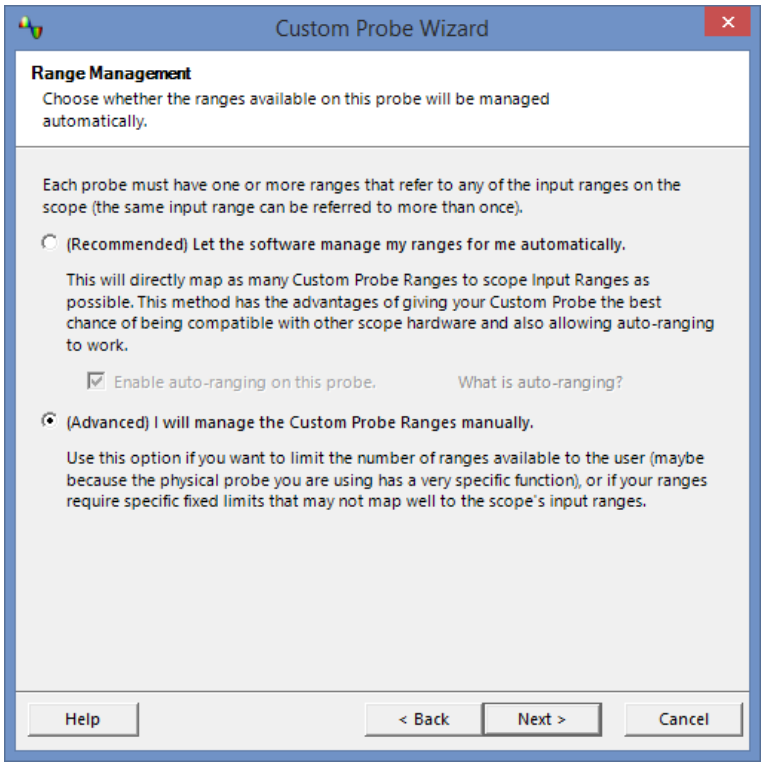
The units will be the same since only the negative voltages will be removed from view:



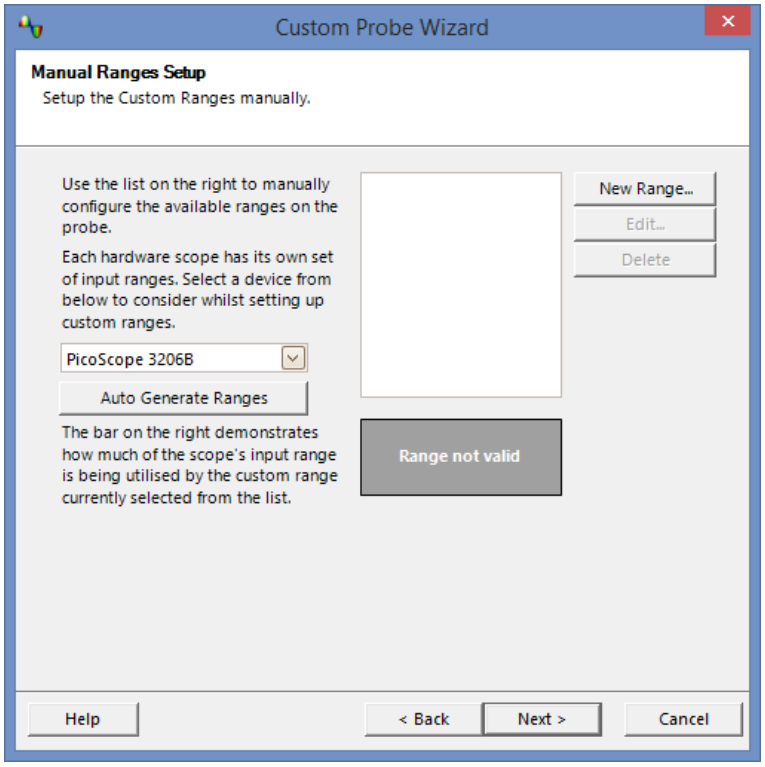
No scaling will be applied:



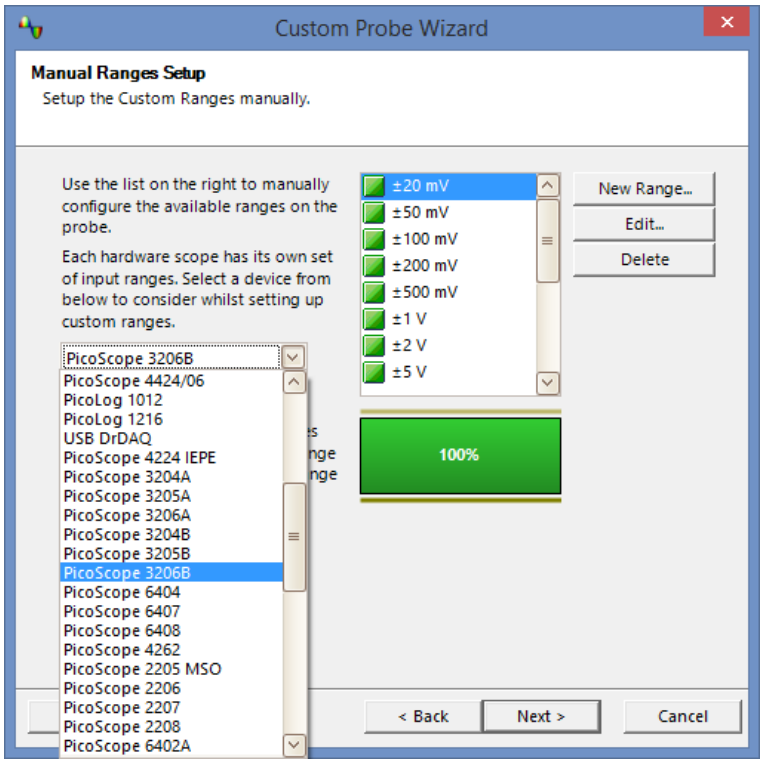
The advanced range management can now be selected:



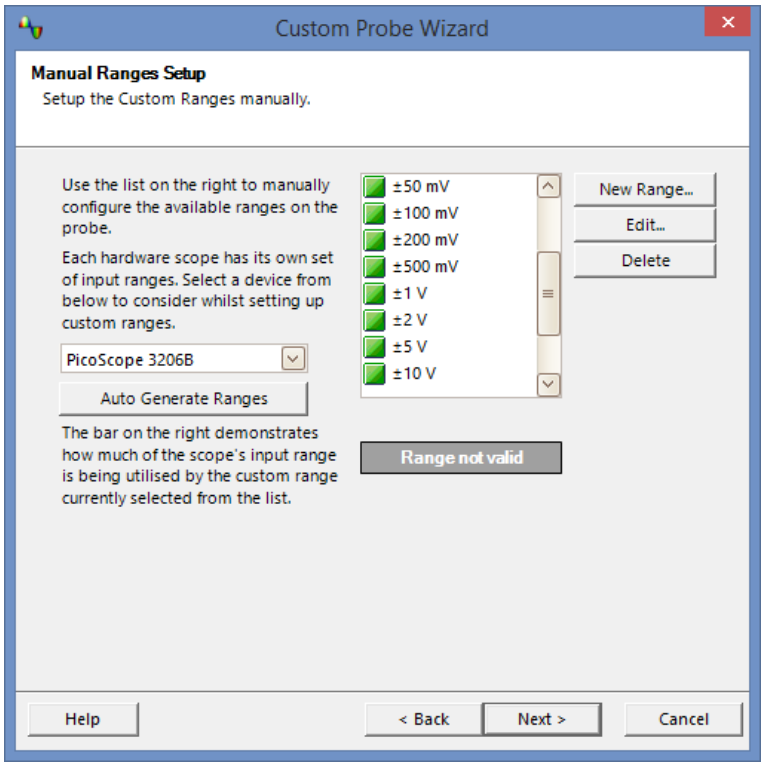
The **Manual Ranges Setup** shows a blank canvas:



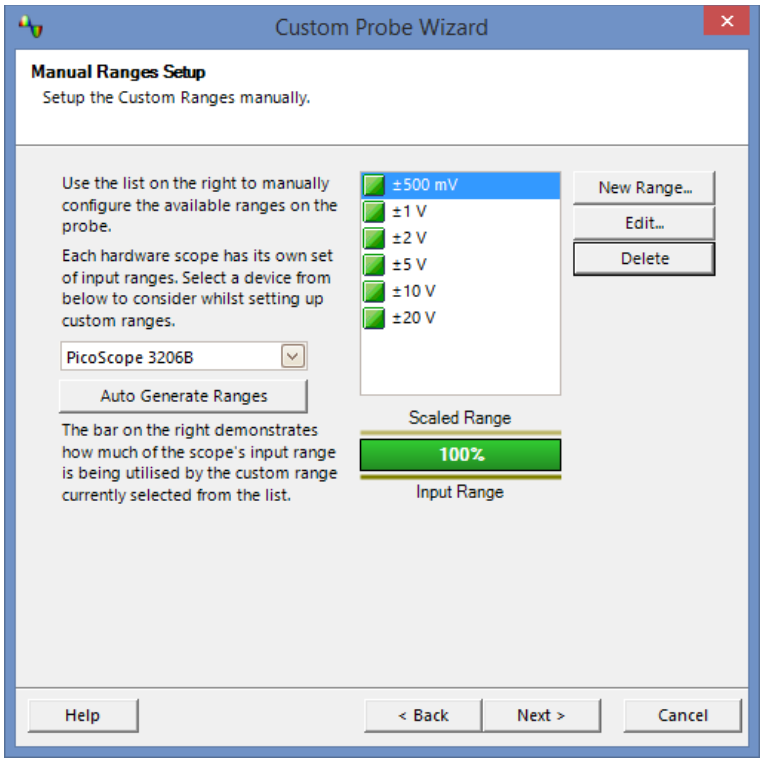
You can run the **Auto Generate Ranges** process for any product:



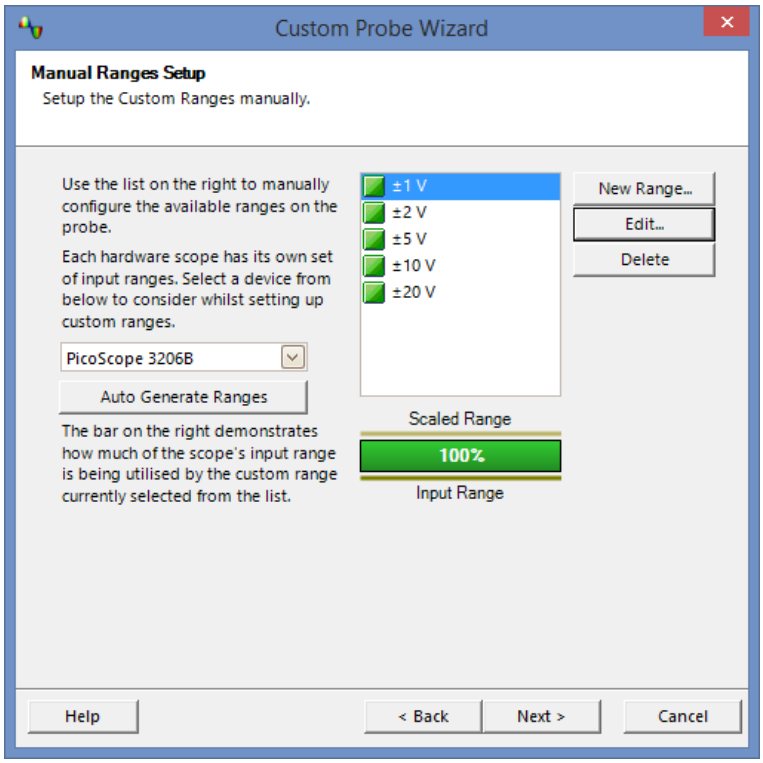
Choose the PicoScope 3206B and click **Auto Generate Ranges** to automatically create a list of suitable scaled ranges for each of the hardware ranges. In this example there is no scaling and the full set of hardware ranges is presented:



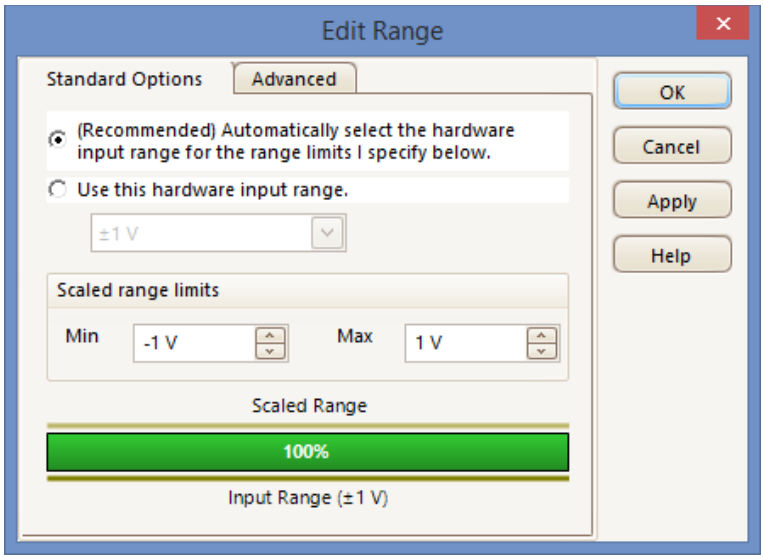
In this example only voltages greater or equal to 1 V are used. The remaining ranges can be deleted; select a range and click **Delete**:

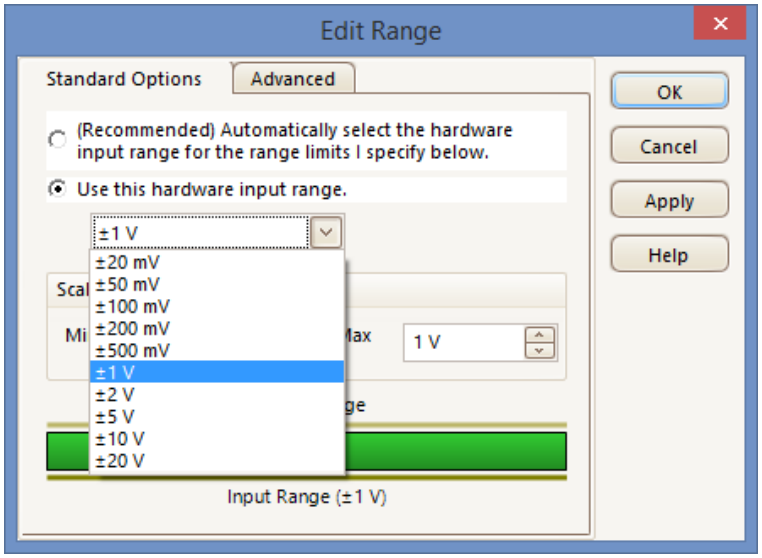


Once complete, the negative component of each of these ranges can be removed. To modify the existing ranges, select the required range and click **Edit**:



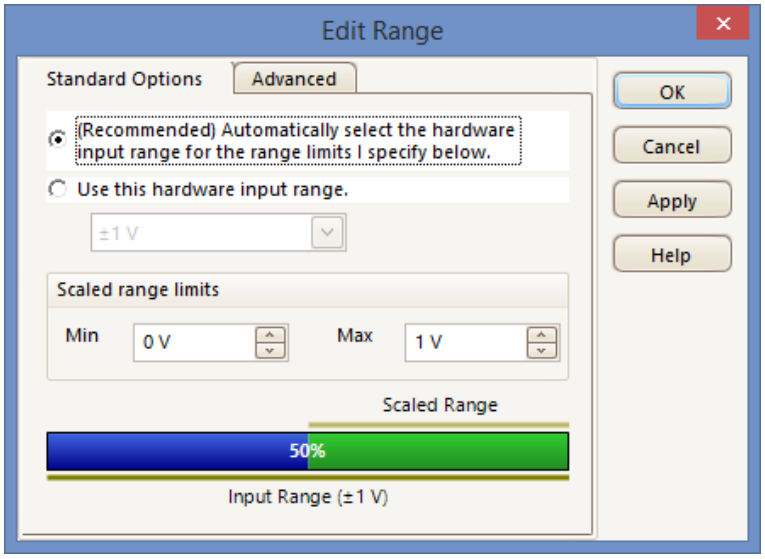
Double-click ±1 V to bring up the **Edit Range** window. You can use the recommended values or manually adjust the hardware range:



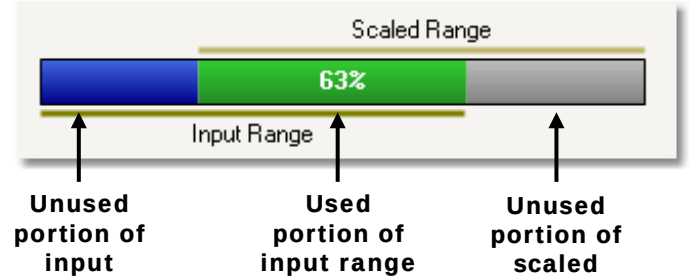


In this case the recommended range is suitable as this will give the greatest voltage and resolution for this range.

To remove the negative component of this range, put 0 as **Min** in the **Scaled range limits**. Once this is selected the range utilisation bar will show that 50% of this range is being used, as below. As this is a software option the hardware resolution will not change:



The image below shows a more detailed description of the range utilisation bar.

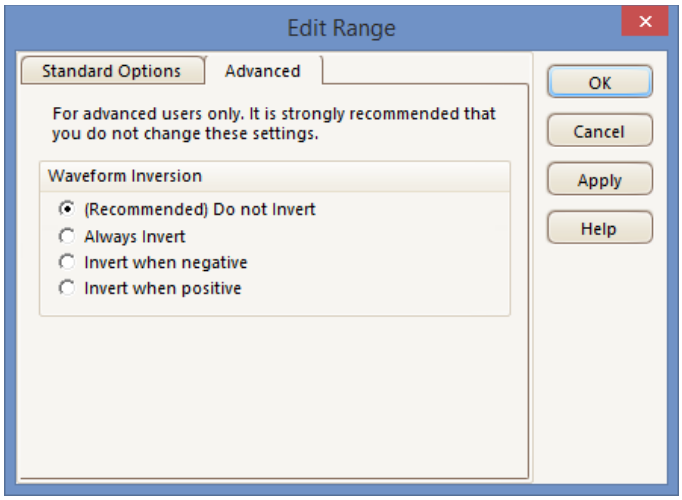


Green The section of the input range that is used by the scaled range. This should be as large as possible, to maximize the use of the scope's resolution.

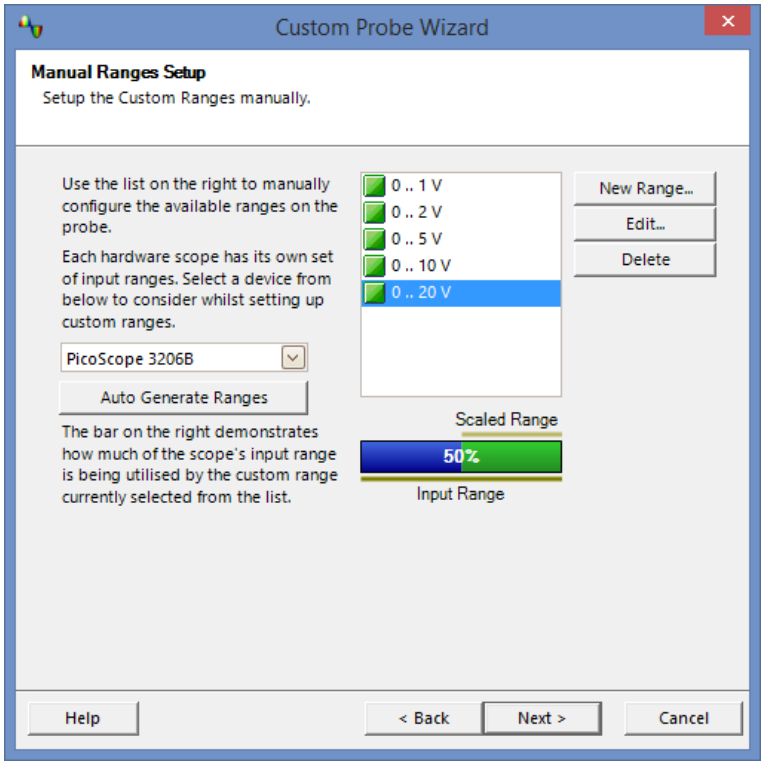
Blue Areas of the input range that are not being used. These indicate wasted resolution.

Grey Part of the scaled range that is not covered by the input range. This will result in wasted space on the graph. The range utilisation bar may not represent this area accurately when non-linear scaling is being used, so testing the scaled range limits on the scope view is advised.

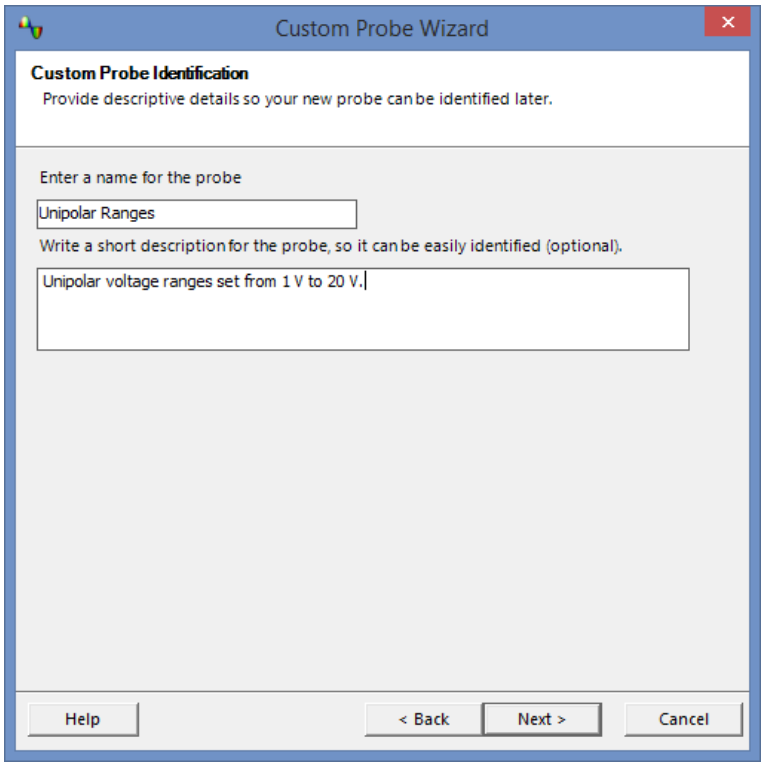
The **Advanced** tab allows you to invert a waveform, but this is not necessary in this example:



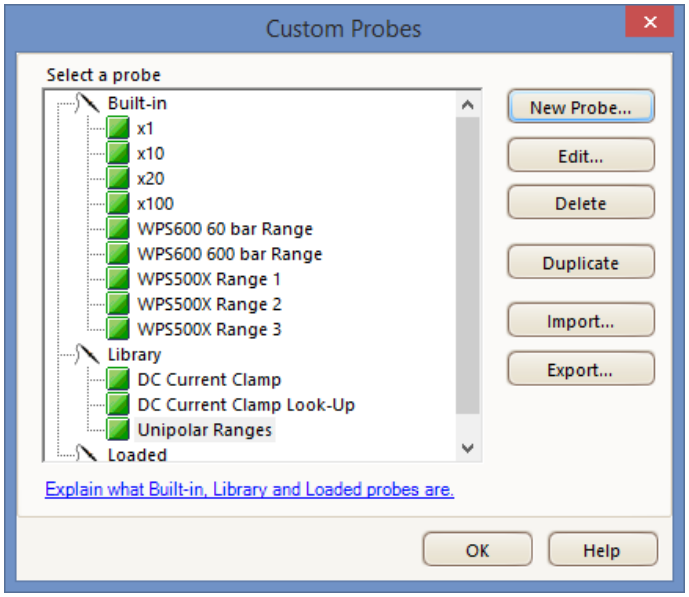
Once completed, the same process can be done for each of the other ranges. The new set of ranges will now be shown from 0 V:



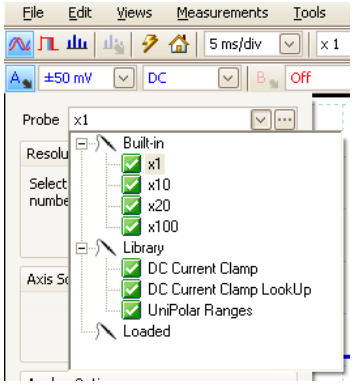
All that is left is to name the sensor and add a short description:



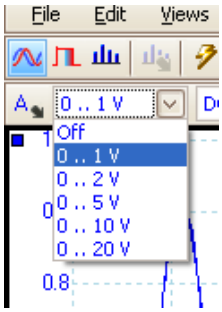
Once completed, the new unipolar Ranges will appear in the **Library** section of **Custom Probes**:



The range will now appear in channel options under **Probe**.



Click **Input Range** to see the newly created set of ranges.



The example below shows a signal that is now unipolar on the 0 to 2 V range.