Tips & Techniques

Feature: Post Process Design-Time Mode

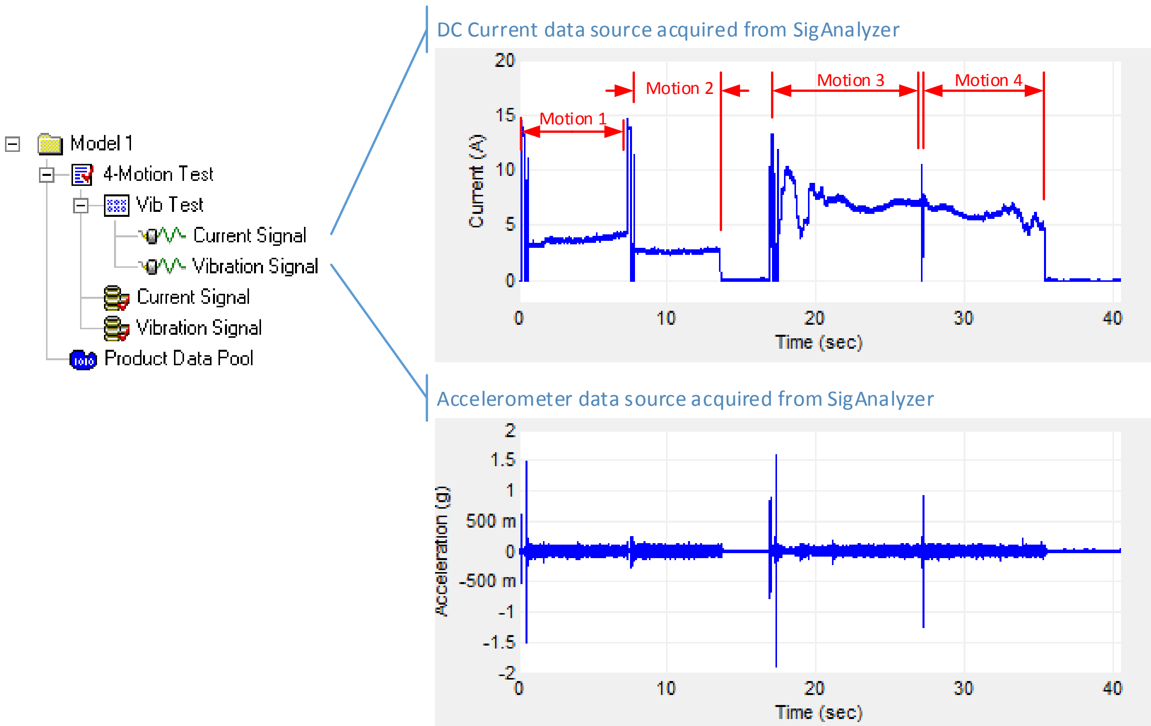
SigQC Release: 14.3.2.1

Overview

SigQC version 14.3.2.1 and greater supports a feature termed *Post Process Template Design-Time Mode*. This feature is designed to aid development of post process templates by allowing intermediate results to be viewed interactively. In addition to all of the existing post process calculation steps, data graph steps have been added to allow display of data from the output of any calculation step. While in design time mode, access to data stored in the database is readily available for temporary execution of the process. This allows users to quickly visualize the effect of calculations and verify results as the post process template is created.

To better explain the feature, an example will be developed based on a common problem that is encountered on many test stations; data trimming. For this example, assume production units are exercised through four motions; a DC motor is used to drive the unit. All that is given to SigQC is a start and stop signal to indicate the beginning and ending of the entire test. SigAnalyzer is used to acquire a DC current signal and one vibration channel using an accelerometer. The task is to separate the complete test sequence into the four separate motions using the DC current signal, then generate a maximum vibration metric over these motions.

Development of post process templates only makes sense after some raw time measurements have been acquired. This example begins after creating an initial database and collecting current and vibration data over the complete sequence of motions. The product database is simple; one model containing one acceptance test that has two data sources. A preliminary look at the measured current signal reveals the 4 separate motions, separated by momentary open circuit conditions where the current signal falls to zero between motions. Note, the vibration signal is acquired synchronously with the current signal and reflects the transient vibration as the motor starts and stops.

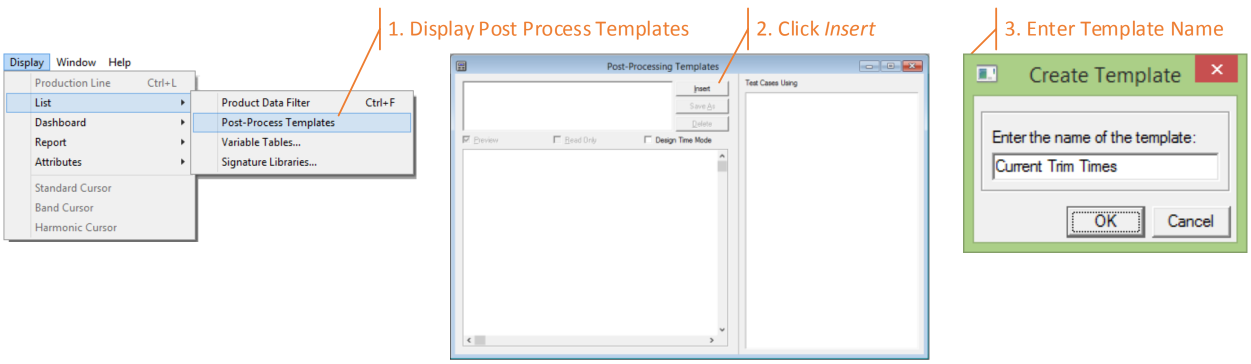


Entering Design-Time Mode

With the measurements previewed, it is now time to focus on the development of a post process template to isolate the 4 separate motions. The goal of this post process will be to calculate the times within the total test sequence that represent the start and stop times of each motion based on the current signal. For brevity throughout this document, the motions will be referred to as M1, M2, M3 and M4 for Motion 1, Motion 2, Motion 3 and Motion 4 respectively.

To begin…

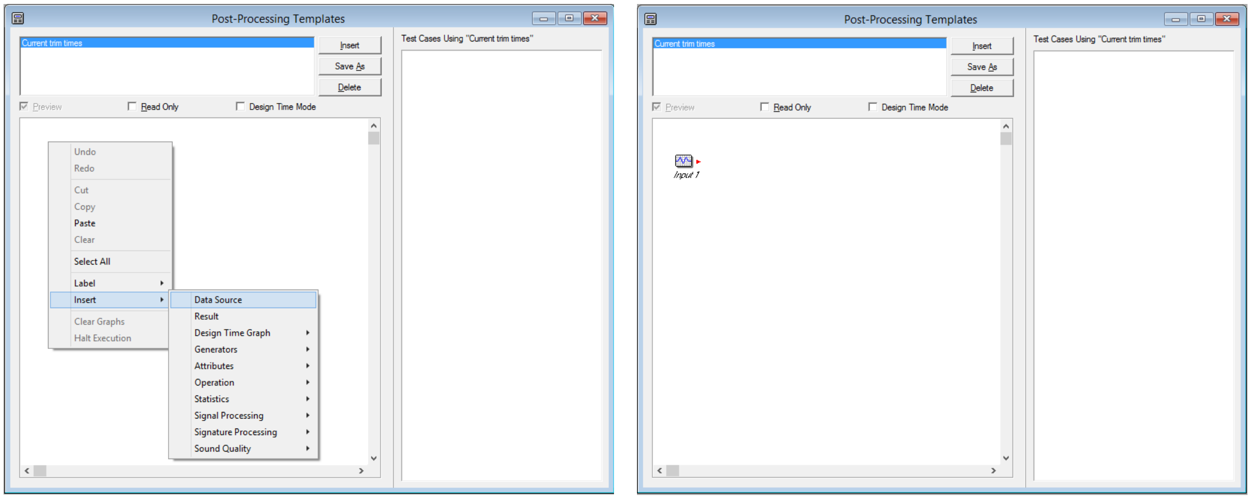
1. From the application menu select *Display – List – Post Process Templates*.
2. Click the *Insert* button to create a new post process template.
3. Enter the template name *Current Trim Times*, and then click the OK button.



Add a Data Source step

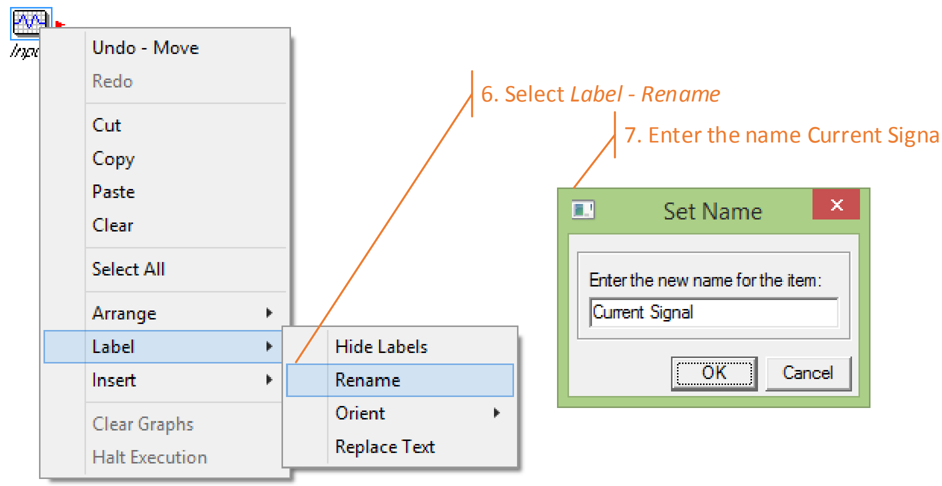
All post process templates require at least one input step, termed a *Data Source* step. In this case, we will insert a data source step that will represent the current signal.

4. From the post process template content region, right-click and select *Insert – Data Source* from the speed menu.



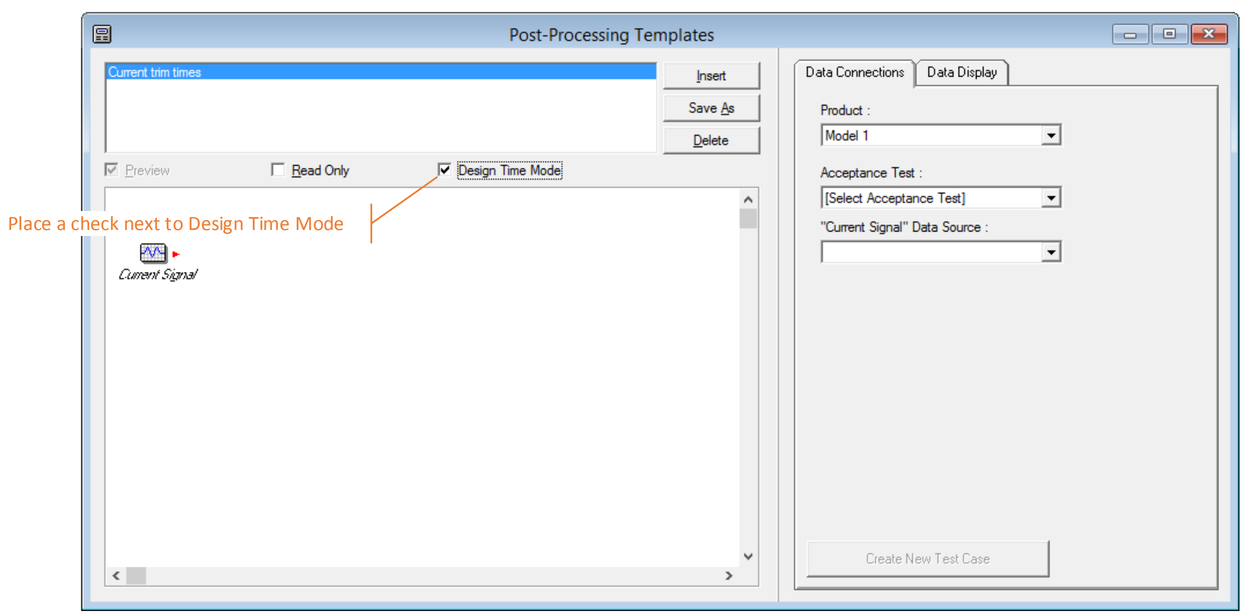
SigQC will place a data source step and assign it a default name of *Input 1*. The name of the step has no special meaning; it is just a name. Users are responsible for mapping an existing SigQC data source to a post process data source step. However, it is a best practice to name data source steps by the same name as the actual SigQC data source they are meant to represent. This helps users understand the inputs to a post process template at a glance. In addition, design time mode will automatically map post process data source steps and SigQC data sources by name when a match is found.

5. Left-click on the *Input 1* data source step.
6. Right-click within the step and select the *Label – Rename* speed menu option.
7. When prompted, enter *Current Signal* and then click *OK*.



Enable Design Time Mode

Place a check in the checkbox that reads *Design Time Mode* located just below and to the right of the post process templates list. When you enter design time mode, the *Test Cases Using "…"* region located to the right of the post process template workspace will be replaced by two tabs; *Data Connections* and *Data Display*.



Working in Design Time Mode

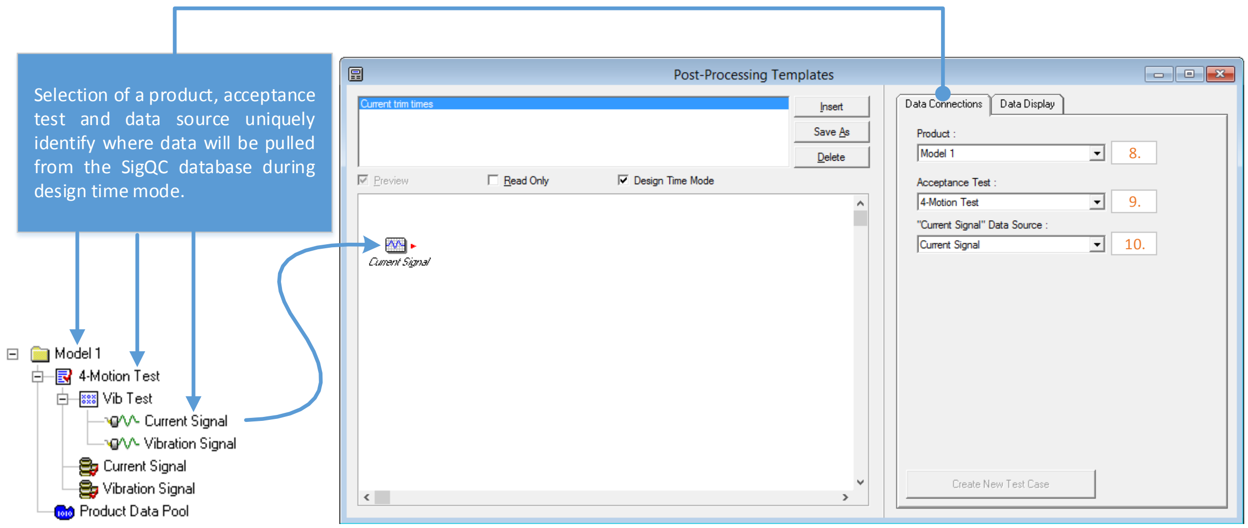
Define Data Connections

The *Data Connections* tab is used to assign the data sources that are associated with each input step of the active post process template. In other words, it lets you specify where you want to pull data from the SigQC database to feed the inputs of the calculation. Where to pull the data is defined by three pieces of information that must be chosen in the following order:

a) A product model
b) An acceptance test within the targeted product model
c) A data source within the targeted product and acceptance test for each input step

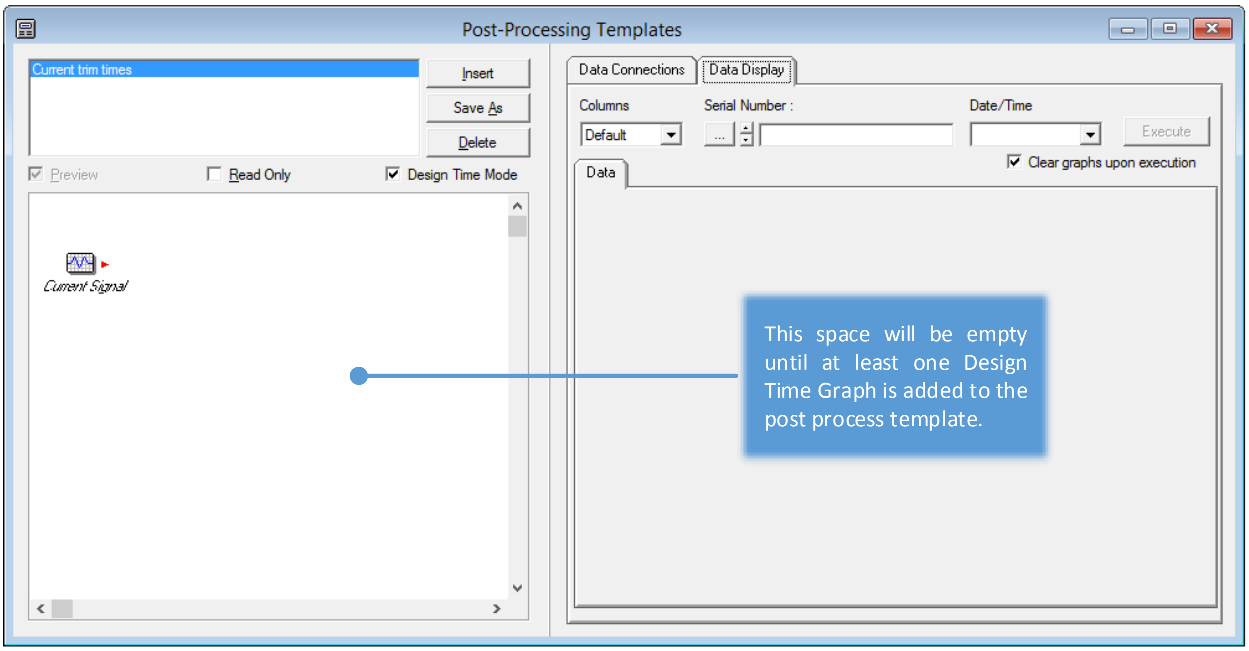
In this example, the post process template has only one input step, named *Current Signal*. To establish a connection of this input to the *Current Signal* measurement of the *4-Motion Test* acceptance test within the product *Model 1*:

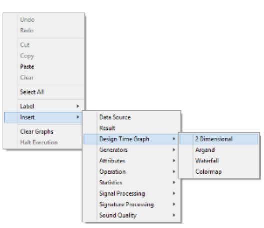
8. From the *Product* drop down list, select *Model 1*.
9. From the *Acceptance Test* drop down list, select *4-Motion Test*.
10. Within the *"Current Signal" Data Source* drop down list, notice that *Current Signal* data source has been selected automatically. Once a product model and acceptance test have been selected, design time mode searches for a data source with the same name as the input step. Since we renamed the input step from *Input 1* to *Current Signal* above, the automatic assignment was made. Had we left the step named as *Input 1*, the association would need to be made by selecting the *Current Signal* data source from the drop down list.



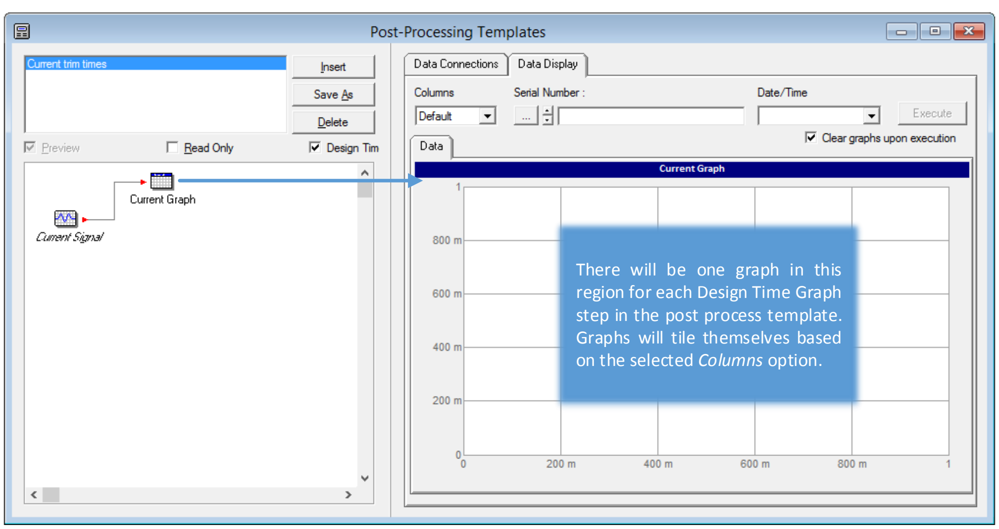
Create a Design Time Graph

Click the *Data Display* tab to show its content. When initially shown in this example, the content of the *Data* tab is empty. This region will be populated as you insert *Design Time Graphs* into the active post process template.





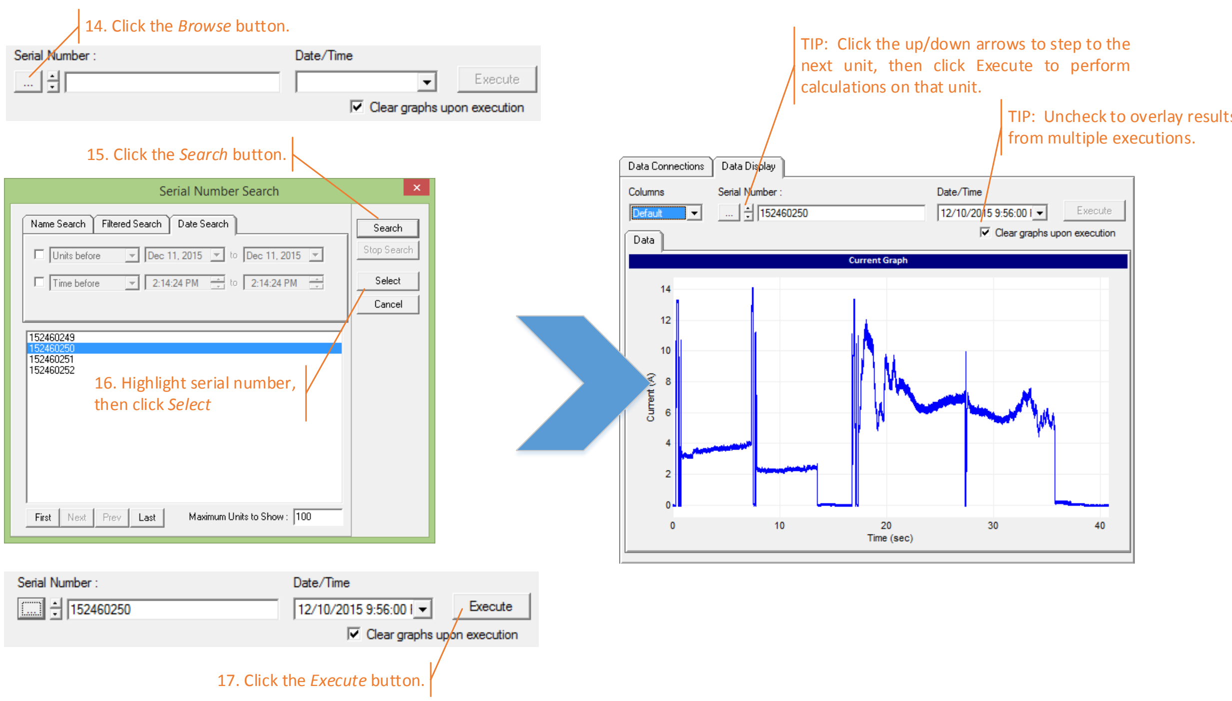
11. Right click within the post process template workspace and select *Insert – Design Time Graph – 2 Dimensional*. When inserted, notice that an empty graph has been added to the *Data* region to the right.
12. Rename the graph step to *Current Graph*. Note, the title of the graph to the right is also updated to reflect the change.
13. Connect the output of the *Current Signal* step to the input of the *Current Graph*.



Execute the Post Process Template

The final information required is to select a specific production unit to review. Production unit selection is performed using the *Serial Number* selection controls above the graphs region. You may either type in a known serial number, or use the browse button to search for one.

14. Click the browse button (labeled "…") to search for a production unit.
15. From the search dialog, click *Search*.
16. Highlight a serial number from the production units list, then click the *Select* button.
17. Click the *Execute* button. When clicked, SigQC will:
    a) Search the database for the targeted production unit.
    b) Locate data from that production unit within the database for each associated data source of the input steps.
    c) Once the production unit and all data have been retrieved from the database, the calculation executes until the last result step or design time graph step has completed.



You have just completed the basics on using *Design Time Mode*;

1) Defining data connections
2) Selecting a production unit
3) Executing the calculations.

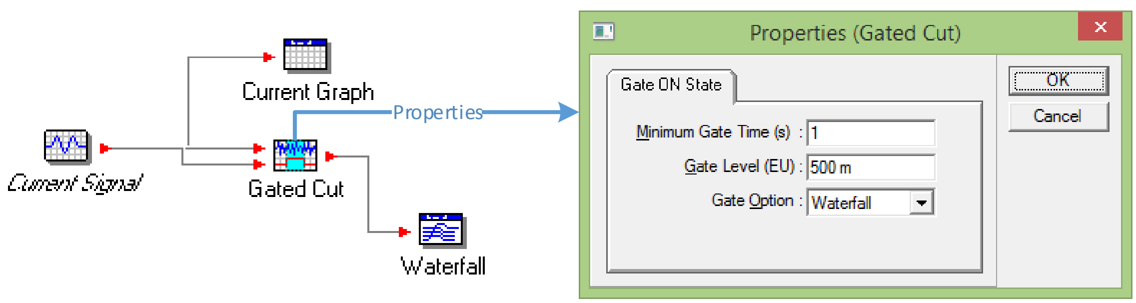
Now that the basics are covered, the remainder of this document focuses on how design time mode typically aids post process template development.

Overlaying Data Pre-Calculation and Post Calculation

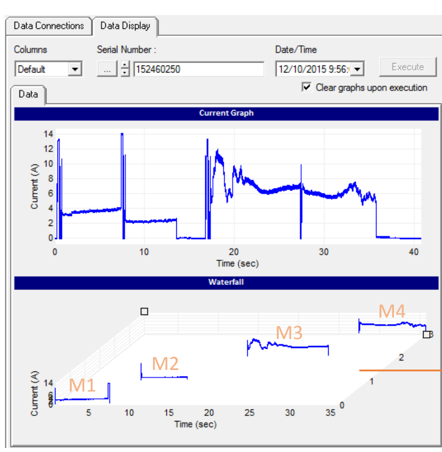
The task of this example is to separate the complete test sequence into the four separate motions so that the maximum vibration can be calculated for each motion. Job #1 is to determine the time at which each motion starts and stops, based on the current signal. From the example graph above, it is clear that the current drops to zero with each change in motion. This type of signal can be broken up with a gated domain cut step. The *Domain Gate* step requires a user defined trigger level to separate portions of a measurement. Basically, data values above the trigger level may be retained, while data below the trigger level will be removed.

To see the effect of a Domain Gate step…

1. Right click and select *Insert – Operations – Transformations – X Dimensions – Domain Gate* from the post process template speed menu.
2. Connect the output of the *Current Signal* input step to both the upper and lower input connection sites of the *Gated Cut* step. Essentially, we will use the current signal to trim itself into a waterfall of functions.
3. Double-click the *Gated Cut* step to display its properties. Select a minimum gate time of 1.0 second, a gate level of 0.5 Amps, and set the gate option to *Waterfall*. A minimum gate time of 1 second means that all data must be above the gating level of 0.5 amps for at least 1 second to be included. Sections of the current signal that are not above 0.5 amps for at least 1 second will be cut from the result signal. Selecting a gate option of waterfall means that each section of the current signal that passes the minimum gate time and level restrictions will be appended to a waterfall output.
4. Since the Gated Cut step has been configured to produce a waterfall of individual data sets, the output is best displayed with a waterfall. Again from the speed menu, select *Insert – Design Time Graph – Waterfall*.



When executed, the waterfall graph is updated to show the results from the *Gated Cut* step.



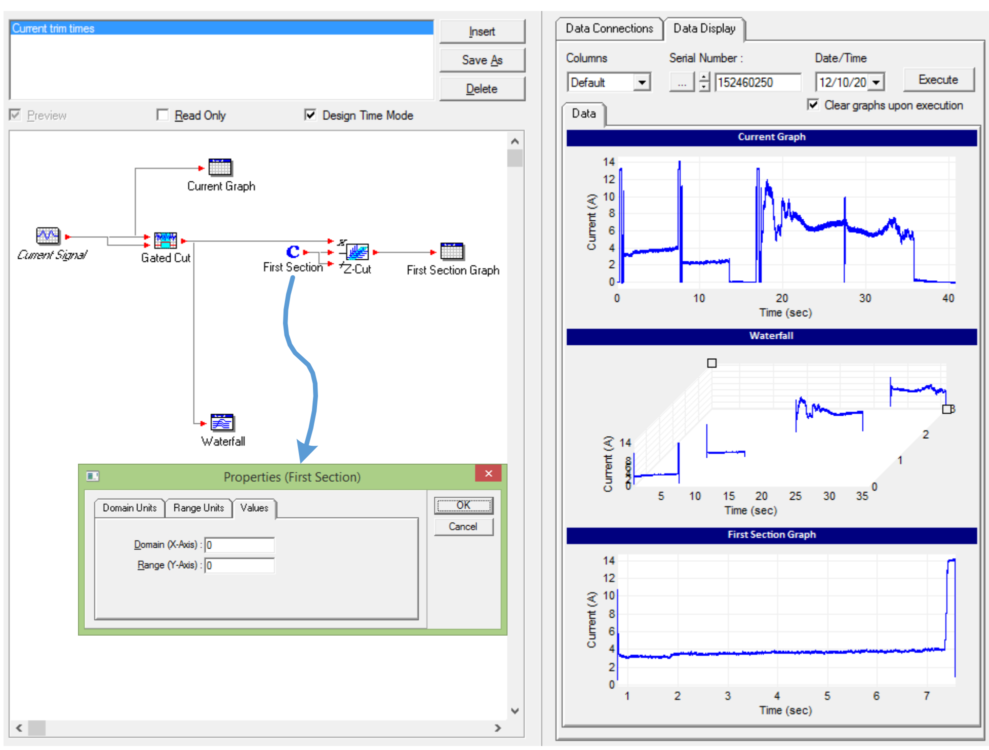
The waterfall displays each section produced by the gated cut step. There are four sections separated by the current signal dropout in this example. Note, the Z-axis ranges from 0 to 3; 0 representing the first motion, 1 representing the second motion, etc.

Extracting Start and Stop Times of Each Motion

Having segmented the current signal into distinct sections, the goal now is to find the start and stop times of each motion and create a test case for each. This can be done by extracting each section from the waterfall, and then using the *X-Minimum* and *X-Maximum* step on each one separately.

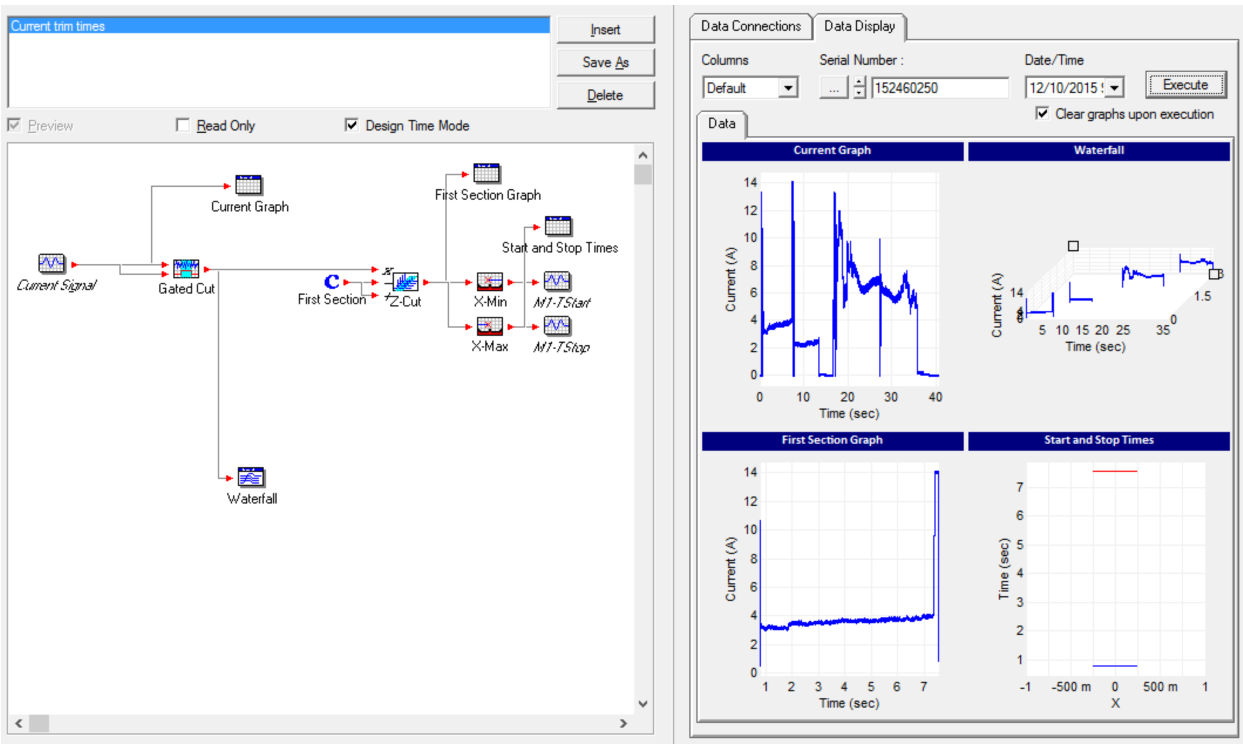
To extract a segment from a waterfall…

1. Right click and select *Insert – Operations – Transformations – Z Dimensions – Variable Z Cut* from the post process template speed menu.
2. Connect the output of the *Gated Cut* step to the top input of the new *Z-Cut* step.
3. Right click and select *Insert – Generators – Constant* from the post process template speed menu.
4. Rename the constant step to *First Section*.
5. Double-click the *First Section* step and click on the *Values* tab. Ensure that the *Range (Y-axis)* property is set to 0. When finished, click the *OK* button. The *First Section* constant essentially produces a single x-y pair whose x and y values are both defined as 0.
6. Connect the output of the *First Section* constant to both the negative and positive inputs of the *Z-Cut* step.
7. Right click and select *Insert – Design Time Graph – 2 Dimensional* from the post process template speed menu.
8. Rename the *2 Dimensional* graph step to *First Section Graph*. (Note, since we are only targeting one section by use of the *Z-Cut* step and specification of the same value as the z-limits, the result is no longer a waterfall. That is why a 2-dimensional graph is appropriate.)
9. Connect the output of the *Z-Cut* step to the input of the *First Section Graph* step. When finished, execute the calculation and the output should be as shown below.



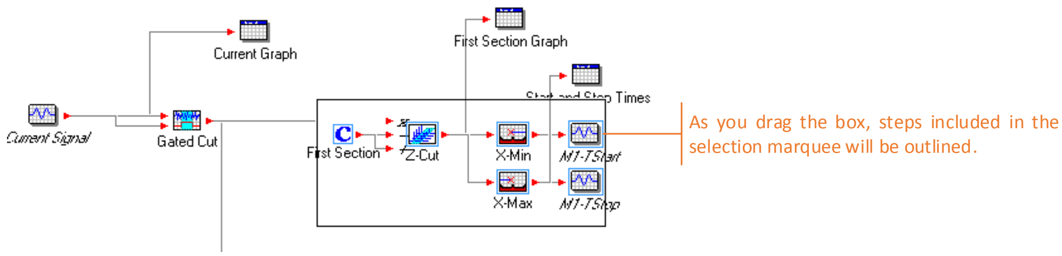
To determine the start and stop time of the section…

1. Right click and select *Insert – Operation – Peak Detection – X Minimum.*
2. Connect the output of the *Z-Cut* step to the input of the *X-Min* step.
3. Right click and select *Insert – Operation – Peak Detection – X Maximum.*
4. Connect the output of the *Z-Cut* step to the input of the *X-Max* step.
5. Right click and select *Insert – Result* twice to create two result steps.
6. Rename the *Output 1* step to *M1-TStart* and the *Output 2* step to *M1-TStop*.
7. Connect the output of the *X-Min* step to *M1-TStart* and the output of the *X-Max* step to *M1-TStop*.
8. Right click and select *Insert – Design Time Graph – 2 Dimensional.*
9. Rename the graph step *Start and Stop Times*.
10. Connect the output of the *X-Min* step and the output of the *X-Max* step to the *Start and Stop Times* step.
11. Execute the calculation and the output should be as shown below.

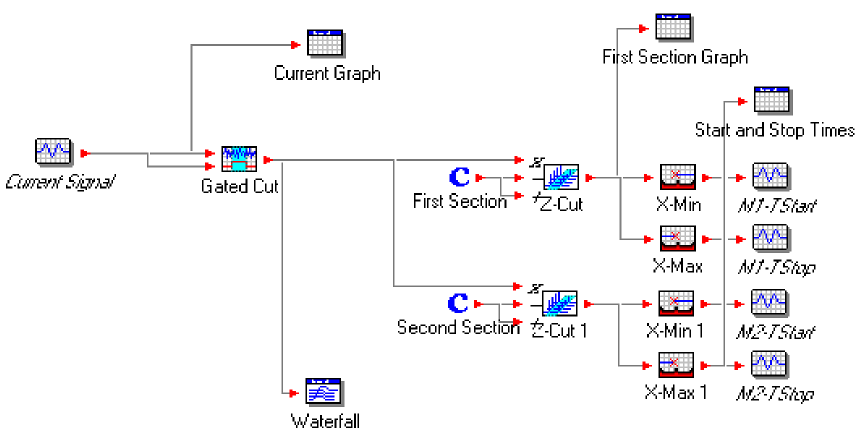


Duplicate the steps for extracting additional sections…

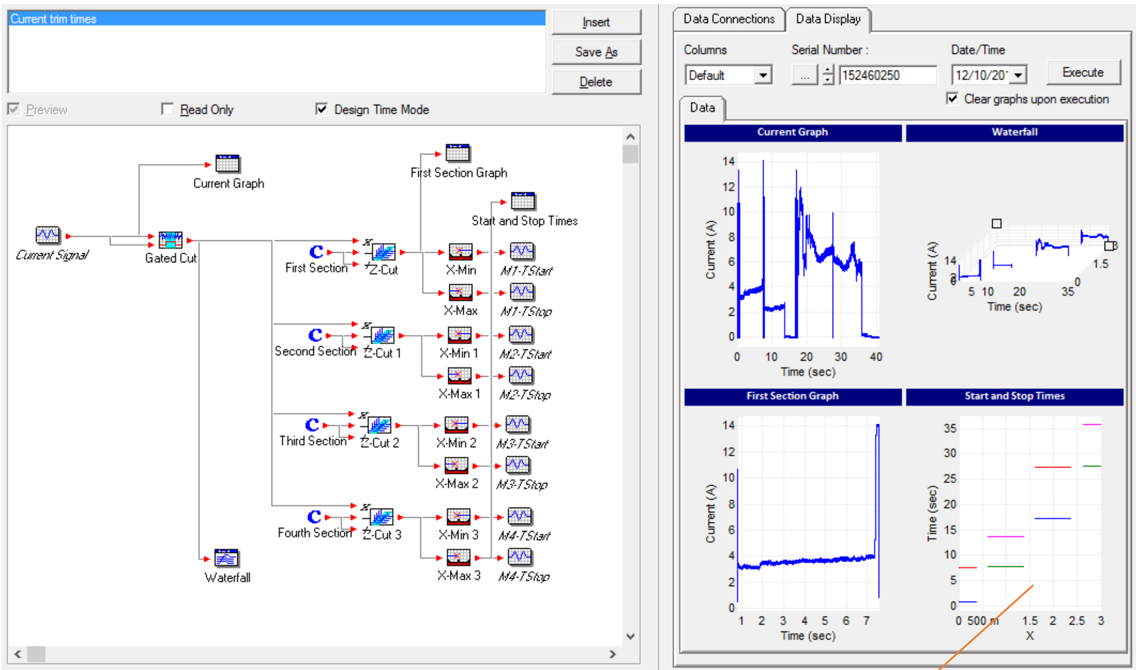
1. Using the left mouse button, click and drag a box around the 6 steps including *First Section, Z-Cut, X-Min, X-Max, M1-TStart and M1-TStop*. This will select all 6 steps.



2. With the 6 steps selected, right click and select *Copy*.
3. Right click and select *Paste*. When the paste is performed, notice the names of the steps pasted are now appended with a number. For example, *First Section* is appended to be *First Section 1*. This is because all steps within a single post process template must have a unique name.
4. Rename the step *First Section 1* to *Second Section*.
5. Double click the *Second Section* step, activate the *Values* tab, and then enter a *Range (Y-axis)* value of 1.
6. Rename *M1-TStart 1* to *M2-TStart*, and *M1-TStop 1* to *M2-TStop*.
7. Connect the output of the *Gated Cut* step to the top input (X) of the *Z-Cut 1* step.
8. Connect the outputs of the *X-Min 1* and *X-Max 2* steps to the *Start and Stop Times* graph step.



9. Repeat the duplication process (steps 1-8) for the third and fourth sections. When complete, you should have 8 result steps named *M1-TStart, M1-TStop, M2-TStart, M2-TStop, M3-TStart, M3-TStop, M4-TStart,* and *M4-TStop*. There should also be 4 unique constant steps named *First Section*, *Second Section*, *Third Section* and *Fourth Section*. The *Range (Y-axis)* value assigned to each should be 0, 1, 2, and 3 respectively.
10. Execute the calculation and the results should be as shown below.



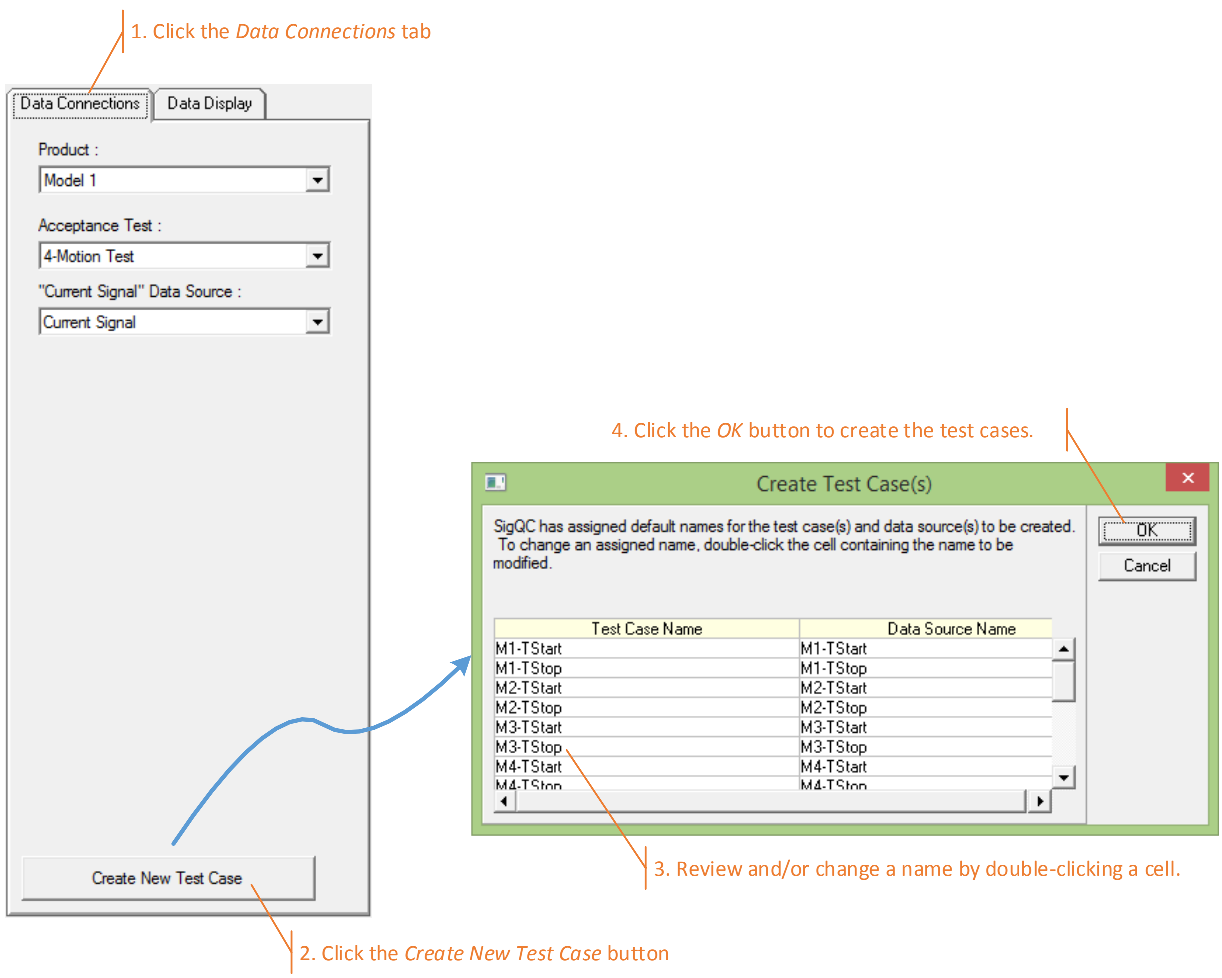
Since all 8 outputs of the *X-Min* and *X-Max* steps were attached to the *Start and Stop Times* graph step, all 8 values are overlaid within the graph. Since each output is just a single number, the data is represented as a horizontal bar within the graph.

Creating Test Cases in Design Time Mode

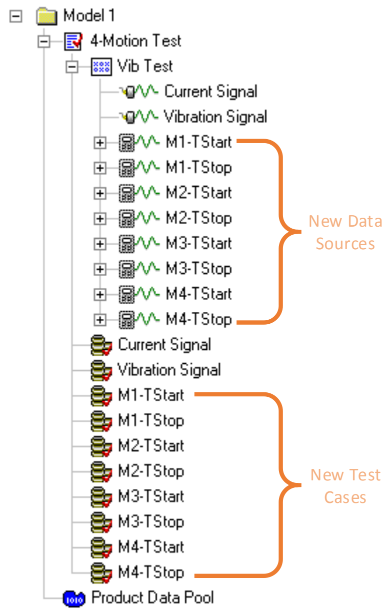
Once you have completed the post process template, it is time to create test cases for each result step defined. In this example, there are 8 result steps; *M1-TStart, M1-TStop, M2-TStart, M2-TStop, M3-TStart, M3-TStop, M4-TStart,* and *M4-TStop*.

To create a test case for each result…

1. Click the *Data Connections* tab.
2. Click the *Create New Test Case* button.
3. When clicked, you will be presented with a dialog containing the names of the data sources and associated test cases that will be created. By default, the data source and test cases names are the same as the result step. In this example, do not change names. In the future, you can change names as follows:
    a. To change the name of a data source, double-click the name to be changed within the *Data Source Name* column.
    b. If you want to change the name of a test case, double-click the name to be changed within the *Test Case Name* column.
4. Click the *OK* button.



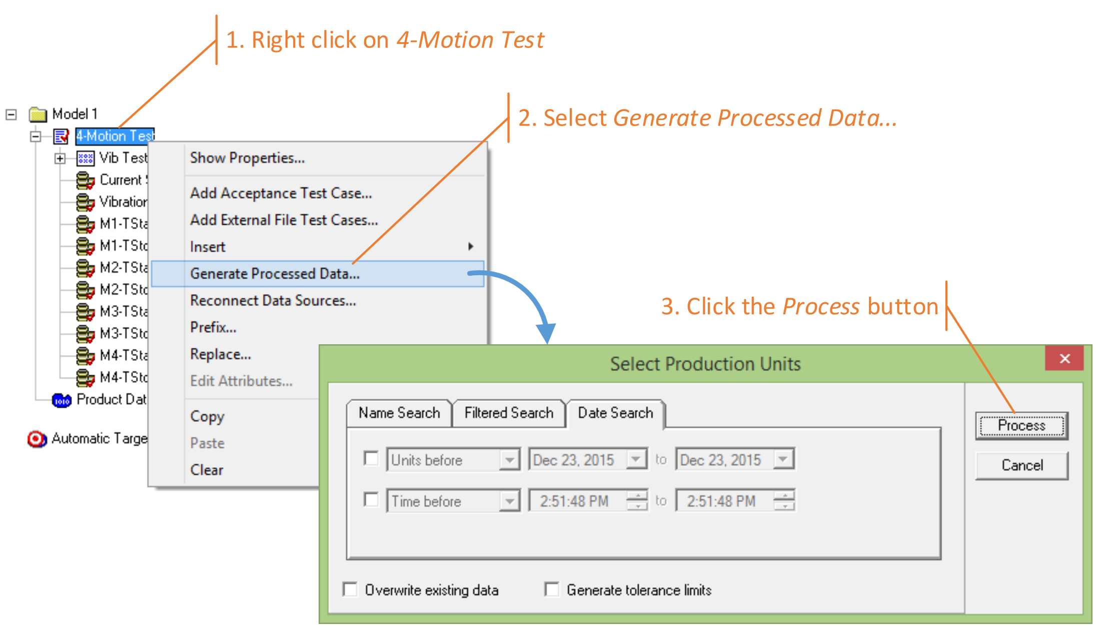
When complete, the *Product Database* will be appended with the newly created test cases and their associated post-process data sources.



At this point, there is no data available within any of the test cases just created. However, it is possible to generate the calculated results for each new test case from all current measurements that exist within the database.

To generate all results of the created test cases at once…

1. From the *Product Database* window, right click on the *4-Motion Test* item.
2. Select the *Generate Processed Data…* option.
3. From the *Date Search* tab with no date restrictions specified, click the *Process* button.

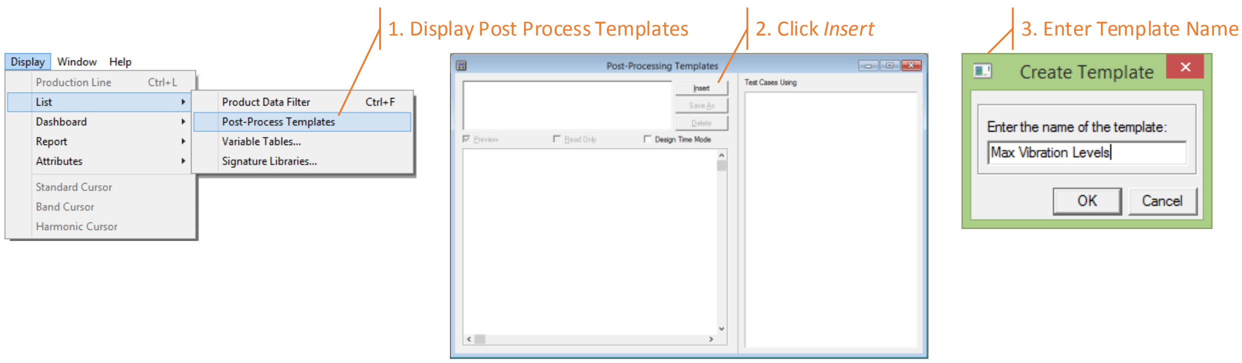


Calculating the Maximum Vibration Level for Each Motion

Now that the start and stop times have been determined, an additional post process template will be developed to calculate the maximum vibration level within each section. The new post process template will use the start and stop times, plus the vibration signal as inputs.

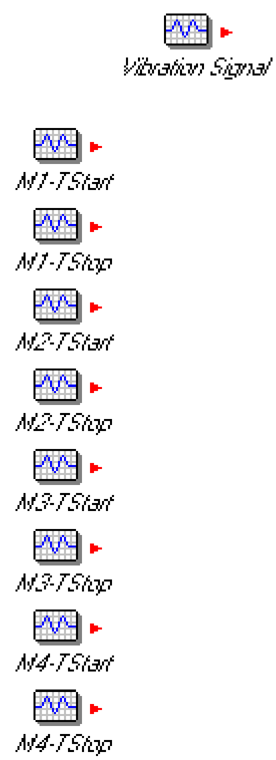
To begin…

1. From the application menu select *Display – List – Post Process Templates*.
2. Click the *Insert* button to create a new post process template.
3. Enter the template name *Max Vibration Levels*, and then click the *OK* button.



Add Data Source Steps…

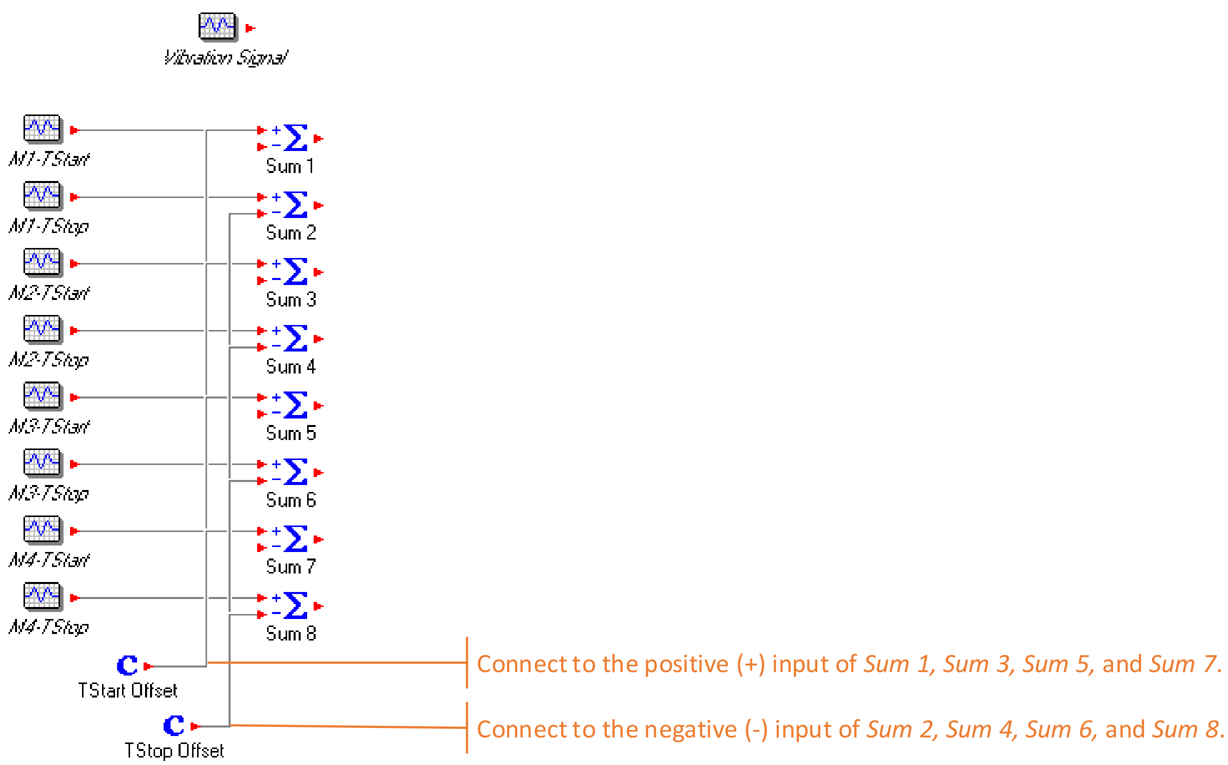
1. From the post process template content region, use the right click *Insert – Data Source* menu option to insert a new data source.
2. Rename the *Input 1* data source step to *Vibration Signal*.
3. Right click and select *Insert – Data Source* again to create a new data source step, and rename this step as *M1-TStart*.
4. Repeat step 3 seven more times, renaming each new data source step according to the motion time test cases created earlier; *M1-TStop, M2-TStart, M2-TStop, M3-TStart, M3-TStop, M4-TStart,* and *M4-TStop*.
5. Arrange the data source steps as shown to the right.



Adding Constant Offsets

The steps to be added in this section are in preparation for trimming the vibration signal based on the trim times determined from the current signal. Often, the very start of a motion and the end of a motion contain transient type vibrations that are not intended to be analyzed as part of the normal steady state operation. The following steps allow specification of an offset to be added to the start time, and an offset to be subtracted from the stop time for each motion.

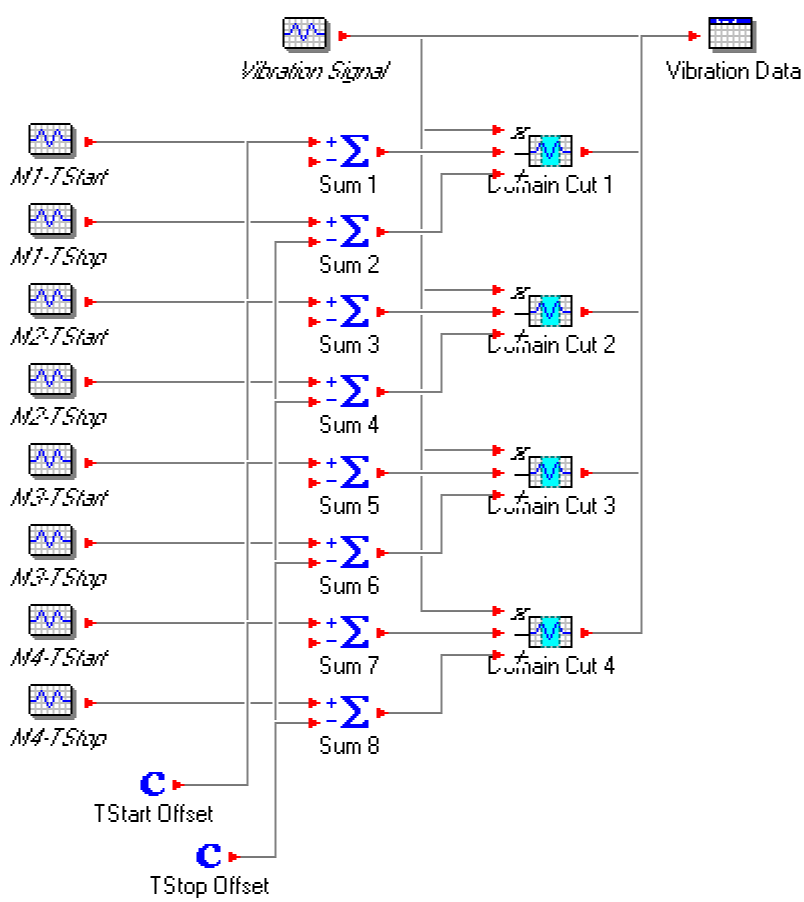
6. Using the *Insert – Generators – Constant* speed menu option, create 2 constant value steps, renaming these as *TStart Offset* and *TStop Offset*. For now, leave the properties unchanged so that each defaults to a constant value of 0.
7. Using the *Insert – Operation – Basic Math – Sum* speed menu option, insert 8 summation steps, aligning them vertically to the right of each of the 8 *TStart* and *TStop* data source inputs.
8. Connect the output of *M1-TStart* to the positive (+) input of the *Sum 1* step, the output of the *M1-TStop* to the positive input of the *Sum 2* step, and so on in order until all of the *TStart* and *TStop* data source steps are connected to the positive input of a summation step.
9. Connect the output of the *TStart Offset* step to the positive input of the *Sum 1*, *Sum 3*, *Sum 5* and *Sum 7* steps. Notice, here we are adding the value of *TStart Offset* to each of the start times for Motion 1, Motion 2, Motion 3 and Motion 4.
10. Connect the output of the *TStop Offset* step to the negative (-) input of the *Sum 2*, *Sum 4*, *Sum 6* and *Sum 8* steps. Notice, here we are subtracting the value of *TStop Offset* from each of the stop times for Motion 1, Motion 2, Motion 3 and Motion 4



Adding Variable Domain Cut Steps

Now that the preferred start and end time for each motion are present, it is time to trim the vibration signal to the calculated times.

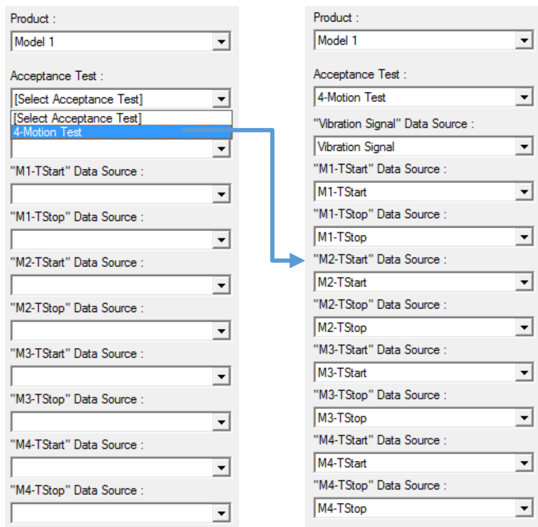
11. Using the *Insert – Operation – Transformations – X Dimensions – Variable Domain Cut* speed menu option, insert 4 *Variable Domain Cut* steps, and align them horizontally to the right of the output of the *Sum 1*, *Sum 3*, *Sum 5*, and *Sum 7* steps.
12. Connect the output of the *Sum 1* step to the negative (-) input of the *Domain Cut 1* step.
13. Connect the output of the *Sum 2* step to the positive (+) input of the *Domain Cut 1* step.
14. Repeat 12 and 13 in a similar manner for the remaining domain cut steps.
15. Connect the output of the *Vibration Signal* data source step to the top (X) input of each variable domain cut step; *Domain Cut 1, Domain Cut 2, Domain Cut 3,* and *Domain Cut 4*.
16. Right click and select *Insert – Design Time Graph – 2 Dimensional* and rename the new graph step as *Vibration Data*.
17. Connect the outputs of *Vibration Signal*, *Domain Cut 1, Domain Cut 2, Domain Cut 3,* and *Domain Cut 4* to the input of the *Vibration Data* graph step.



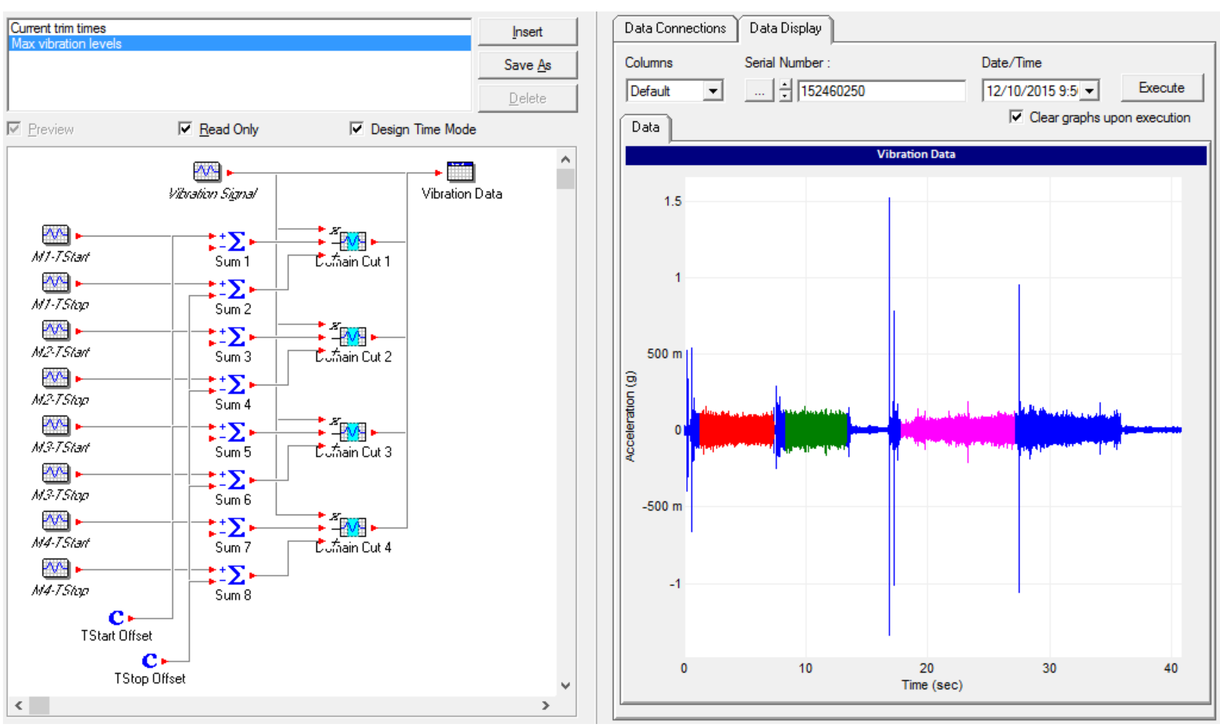
Reviewing the Trimmed Data

As before, enter Design Time Mode by placing a check within the *Design Time Mode* checkbox.

18. From the *Data Connections* tab, select the *4-Motion Test* of *Model 1*. If all of the steps to this point were followed with regard to naming conventions, the data source connections will automatically populate based on the matching data source names and template input step names.



19. Click the *Data Display* tab.
20. Select a production unit, then click the *Execute* button.

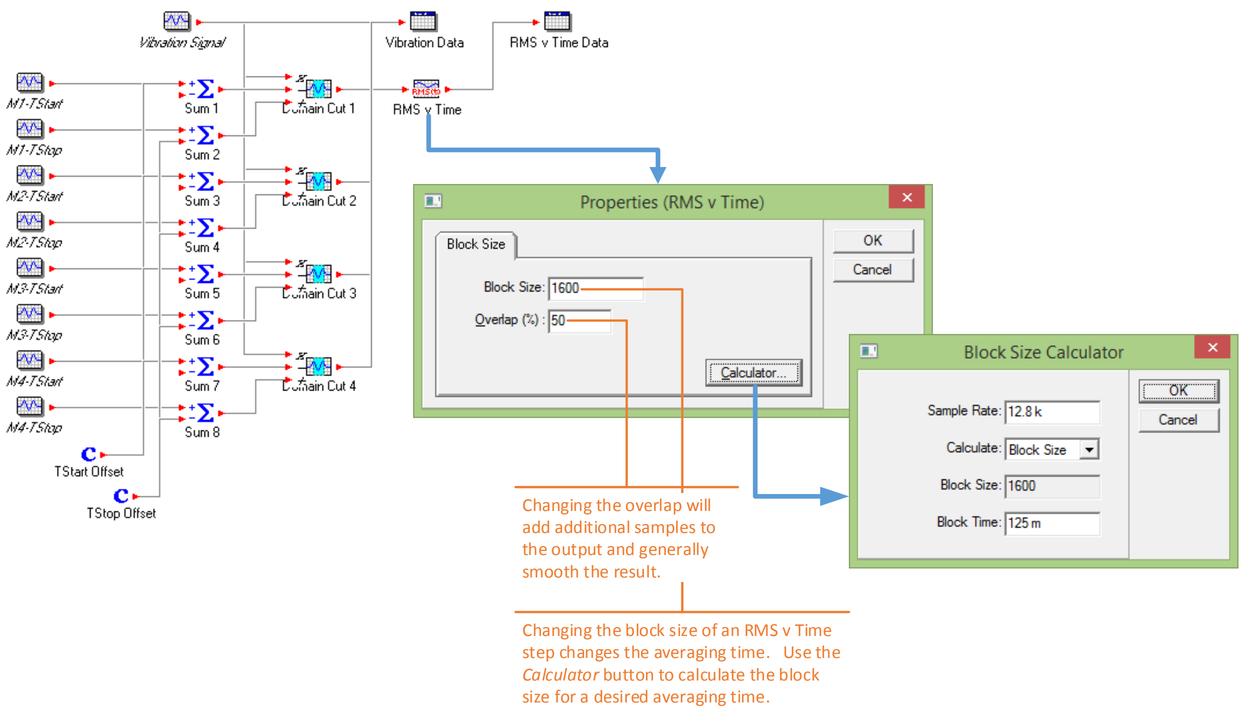


See FAQ's at the end of this document for "How do I get multiple colors in a plot"?

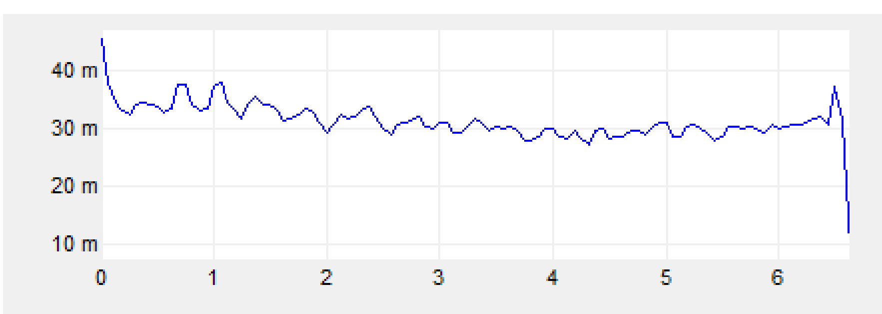
Creating the Maximum Vibration Level Results

Now that the vibration signal is trimmed to each motion, it is time calculate the maximum vibration levels. To keep things simple, an RMS v Time step will be used to calculate the RMS value as each motion progresses. This yields an overall running average estimate of vibration level. From the result of the *RMS v Time*, we will find the maximum value, which will be used as the output of the calculation.

21. Right click and select *Insert – Statistics – RMS v Time* and align the step to the right of the *Domain Cut 1* step.
22. Connect the output of *Domain Cut 1* to the input of the *RMS v Time* step.
23. Right click and select *Insert – Design Time Graph – 2 Dimensional* and rename the graph step to *RMS v Time Data*.
24. Connect the output of the RMS v Time step to the *RMS v Time Data* graph step.
25. Double-click the RMS v Time step and enter the properties as shown.

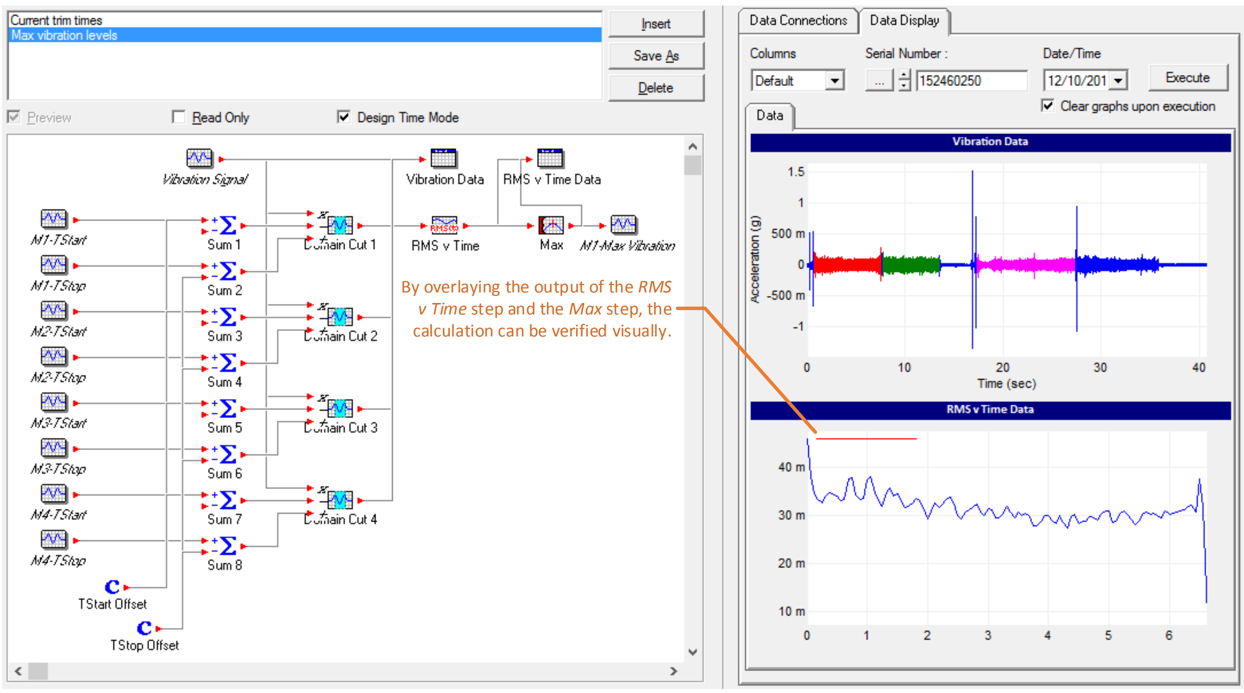


26. Execute the calculation to see the results of the RMS v Time calculation.



To calculate the maximum value of the RMS v Time step…

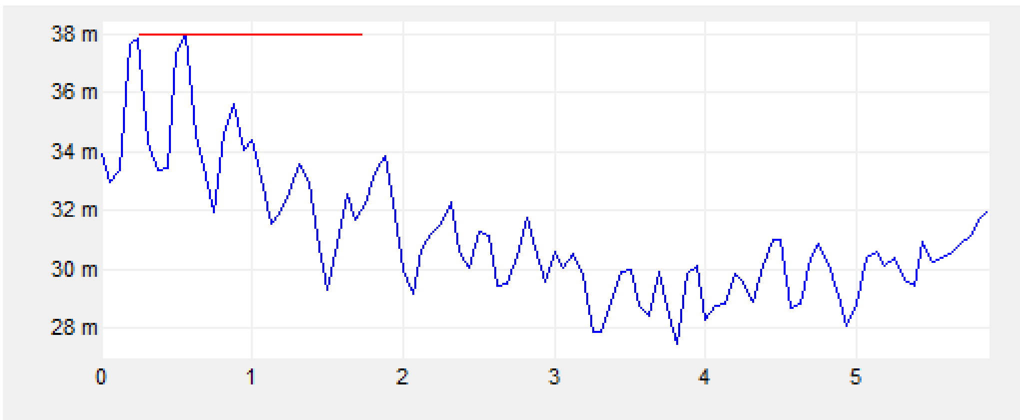
27. Right click and select *Insert – Operation Peak Detection – Maximum*.
28. Connect the output of *RMS v Time* to the input of the *Max* step.
29. Right click and select *Insert – Result*.
30. Rename the result step to *M1-Max Vibration*.
31. Connect the output of the *Max* step to the input of the *M1-Max Vibration* step.
32. Connect the output of the *Max* step to the input of the *RMS v Time Data* graph step.
33. Execute the calculation.



From the RMS v Time Data graph, the maximum level occurs at the beginning of the motion; the start transient. Cycle through several units, executing the calculation for each. Notice that all of them exhibit the largest level from the start transient. Fortunately, by adding the *TStart Offset* and *TStop Offset* constants, the data trimming can be adjusted at the boundaries of each motion.

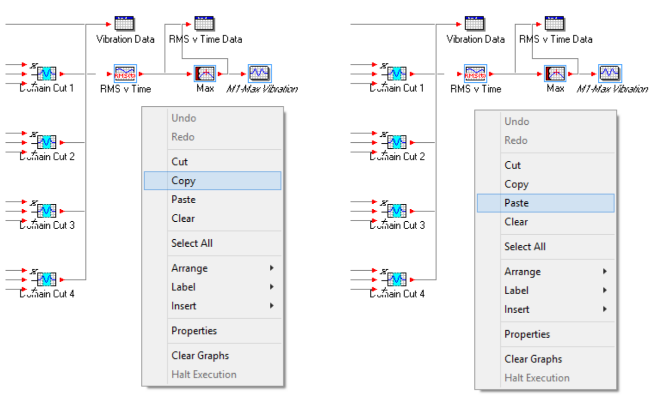
To trim to the steady state portion of the vibration signal…

34. Double-click the *TStart Offset*, then click the *Values* tab.
35. Enter a value of 0.5 within the *Range (Y-axis)* property, then click the OK button. This will trim ½ second from the beginning of the measurement.
36. Double-click the *TStop Offset*, then click the *Values* tab.
37. Enter a value of 0.25 within the *Range (Y-axis)* property, then click the OK button. This will trim ¼ second from the end of the measurement.
38. Execute the calculation.



Duplicate the remaining vibration metric levels…

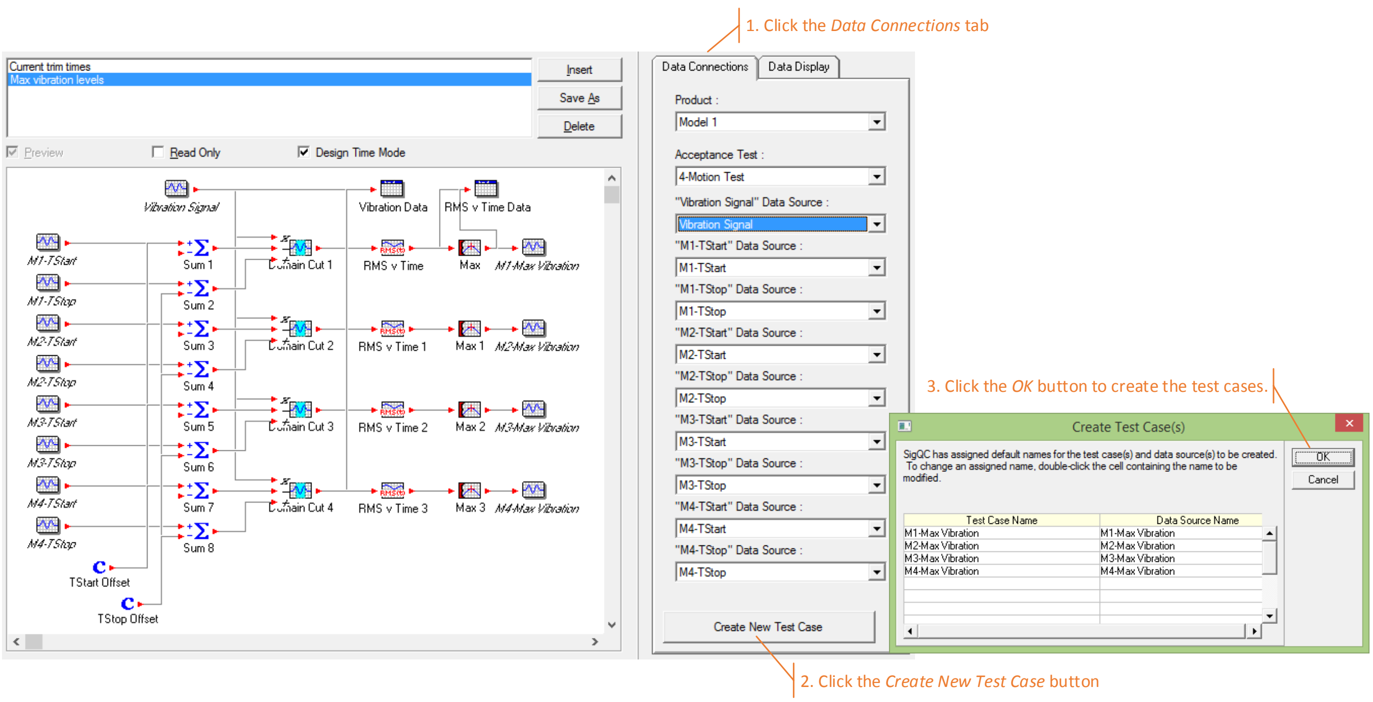
39. Using the left mouse button, click once on the *RMS v Time* step.
40. Holding the *CTRL* key, click once on the *Max* step, then click once on the *M1-Max Vibration* step to extend the selection of calculation steps to include *RMS v Time, Max,* and *M1-Max Vibration*.
41. Right click and select *Copy*, then right click and select *Paste*.



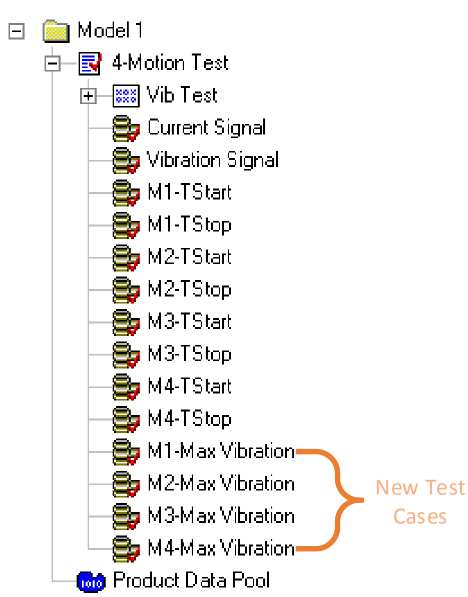
42. Rename the duplicated result steps to *M2-Max Vibration, M3-Max Vibration* and *M4-Max Vibration*.

Create the Maximum Vibration Level Test Cases

1. Click the *Data Connections* tab.
2. Click *Create New Test Cases*.
3. When prompted to accept the test case and data source names, click the *OK* button.



When you click the *OK* button, the new test cases will be created and displayed within the *Product Database* window.



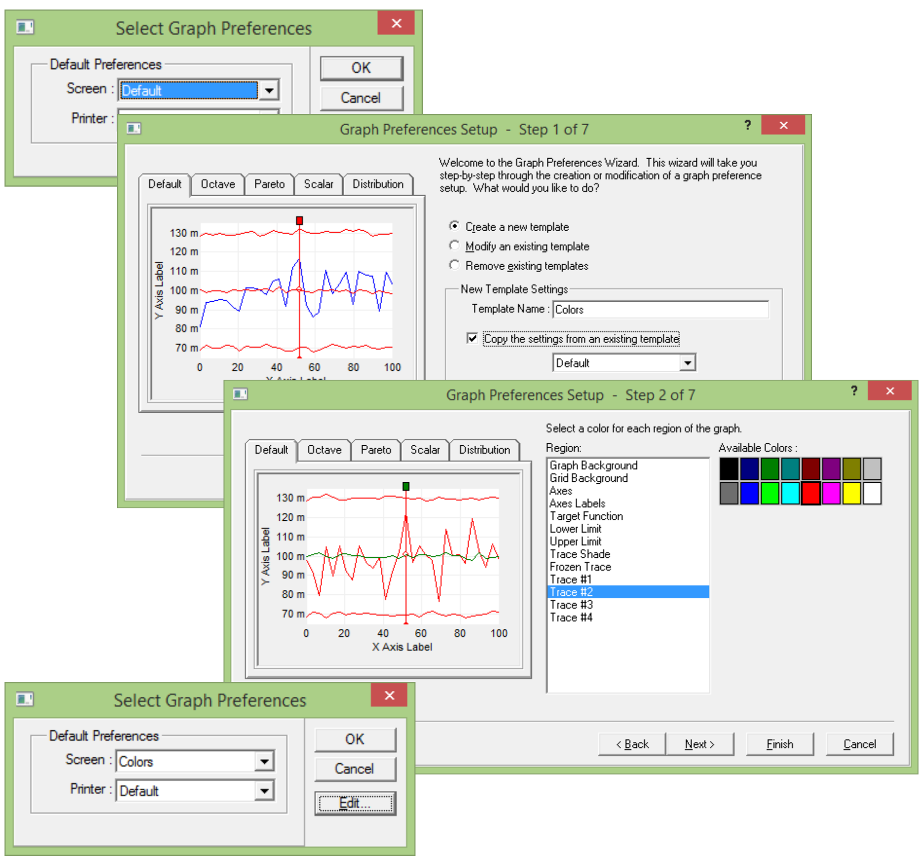
Frequently Asked Questions about Design Time Mode

- Why is the *Execute* button disabled?
    1. One or more input data source steps have not been assigned within the *Data Connections* tab.
    2. The post process template does not contain at least one design time graph step.
    3. A calculation has already been performed. To force a recalculation, either select a different serial number or reselect the measurement time using the *Date/Time* control.
    4. An error was encountered during the previous execution that left the calculation in execution mode. The most common error is the absence of data within the database for the selected serial number and timestamp combination. Right click within the post process template region and select *Halt Execution*.

- Why is the *Create New Test Cases* button disabled?
    1. One or more input data source steps have not been assigned within the *Data Connections* tab.
    2. All of the test cases that could be created from the selected data source combinations within the *Data Connections* tab have already been created. This effectively prevents you from duplicating test cases that already exist.

- Why does my colormap design time graph appear empty?
    1. Colormap graphs are used to display results from calculations that produce a *waterfall* of functions (multiple data sets from one output connection site). If the output of a step produces a 2-dimensional result only, then the colormap graph appears empty.
    2. Sometimes a waterfall of spectrum will contain a high DC content or the low frequency content dominates the spectrum. Meanwhile the x-axis of the colormap extends to high frequencies. In this case, the color scale is based on the large DC level, so most of the data from low frequencies on up appears black. You may see a thin vertical line or two located on the left edge of the colormap. To remedy this, trim the low frequency content from the data using a domain cut step, then plotting it in a colormap.

- How do I get multiple colors to show up on my graphs?
    1. From the application menu select *File – System Preferences – Graphs…*
    2. Click the *Edit…* button to display the graph preferences wizard.
    3. Select *Create a new template.*
    4. Within the *New Template Settings* region, enter a name such as *Colors*.
    5. Check *Copy the settings from an existing template*, then choose the *Default* item beneath the checkbox.
    6. Click *Next*
    7. From the *Region* list, click the item labelled *Trace #1*, then click your preferred color for the first trace of a graph.
    8. Repeat this process for *Trace #2*, *Trace #3* and *Trace #4*, making sure you choose different colors for each trace.
    9. Click the *Finish* button.
    10. The wizard will close, and return to the *Select Graph Preferences* window. From the *Screen* option, choose the newly created preference (*Colors* in this example).



Where can I find a list of all available post process steps?