IMAT⁴XL

Connecting Engineers with Microsoft Excel®

ATA

Overview

IMAT^{4}XL™ provides an interface between test and simulation data and Microsoft Excel® through a graphical user interface that also provides plotting and math functions for engineering analysis beyond standard Excel capabilities. IMAT^{4}XL is a convenient, standalone GUI that leverages ATA Engineering's IMAT™ family of MATLAB® toolboxes. It allows users to harness the engineering analysis power of IMAT without requiring proficiency in MATLAB, while facilitating custom post processing in Excel.

Features

- Import function data, mode shapes, analysis results, and matrices from widely used file formats.
- Bidirectional transfer with Microsoft Excel for custom post-processing, plotting, and documentation.
- Create new functions or modify existing functions by applying powerful math operations to imported functions.
- Export loaded functions, shapes, analysis results, and matrices in a number of common analysis formats for use in other computer-aided engineering (CAE) software.
- Plot function data directly to Microsoft Excel, or preformat plot using intuitive interface for export to Excel.
- Directly modify analysis data without any programming syntax.
- Supports Microsoft Excel 2003, 2007, 2010, 2013, 2016, and 2019.

IMAT^{4}XL Revision History

The following lists provide a history of changes to IMAT^{4}XL.

v8.1.0

New Features/Enhancements

- Updated to use IMAT v8.1.0

v8.0.0

New Features/Enhancements

- Updated to use IMAT v8.0.0

v7.9.0

New Features/Enhancements

- Updated to use IMAT v7.9.0

v7.8.0

New Features/Enhancements

- Updated to use IMAT v7.8.0
- Link plot series names to the corresponding data item cell rather than hard-coding
- Move abscissa column outside the first function for condensed data items so that all of the data item columns are the same width

Bug Fixes

- Minor documentation and bug fixes
- Fix TIER#3855 and 3856

v7.7.0

New Features/Enhancements

- Updated to use IMAT v7.7.0

v7.6.0

New Features/Enhancements

- Updated to use IMAT v7.6.0

v7.5.0

New Features/Enhancements

- Updated to use IMAT v7.5.0

v7.4.0

New Features/Enhancements

- Updated to use IMAT v7.4.0

v7.3.0

New Features/Enhancements

- Updated to use IMAT v7.3.0

v7.2.0

New Features/Enhancements

- Updated to use IMAT v7.2.0

v7.1.0

New Features/Enhancements

- Updated to use IMAT v7.1.0
- Templates now have option to create multiple functions, shapes, or results
- Templates have been enhanced to be more flexible. Please see the template instructions for more details.
- Improved attribute reading abilities
- Added a Math History button to retrieve previously used math expressions
- The math parser now understands more Matlab syntax

Bug Fixes

- Minor documentation and bug fixes

v7.0.0

New Features/Enhancements

- Updated to use IMAT v7.0.0
- Overhaul internal workings to use the Excel .Net interface rather than COM
- Add "+" button to the upper right corner of the Data Region allowing you to customize the attribute columns displayed
- Add a Statistics capability that calculates the statistics of selected functions and displays them in a window
- Users now have the ability to overwrite records in IMAT^{4}XL when adding data
- Add current directory readout to menu bar
- Improved matrix handling capabilities

Bug Fixes

- Fix TIER# 2614, 2615, 2617, 2618 and 2625
- Minor documentation and bug fixes

v6.3.0

New Features/Enhancements

- Updated to use IMAT v6.3.0.
- Updated to support Microsoft Excel 2016

Bug Fixes

- Minor documentation and bug fixes

v6.1.0

New Features/Enhancements

- Updated to use IMAT v6.1.0.
- Add support for imat_fn/diag

Bug Fixes

- Minor documentation and bug fixes
- Fix imat_shp to imat_result conversion method call
- Fix retrieve imat_result

v1.2.0

New Features/Enhancements

- Updated to use IMAT v4.5.1.
- Add support for importing MATLAB MAT files
- Add Units as an optional attribute to transfer to/from Excel

Bug Fixes

- Minor documentation and bug fixes
- Fix TIER reports 1036, 1087

v1.0.0

New Features/Enhancements

- Initial release.

User Guide

Introduction

IMAT^{4}XL is designed for engineers who work with test or simulation data. It imports ASCII and binary engineering data into an easy-to-use interface which allows for complex matrix math and data manipulation combined with Microsoft Excel for visualization. IMAT^{4}XL organizes data in an intuitive format and allows users to plot functions with an engineering plotting tool, not the built-in Excel tool. Data sent to Excel is formatted efficiently for further calculations, and plots are automatically recreated, saving the time and hassle of replotting in Excel. Loaded data is stored in memory, protecting the original file from accidental modification.

IMAT4XL is built on ATA Engineering's IMAT product, a family of software tools built for MATLAB. It offers a convenient and intuitive interface for many of IMAT's powerful features, without the need for MATLAB. Some of the basic IMAT features encapsulated in IMAT^{4}XL are described here.

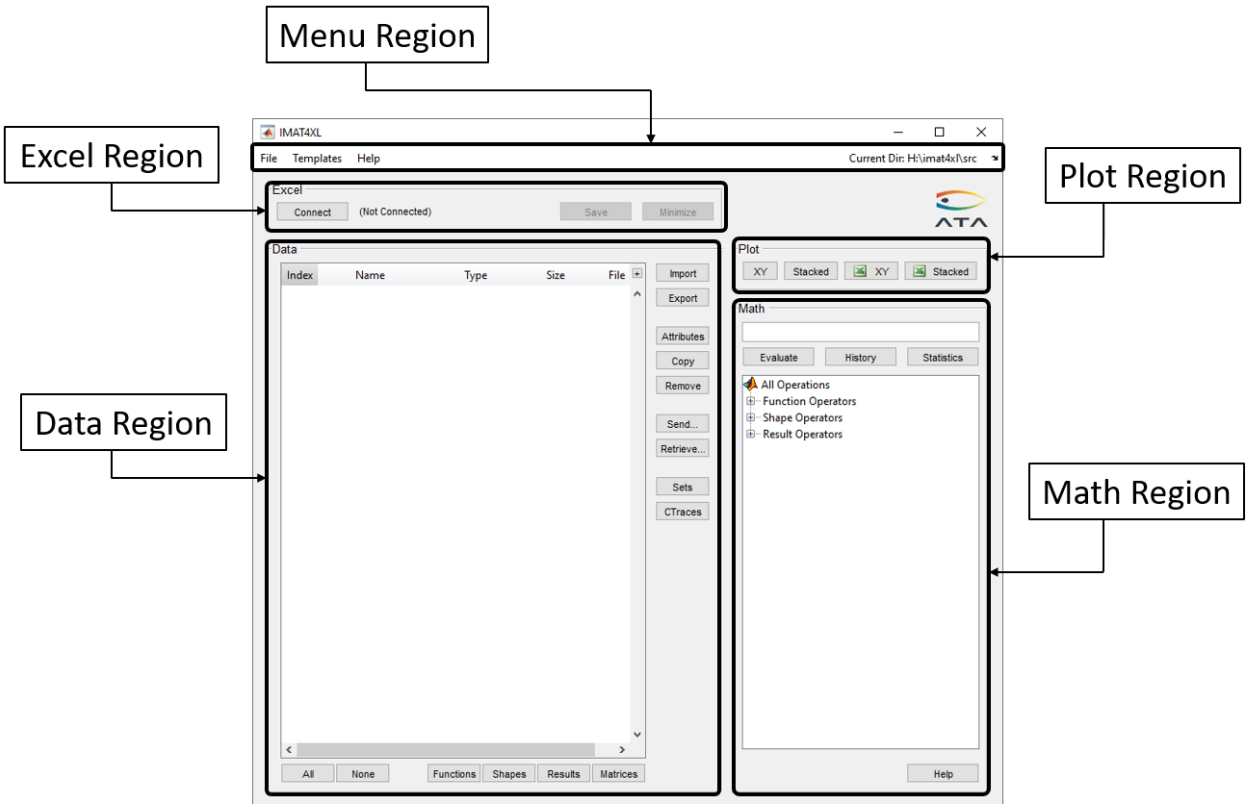
IMAT stores different types of information in four basic formats called classes: functions, shapes, results, and matrices. The function data class stores X-Y functions. The shapes class is used to store nodal results such as mode shapes or static deformations. The results class is a more generic class that stores various types of results such as force and stress. The matrix class is a structure that stores information about the matrix. Individual data entities are called objects.

Each class has a series of attributes that stores descriptive attributes about the data. For function data, this includes information such as the function type (FRF, PSD, etc.), the abscissa and ordinate labels, and a description of the data. Understanding these values can help to fully harness the power of IMAT^{4}XL.

For more information on IMAT, as well as the attributes of each class, see the full IMAT documentation. This is available by selecting *IMAT Help* from the help menu at the top of the main IMAT^{4}XL window.

Main Form

The main window for IMAT^{4}XL is divided into five regions: The Menu region, the Excel region, the Plot region, the Math region, and the Data region. Each region has a distinct purpose which will be thoroughly explained in the following sections.



Menu Region

The Menu region contains a number of convenient shortcuts to control IMAT^{4}Excel. They are divided into three menus: *File, Templates*, and *Help*.

The *File* menu contains four items: *Import*, *Change Working Directory*, *Change Current Units*,and *Quit*. The *Import* menu works the same as the **Import** button – it imports data into IMAT^{4}XL. The *Change Working Directory* menu changes the current directory. All file dialogs that are opened will default to this directory. By default, the current directory is the directory in which the IMAT^{4}XL application. The current directory is also displayed on the far right side of the menu region. The current directory can also be changed by clicking there. The *Change Current Units* shows the abbreviation of the current units set in parentheses in the menu. This brings up the Select Unit System window - it allows the user to select a predefined units system or define a user defined units system.

The *Templates* menu contains links to each of the Excel Templates: *Functions*, *Shapes*, and *Results*.

The *Help* menu provides the items: *IMAT Help*, *IMAT^{4}XL Help*, and *About*. *IMAT Help* shows the help for all of IMAT. This is useful for researching attributes for specific objects or for learning more about other features of ATA Engineering's IMAT software product. *IMAT^{4}Excel Help* shows the help for IMAT^{4}XL. The *About* menu item shows version information.

Excel Region

The Excel Region contains three buttons that control the connection to Excel.

The **Connect** button initiates the connection to Excel. After pressing the button, a dialog will appear with the option to connect to a new or existing workbook. After connecting, the name of the connected

workbook will appear to the right of the button. If a new workbook was created, this text will indicate that the workbook was created but hasn't been saved.

To save the workbook, press the **Save** button in the Excel region.

When the workbook is first connected, it will be shown on the Desktop. IMAT^{4}XL provides the option to hide the workbook from view, but the workbook will remain connected. The **Minimize** button will hide the workbook; pressing the **Restore** button will make the workbook visible again.

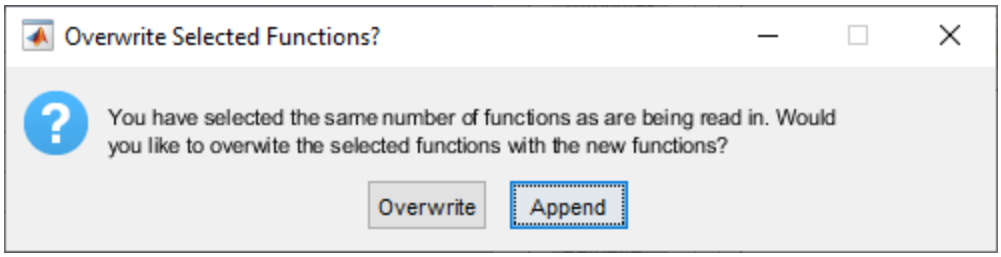
It is important to note that Excel has some limitations that can affect its ability to work with engineering data. In Excel 2003 and prior, there is a strict limit on the number of columns and rows that can be in a single worksheet. This limit is 256 columns and 65,536 rows. If data larger than this is sent to Excel, an error will appear stating that the data cannot be written. To avoid this error, reduce the size of your data set by using sets to select a subset of results, decimate functions with a high resolution, or use Excel 2007 or later where the limits were increased to 16,384 columns and 1,048,576 rows.

Data Region

The Data region displays the data that has been loaded into IMAT^{4}XL. This includes data read from a local analysis file, data read in from Excel, and data created by math operations. To track where the data originated, data read from Excel is colored green, and data created by a math operation is colored blue.

Any data that is too large to export to Excel will be colored red. This means that the data has too many rows or columns to export to the connected Excel. The rows will only be colored if an Excel application is connected. The number of allowable rows and columns is dependent on the version of Excel.

Whenever any data is loaded , IMAT^{4}XL will run checks to see if the incoming data is suitable to replace the currently selected data. This occurs when the same number of records are selected as are incoming, and when the data classes match. For example, if five functions are selected, the following dialog below will appear if five functions are being read in or created.



It is possible to overwrite data when the proper number and type of data rows are selected prior to clicking **Import**, **Retrieve**, or **Evaluate**.

The Data region table contains a single row for each data item that has been loaded and several columns describing that data. The five default columns are labeled "Index," "Name," "Type," "Size," and "File." The "Index" column contains the nomenclature for referencing that data item during math operations. The "Name" column contains the name of the data item. This corresponds to the *IDLine1* attribute for shapes and the *Name* attribute for function, result and matrix data types.

The "Type" column describes the type of the data. This will be the type of function, shape, or result that was loaded. For functions, this is the *FunctionType* attribute. For shape data, it is the *OrdNumDataType* attribute. For results, it is the *ResultType* attribute. For matrix results, it will simply be "matrix." The "Size" column lists the number of rows and columns that data would take up if exported to Excel.

The "File" column contains the location the particular piece of data came from. If it was loaded from a file, the filename will be shown. If it was retrieved from an Excel file, the column will contain the name of the Excel file and worksheet that the data came from. If it was created by a math operation, the column will contain the name of the math operation that created the piece of data.

Several other column options can be accessed via the **+** button in the upper right corner of the table. Pressing this opens a dialog that allows for changing what columns are visible, and in what order. The available columns are a subset of the attributes of functions, results, shapes and matrices. If a data row doesn't have a value for a given attribute, the cell will be blank.

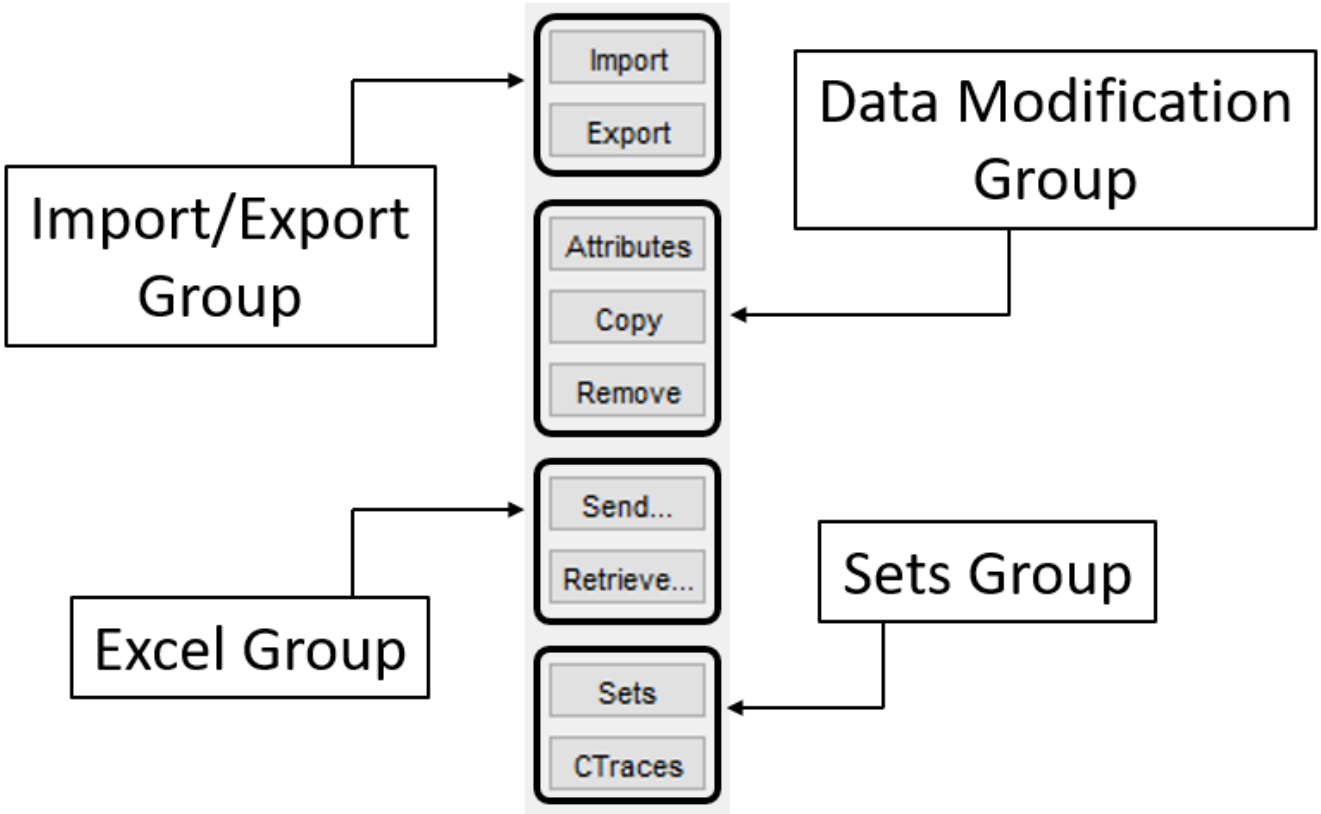
The data region also contains a number of important buttons that control the operation of IMAT^{4}XL. These are divided into the Selection buttons and the Action buttons.

Selection Buttons

The Selection buttons are located below the data table. The **All** button will select all data items in the table. The **None** button will deselect all items. The **Functions** button will select all functions in the data table. The **Shapes** button will select all shapes in the data table. The **Results** button will select all result objects in the table. The **Matrices** button will select all matrix objects in the table.

Action Buttons

The Action buttons are grouped into four groups – "Import/Export," "Data Modification," "Excel," and "Sets." The buttons are located to the right of the Data Table and are labeled below.



Action buttons control many of the functions of IMAT^{4}XL.

Import / Export Group

The "Import/Export" group contains the **Import** and **Export** buttons. The **Import** button is used to load data into IMAT^{4}XL. Once the button is pressed, a dialog box will appear where analysis files can be selected. The types of analysis files that can be imported are:

- I-deas Associated Data Files (ADFs) – These files contain function or shape data. Function ADFs have the extension *.ati* or *.afu*, and shape ADFs have the extension *.ash*.
- I-deas/NX Universal Files – These files can contain a wide variety of data, including functions, shapes, and results.
- Nastran Output2 (OP2) Files – These files can contain a wide variety of data that was created during a Nastran analysis. This includes, but is not limited to, shapes, results, and matrices.
- Nastran PCH Files – These files can contain a wide variety of data that was created during a Nastran analysis. This includes, but is not limited to, shapes, results, and matrices. IMAT^{4}XL can only import function data that is stored in these files.
- Nastran DMIG Files - These files can contain a wide variety of data used during a Nastran analysis, but generally contain matrix information.
- Abaqus ODB Files – These files contain data created during an ABAQUS analysis. This includes xyData and step histories, which are both functions, and step field data, which can be shapes or results.
- Matlab MAT Files – These files contain IMAT variables such as `imat_fn`, `imat_shp`, and `imat_result`. Unsupported variable types are ignored.
- Vibrata Function Files - These files contain function data used by Vibrata, stored in either *.fcn* or *.vra_xyout* files.

The **Export** button is used to send manipulated analysis data back to a format usable by other CAE tools. The available export formats are:

- I-deas ADF (ATI, AFU, and ASH)
- I-deas/NX Universal. (See Note)
- Femap Neutral
- Nastran DMIG
- Vibrata FCN

When exporting, not all IMAT^{4}XL properties can be saved to all file formats. ADF (ATI, AFU, and ASH), Universal, and Neutral files will not save the *Name*, *Filename*, *UID*, *Parent* or *Children* attributes of functions. *RSpectQ* and *InterpolationType* are stored in the *UserValue3* and *UserValue4* attributes, respectively. This will only affect youif the *Name* field was changed in the Data Table, or if the functions were imported from *.fcn* or *.vra_xyout* files and are exported to one of the above-mentioned formats.

Important Note About Exporting Universal Files

It is important to note that NX requires that the associated geometry be included in the Universal file along with any results. Because of this, whenever a Universal file is exported, IMAT^{4}XL will ask if you want to append geometry to the exported file. If so, there are two options: Select the geometry from an existing Universal or Nastran OP2/BLK file, or select from a list of FEMs that have already been imported.

Attribute Modification Group

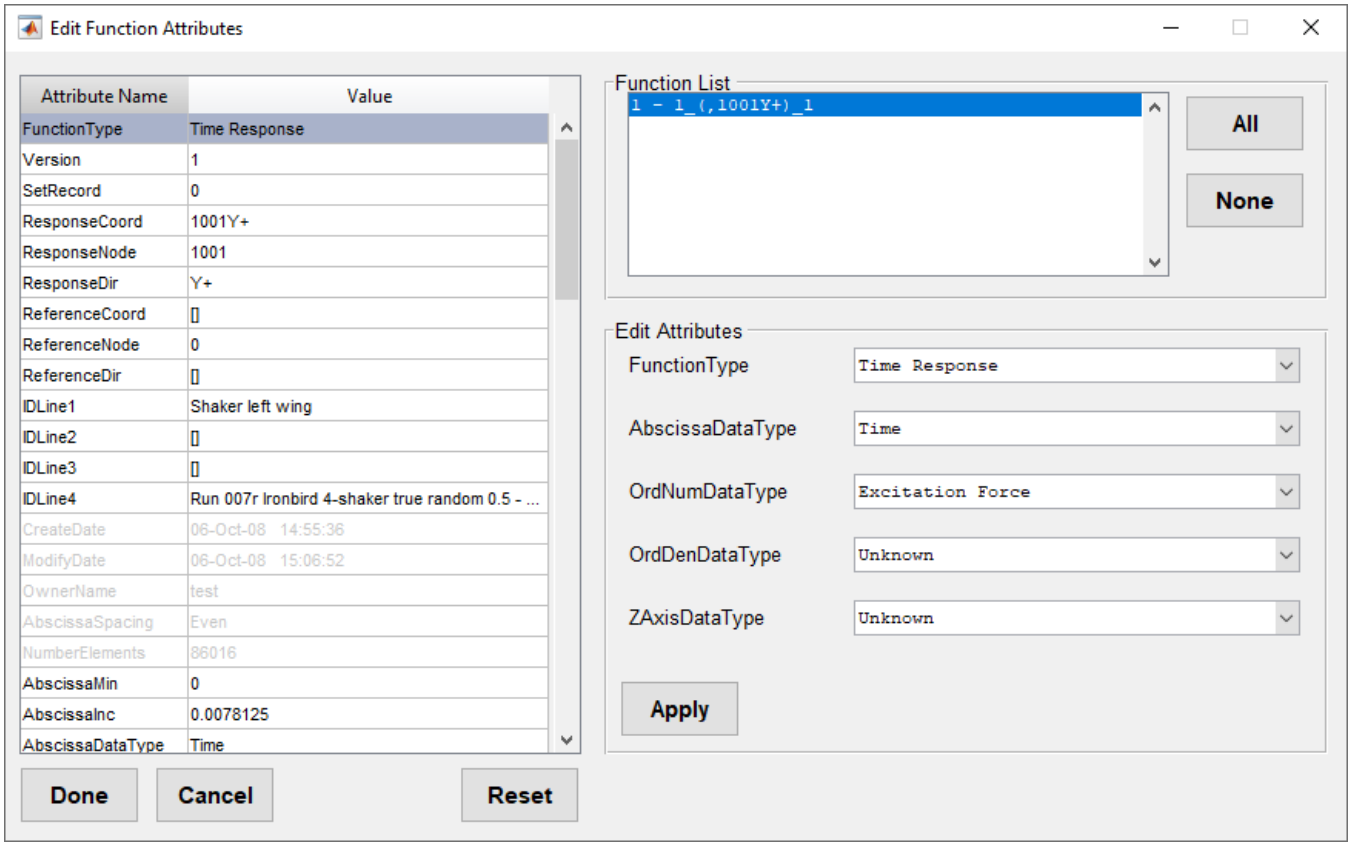
The Attribute Modification group is used to change the attributes of the data that has been loaded into IMAT^{4}XL. This group also includes the ability to remove loaded data from memory.

Data can be removed from memory by selecting it in the table and pressing the **Remove** button.

Data attributes can be modified by selecting a data item in the table and pressing the **Attributes** button. For functions, shapes, and results this will bring up the Edit Attributes form, shown below. This form can be used to change the descriptive attributes of the item. The Known Attributes table on the left side of the window contains all of the attributes of the selected object and their current value. Any attribute can be modified unless it is grayed out. To edit an attribute, select it from the Known Attributes table. When an item is selected from this table, that attribute, either by itself or with a group of related attributes, is automatically loaded into the Edit Attributes panel where it can be modified. To save the changes, press the **Apply** button.

The Object List panel at the top right of the panel contains all of the data objects that were selected when the **Attributes** button was pressed. By default, all of the objects are selected; they can be deselected using the **None** button. Any attribute change made in the Edit Attributes panel will only apply to the objects that are highlighted in this panel.

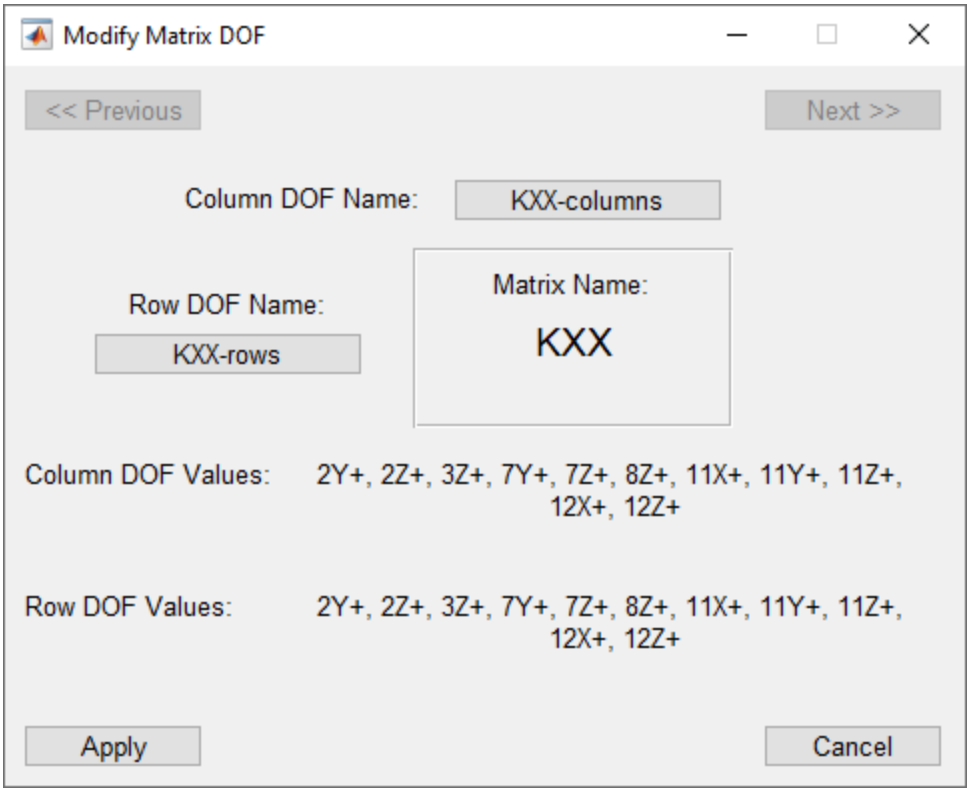
It is important to use the Edit Attributes form when retrieving data from Excel because default values will be assigned for any attributes that are not specified in Excel.



The Edit Attributes form is used to change the attributes of imported data.

Pressing the **Attributes** button with matrices selected will not open the Edit Attributes form, as matrices don't have many attributes that can be changed. Pressing **Attributes** will instead allow you to change the *Coordinate Traces* for the matrix through a special matrix modification form. The buttons above and to the left the *Matrix Name* box will open the Coordinate Trace dialog to select new sets of degrees of

freedom to assign to the matrices rows and columns. The text in the buttons will update to the name of the Coordinate Trace selected, while the values of the trace will be shown in the lower half of the window. The **<< Previous** and **Next >>** buttons allow you to change which of the selected matrices are being modified. **Apply** will save all the changes that have been made since the window was opened and **Cancel** will discard all the changes.

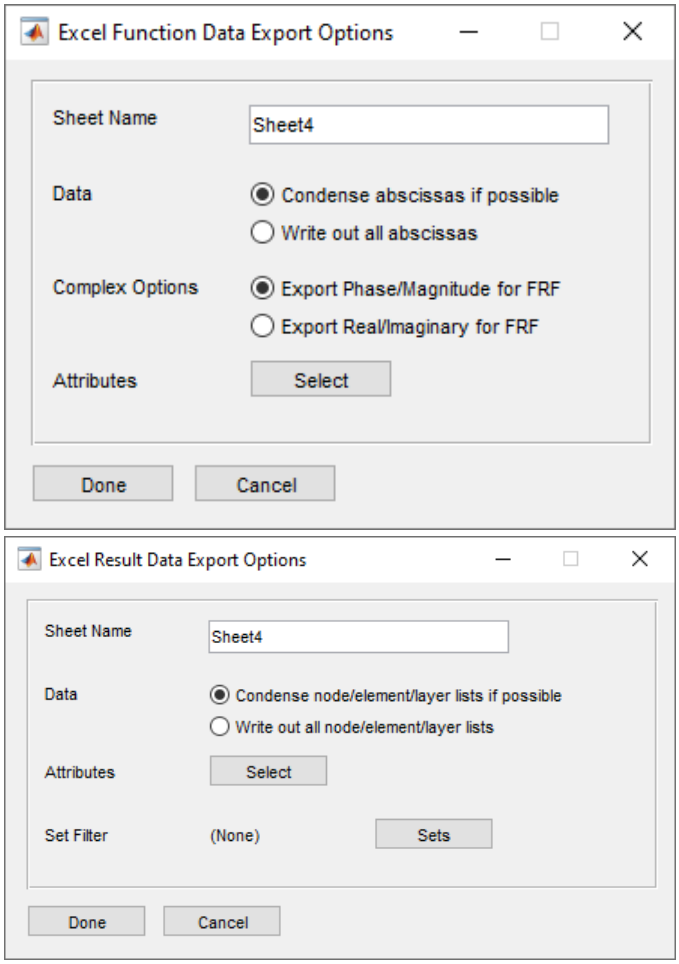


There is a special window to allow for modification of matrices *Coordinate Traces*

Excel Group

The Excel group is used to send data to Excel and retrieve data from Excel. Sending data to Excel is an instrumental part of IMAT^{4}XL, especially for shape and result data, which is traditionally more difficult to load and operate on in Excel. These functions allow convenient viewing and post processing of analysis data in Excel.

To send data to Excel, select the desired data from the table in the Data Region and press the **Send** button. This will open a form that helps set options for the export. The form will have slightly different options based on the data being sent to Excel. Two examples, which encompass all of the possible options, are shown and explained below.



The Options Forms set parameters controlling the data export.

The "Sheet Name" edit box selects the sheet to send the exported data to. By default, this is the next available sheet name in the connected workbook. Any name can be entered so long as it does not conflict with the name of an existing sheet.

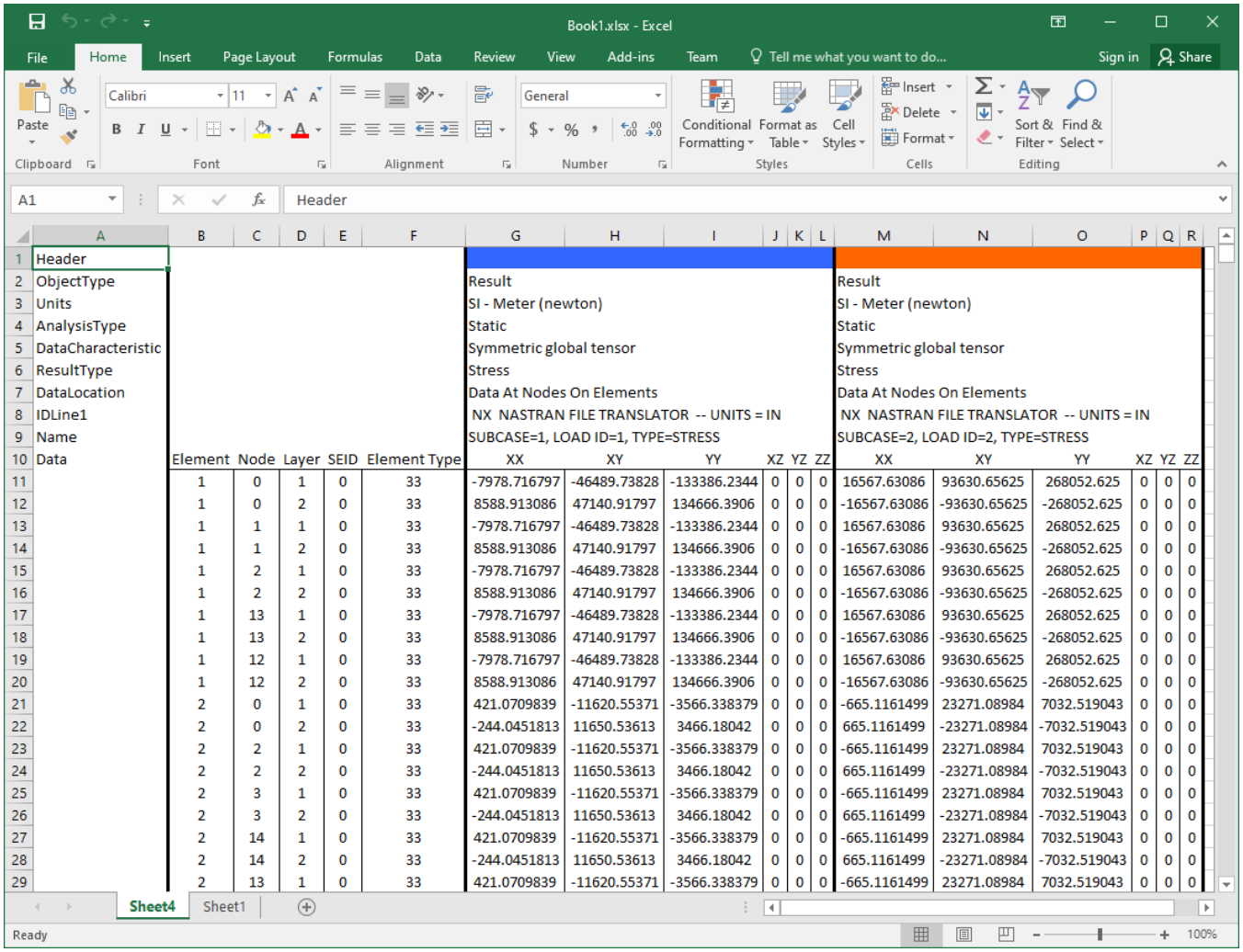
The "Data" radio buttons select whether the "Abscissa" data should only be written out once. This is the case when function data has the same abscissa or multiple shapes reference the same nodes, etc. If data does not all have the same abscissa (or node list), all abscissas will be written regardless of how this option is set.

The "Complex Options" radio buttons select whether complex data should be written out as Phase/Magnitude or Real/Imaginary. When plotting, it defaults to Phase/Magnitude unless it was set as Real/Imaginary in UIPLOT. This option only appears when exporting function data.

The **Attributes** button provides a convenient way of selecting which attributes to send to Excel, along with data. The recommended attributes are automatically selected, though additional attributes can be added or removed. The selected attributes will be remembered for future exports.

The **Set Filter** button will bring up the Sets form mentioned earlier. It functions in the same way, but in this case is used to select the set to use for filtering. To select the desired set for filtering, simply select it from the Sets Form and press **Done**. Only one set can be selected at a time. If no filtering is desired, press **None** on the Sets form, and then press **Done**. The set used for filtering will appear to the right of the "Set Filter" text on the options form. If no set will be used, it will display "(None)."

Once the desired options are set, press **Done** on the Options form. This will create a new worksheet in the connected Excel workbook and send the data to it in an easy-to-understand format. An example of the resulting data from sending stress results is shown below.

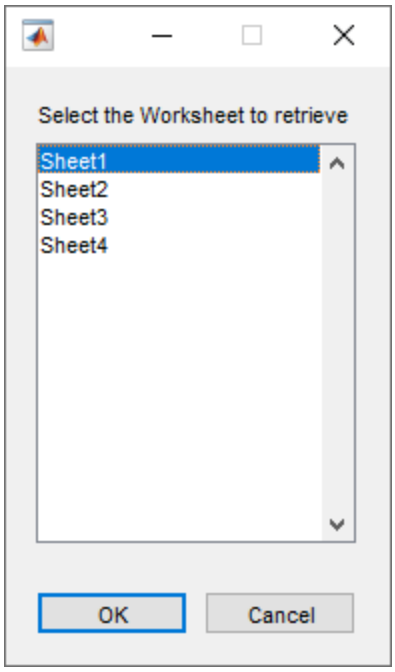


	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R
1	Header																	
2	ObjectType						Result						Result					
3	Units						SI - Meter (newton)						SI - Meter (newton)					
4	AnalysisType						Static						Static					
5	DataCharacteristic						Symmetric global tensor						Symmetric global tensor					
6	ResultType						Stress						Stress					
7	DataLocation						Data At Nodes On Elements						Data At Nodes On Elements					
8	IDLine1						NX NASTRAN FILE TRANSLATOR -- UNITS = IN						NX NASTRAN FILE TRANSLATOR -- UNITS = IN					
9	Name						SUBCASE=1, LOAD ID=1, TYPE=STRESS						SUBCASE=2, LOAD ID=2, TYPE=STRESS					
10	Data	Element	Node	Layer	SEID	Element Type	XX	XY	YY	XZ	YZ	ZZ	XX	XY	YY	XZ	YZ	ZZ
11		1	0	1	0	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0
12		1	0	2	0	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0
13		1	1	1	0	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0
14		1	1	2	0	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0
15		1	2	1	0	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0
16		1	2	2	0	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0
17		1	13	1	0	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0
18		1	13	2	0	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0
19		1	12	1	0	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0
20		1	12	2	0	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0
21		2	0	1	0	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0
22		2	0	2	0	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0
23		2	2	1	0	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0
24		2	2	2	0	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0
25		2	3	1	0	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0
26		2	3	2	0	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0
27		2	14	1	0	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0
28		2	14	2	0	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0
29		2	13	1	0	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0

Stress results sent to Excel are easy to post-process.

The **Retrieve** button has the purpose of importing data that is stored in the connected Excel. This can either be done by retrieving previously sent function, shape, or result data, or by manually forming function data by selecting abscissa and ordinate ranges. When the **Retrieve** button is pressed, a dialog will appear allowing the user to choose the retrieval method.

If previously "sent" data from Excel needs to be re-imported in its entirety, the form below will be shown. Select the worksheet that contains the data to re-imported and press **OK**. More information on the required format is discussed later.



Available sheets from the connected workbook are shown for importing data.

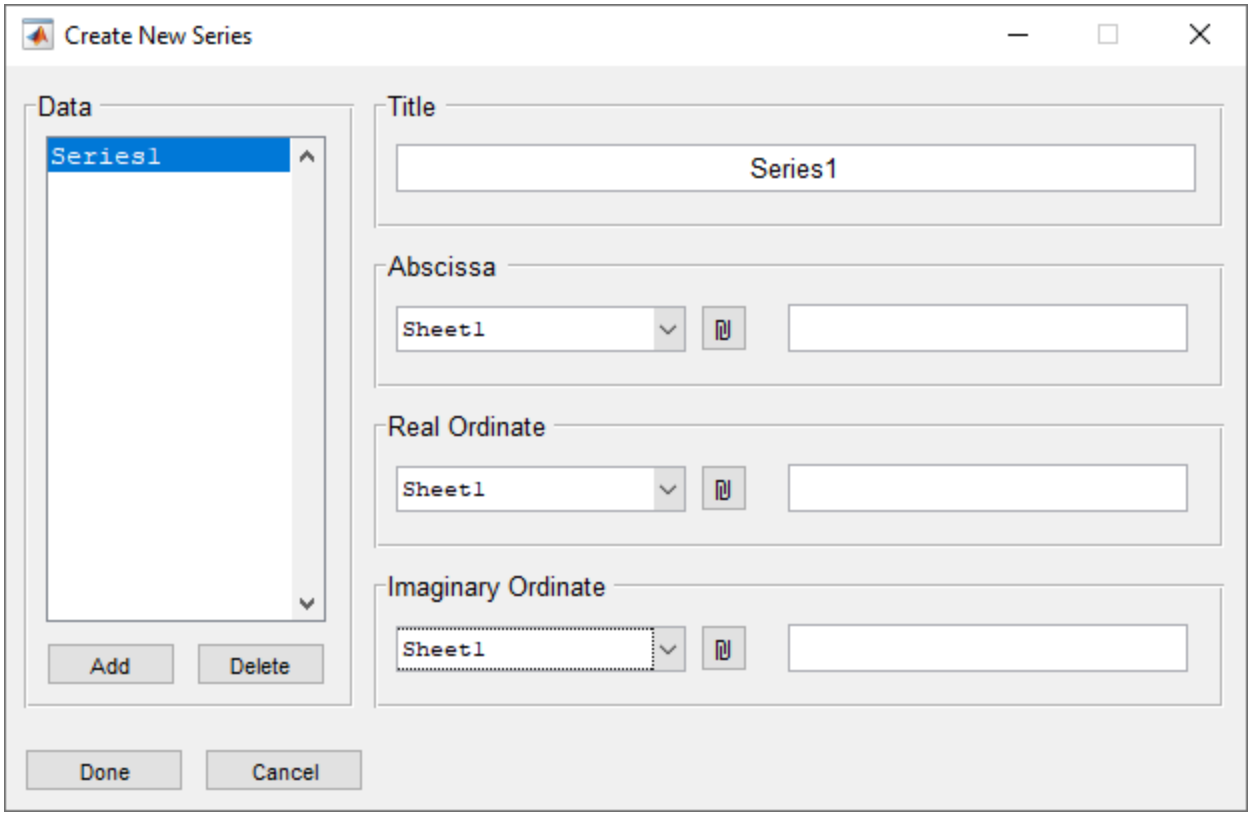
The second method of retrieving data from Excel is to choose the data by columns using the form shown below. This form allows the user to create functions for use in IMAT^{4}XL by creating a series and then selecting the abscissa and ordinate data belonging to each series – much like creating plots in Excel 2003.

To create a new series, press the **Add** button. This will add a series to the listbox and initialize the function data portion of the table. A default series name will be placed in the "Title" panel. To change it, enter a new value in the edit box. To set the Abscissa, Real Ordinate, or Imaginary Ordinate, first select the desired cells with the data from the connected Excel workbook. Make sure the worksheet with your selection on it is shown in the pulldown box (shown as "Sheet1" in the figure below). Press the **Refresh** button, located to the right of the pulldown, to send the selection to the edit box on the right. Alternatively, the cell range in Excel nomenclature can be typed directly into the edit box.

It is important to note that abscissa and ordinate data must be a contiguous column selection. The number of rows must be the same for each as well. If complex data is desired for the ordinate, place the real portion in the "Real Ordinate" panel and the imaginary portion in the "Imaginary Ordinate" panel.

If a created series is no longer needed, select it from the series panel and then press the **Delete** button. The selected series will be removed from the list.

Once all of the desired series have been created, press the **Done** button, and the corresponding function objects will be created and added to the Data Table in the main IMAT^{4}XL window.



New functions can be created by selecting columns of data from a worksheet.

Sets Group

This group contains the **Sets** and **CTraces** buttons. These buttons launch convenient windows for creating, manipulating, and exporting sets and coordinate traces. These items are used to pare down analysis data, so that only the desired data from a large data object is exported. This is convenient when the entire data object is too large to view in Excel.

Sets

To create a new set, press the **New** button. This will launch the New Set form shown below. This form has three edit boxes which help define the set. The first edit box contains the ID of the set. This is a numeric ID assigned to the set. This ID is used if the set is exported for use in Nastran. The second edit box contains the name of the set. This name is used when the set is referenced in IMAT^{4}XL. It is also exported as a comment if the set definition is exported to Nastran. The final edit box contains the definition of the set. The set definition contains space- or comma-delimited numeric values. Ranges of values can be entered by separating the beginning and end of the range with a colon, such as in "1:10." An example of a valid set definition is "1, 2, 4:6, 9, 12:15."

Previously created sets stored in bulk data or OP2 files can be imported by pressing the **Import** button and selecting the desired file. All sets found in the file will be added to the listbox.

Any sets that have been imported or created by the user can be exported for future use by selecting the sets in the listbox and pressing **Export**. The export format follows that of the SET bulk data entry in Nastran.

Sets can be edited by selecting them in the listbox and pressing the **Edit** button. A form identical to that of the **New** button will be shown with the values of the selected set prepopulated. Once the desired changes have been made, press **OK** to save the changes. Only one set can be edited at a time.

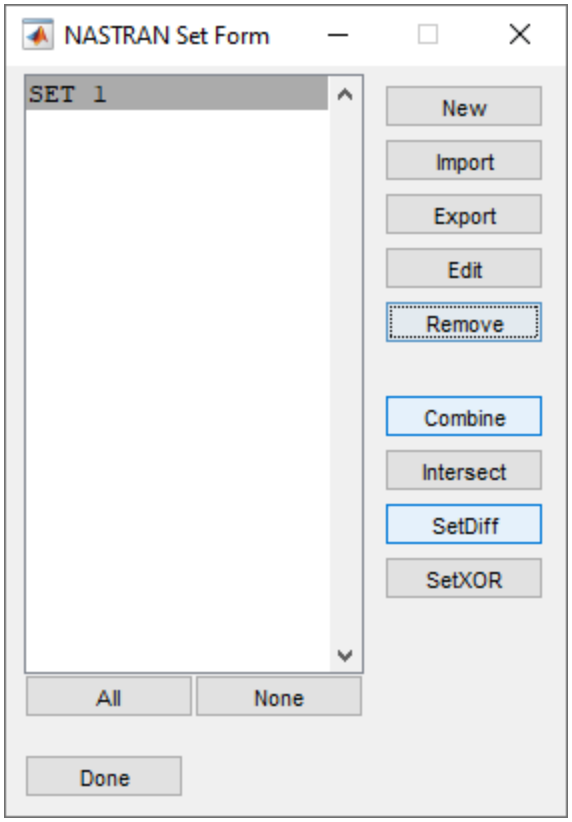
To remove a set from memory, select it from the listbox and press the **Remove** button. It will be removed from the table.

A powerful feature of the NASTRAN set form is boolean operations. These operations can be used to create new sets by applying operations to existing sets. The available operations are listed on the right side of the listbox and operate as follows:

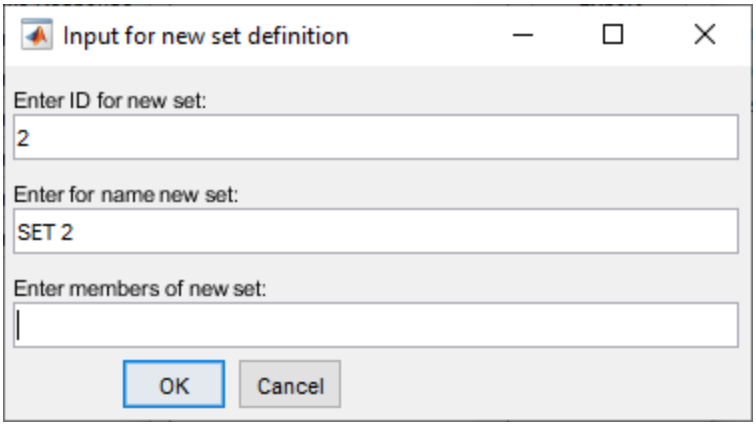
- **Combine**: This operation combines all members of the selected sets and creates a new set.
- **Intersect**: This operation creates a new set with only the members that are part of all selected sets.
- **SetDiff**: This operation creates a set with members that are in the first selected set, but not in the second.
- **SetXOR**: This operation is the opposite of intersect. It creates a new set with members that are in one set but not the other.

When accessed from the main form, the NASTRAN set form is used to define sets. When accessed from the Excel Export Options form, it is also used to select the set to use for paring the data. To set the paring set, select it from the form, and press **Done**. The Options form will display which set will be used for paring. If no set is desired, press the **None** button to deselect all sets, and then press the **Done** button. The Options form will show that no set will be used for paring by displaying "(None)."

It is important to note that sets are used to pare results (stresses, forces, etc) and coordinate traces are used to pare displacement shapes.



The NASTRAN Set form provides an easy way to manipulate sets.



New sets can easily be created from scratch.

Coordinate Traces

Coordinate traces are created just like sets. To create a new coordinate trace, press the **New** button. This will launch the New Coordinate Trace form shown below. This form has three edit boxes, a table,

and a number of buttons to help define the coordinate trace. The first edit box on the left hand side contains the name of the coordinate. This is used to identify the coordinate trace on the main coordinate trace form.

The second and third edit boxes are used to add members to the set. The "Node" edit box is used to define the node numbers for the coordinate trace. This is space- or comma-delimited numeric values. Ranges of values can be entered by separating the beginning and end of the range with a colon, such as in "1:10." An example of a valid node definition is "1, 2, 4:6, 9, 12:15." The Direction edit box is used to define the directions for the coordinate trace. These are combined with the nodes to add members to the trace. Valid directions are "X", "Y", "Z","RX","RY", and "RZ". A direction of "+" or "-" can be optionally added. Like the nodes, these are entered as comma-delimited values.

Once the node and direction values are set, press the **Add** to add the items to the table. If any items in the table need to be removed, simply select the desired rows of the table and press the **Remove** button. To select all items in the table press the **All** button. To deselect all items in the table press the **None** button. To select all items of a specific coordinate, select the coordinate from the pull down, select the desired direction and press **Apply**. When all desired coordinates are in the table, press **Done** to create the new coordinate trace.

Previously created coordinate traces stored in universal files can be imported by pressing the **Import** button and selecting the desired file. All ctraces found in the file will be added to the listbox.

Any coordinate traces that have been imported or created by the user can be exported for future use by selecting them in the listbox and pressing **Export**. They will then be exported to the Universal file format.

Coordinate traces can be edited by selecting them in the listbox and pressing the **Edit** button. A form identical to that of the **New** button will be shown with the values of the selected ctrace prepopulated. Once the desired changes have been made, press **OK** to save the changes. Only one ctrace can be edited at a time.

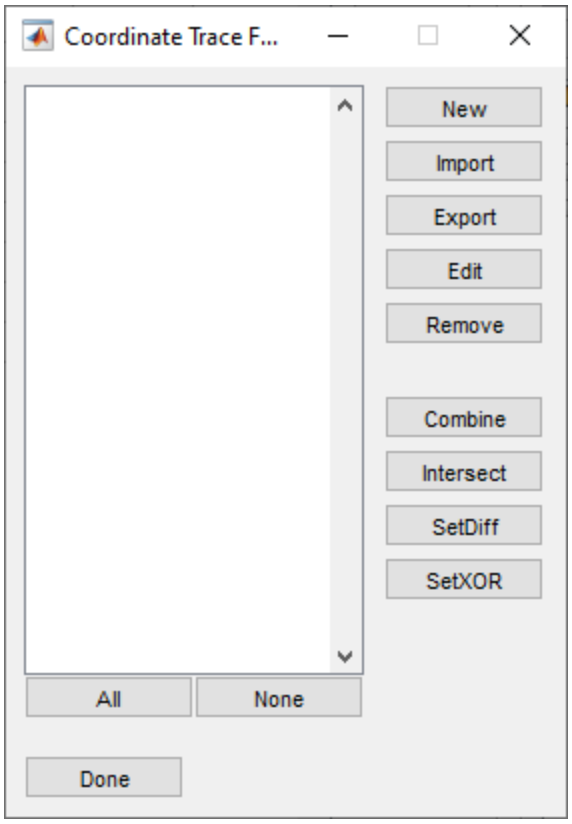
To remove a coordinate trace from memory, select it from the listbox and press the **Remove** button. It will be removed from the table.

A powerful feature of the coordinate trace form is boolean operations. These operations can be used to create new coordinate traces by applying operations to existing ones. The available operations are listed on the right side of the listbox and operate as follows:

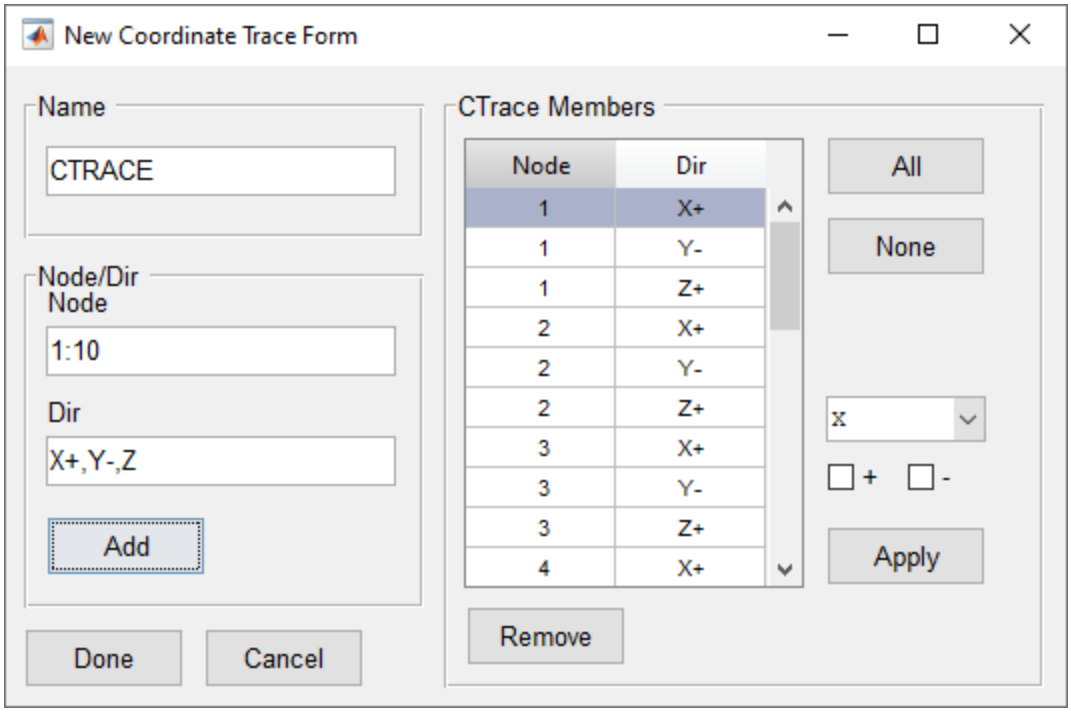
- **Combine**: This operation combines all members of the selected ctrace and creates a new ctrace.
- **Intersect**: This operation creates a new ctrace with only the members that are part of all selected ctraces.
- **SetDiff**: This operation creates a ctrace with members that are in the first selected ctrace, but not in the second.
- **SetXOR**: This operation is the opposite of intersect. It creates a new ctrace with members that are in one ctrace but not the other.

When accessed from the main form, the coordinate trace form is used to define coordinate traces. When accessed from the Excel Export Options form, it is also used to select the coordinate trace to use for paring the data. To set the coordinate trace, select it from the form, and press **Done**. The Options form will display which coordinate trace will be used for paring. If no coordinate trace is desired, press the **None** button to deselect all coordinate trace, and then press the **Done** button. The Options form will show that no coordinate will be used for paring by displaying "(None)."

It is important to note that sets are used to pare results (stresses, forces, etc) and coordinate traces are used to pare displacement shapes.



The coordinate trace form is a convenient way to create coordinate traces.

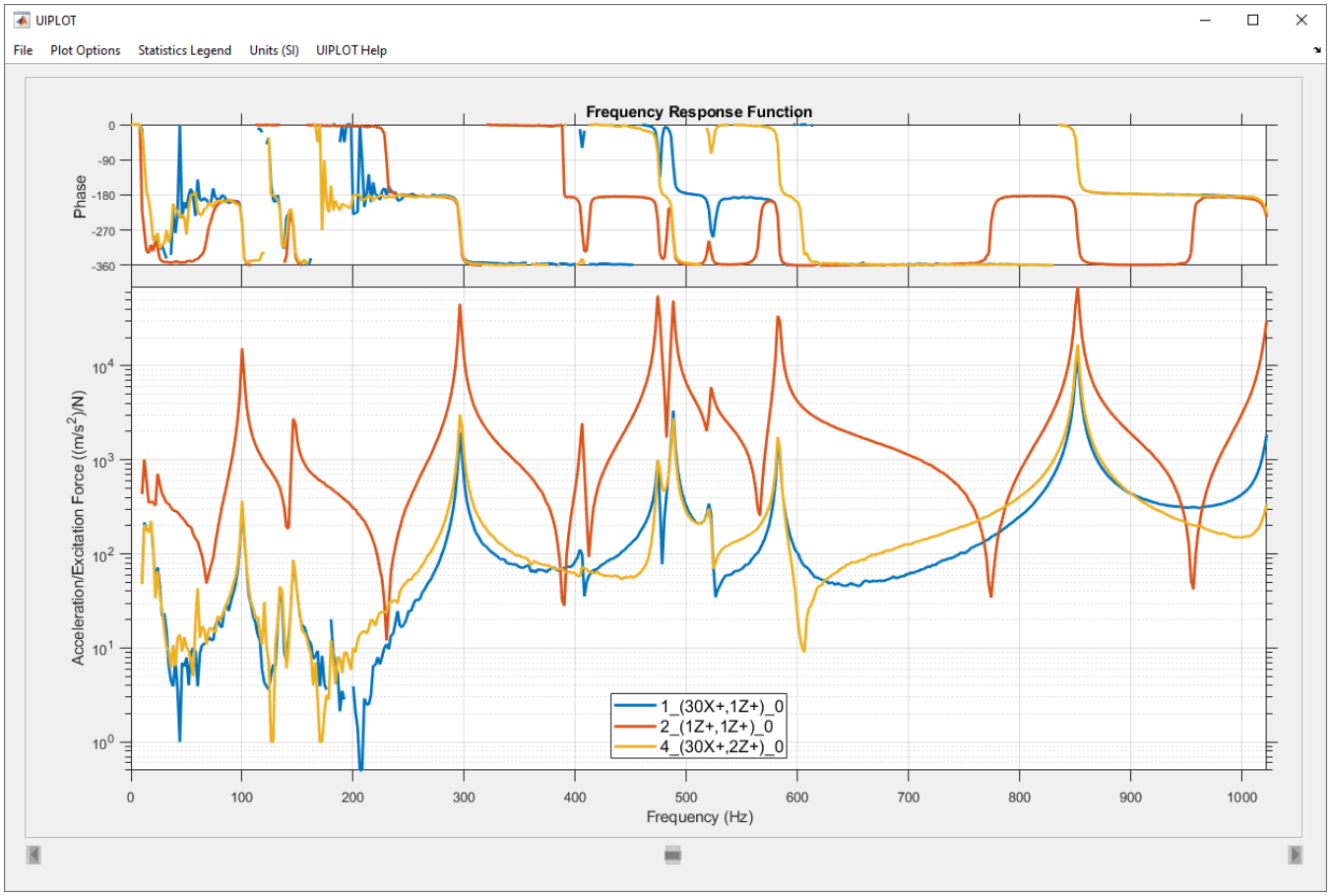


New coordinate traces can be created quickly and easily.

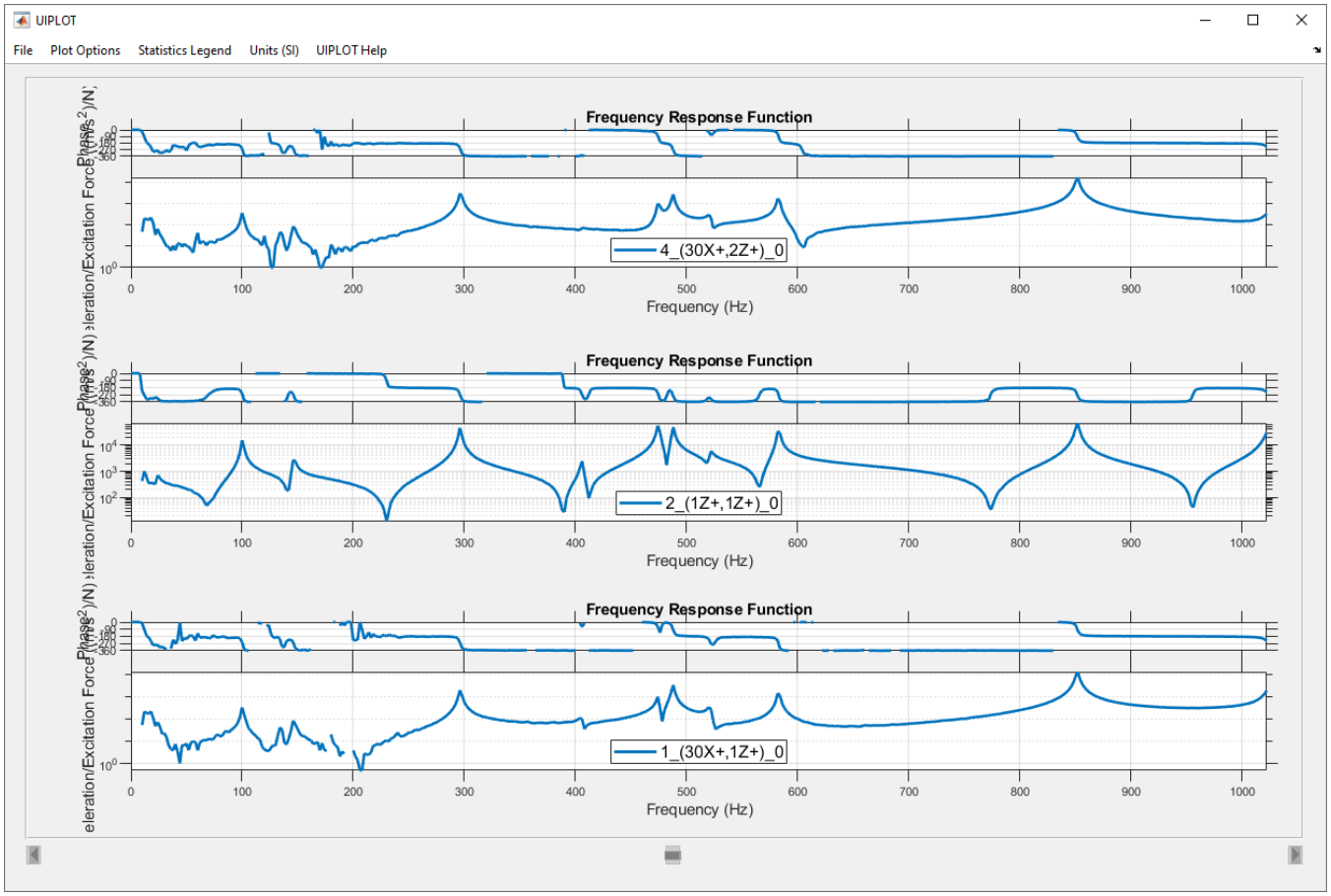
Plotting Region

The Plotting region contains buttons to create plots from loaded functions. There are two different types of plots that can be created in this region, "XY" and "Stacked." IMAT^{4}XL provides two ways to create these plots, using ATA's UIPLOT utility and sending the data directly to an Excel chart.

The XY plot is a plot that groups functions based on type. That is, similar data types will be plotted on the same plot (similar data types mean that the *FunctionType* and *OrdNumDataType* attributes of the function data are the same.) The Stacked plot creates a single plot for each function. An example of each plot, as seen in the UIPLOT mode of plotting, is shown below.



Frequency response functions plotted using "XY." Two axes were created, one for the phase and one for the magnitude.

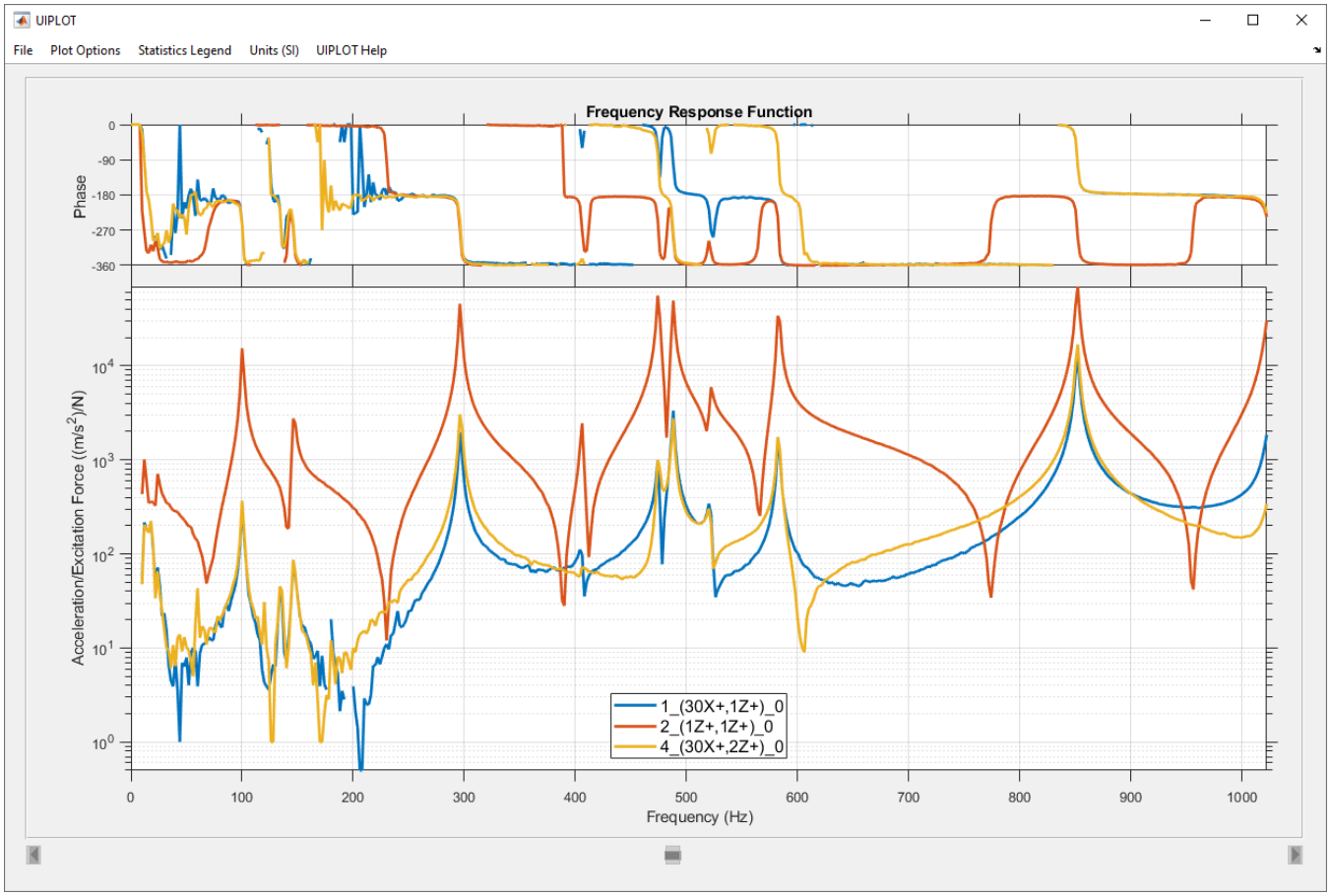


Frequency response functions plotted using "Stacked." A separate plot was created for each function.

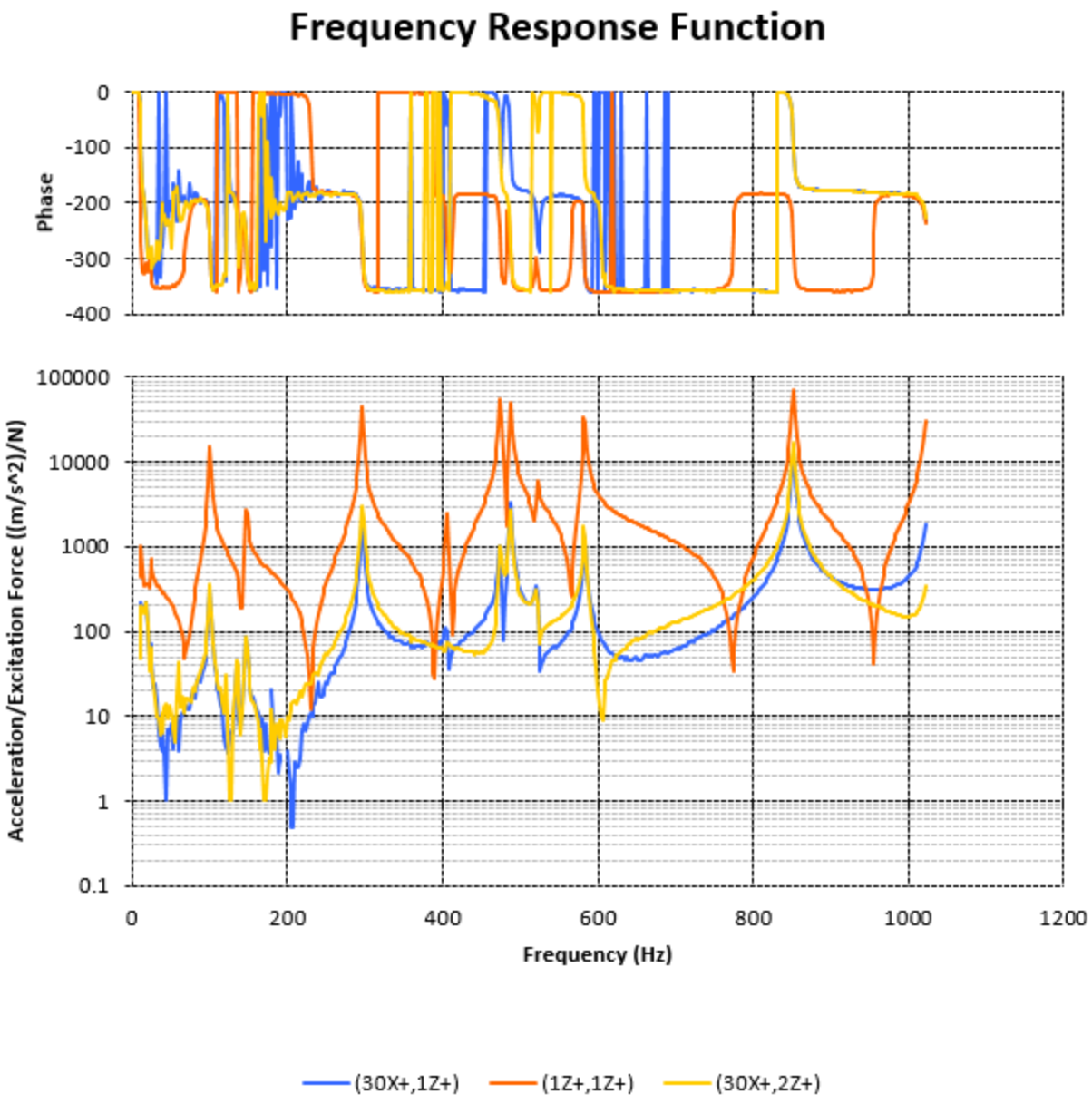
The two different modes of plotting in IMAT^{4}XL are "UIPLOT" and "Excel." The "UIPLOT" mode uses a convenient plotting utility (called UIPLOT) that is also available in ATA Engineering's IMAT software product. This utility allows easy and convenient manipulation of function data. For example, UIPLOT functions provide an easy way to change axis scales, axis labels, and axis windowing. Right-clicking on the plot will provide a list of available operations.

The second plotting mode is "Excel." This mode creates a plot of the selected data directly in Excel. The data for each function to be plotted is written in table form (in the same format as the **Send** button), and the plot is automatically created and placed to the right of the table. The plot will be formatted appropriately for the type of data being plotted.

IMAT^{4}XL is designed to make working with engineering data and Excel as easy as possible. For this reason, UIPLOT can be used as a "preprocessor" where you can set scales and windowing for the eventual export to Excel, essentially using it to design your Excel plot. Once a plot is formatted in UIPLOT, it can be sent to Excel by selecting **Send to Excel** from the contextual menu, which can be accessed by right clicking on any plot in UIPLOT. An example of a formatted plot in UIPLOT and its corresponding exported Excel plot is shown below. More information about the available context menu options is available in the *IMAT Help* menu item.



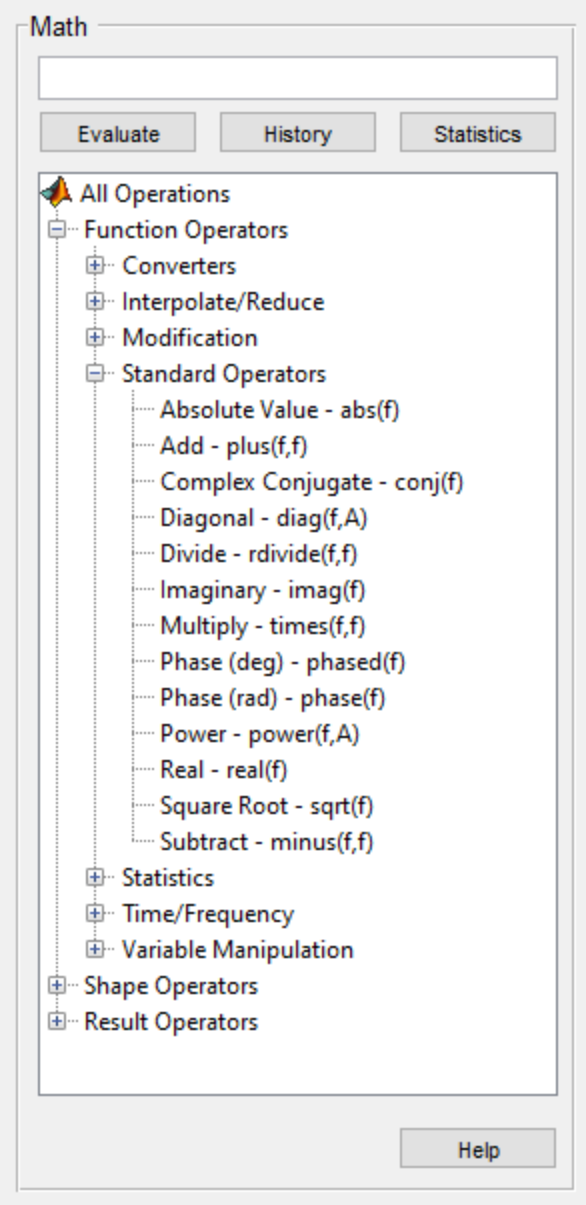
Plots can be easily formatted in UIPLOT.



Plots sent from UIPLOT to Excel will keep the same formatting.

Math Region

Another one of IMAT^{4}XL's powerful capabilities is the ability to modify or create new data using a wide range of mathematical capabilities, or show statistical information on existing data. An extensive list of math operators is available. These can be used individually, or multiple operators can be combined into one expression. There are separate math operator categories for functions, shapes, and results, as shown below.



Math operations can be applied to functions, shapes, and results, while statistics can be calculated on functions only.

The list of available math operations is listed under the "All Operations" tree. This is an expandable tree structure in which the available Math operators are presented in groups (branches of the tree) to assist in finding the desired operator. To apply a math operator, start by selecting the desired data from the Data table. Next, double click the desired math operator from the tree. Operations applicable to functions are listed under "Function Operators." Operations applicable to shapes are listed under "Shape Operators." Operations applicable to results are listed under "Result Operators."

Once the operator is double-clicked, it will appear in the edit box, directly above the **Evaluate** button. If an expression with multiple operators is needed, double-click each operator and arrange them as required in the edit box. The expression can also be typed in directly.

While double-clicking the operation from the tree will assist in creating the math expression, direct user input is often required since many of the math operators have additional input arguments. For instance, the scaling operator needs the user to enter the scale factor that should be applied.

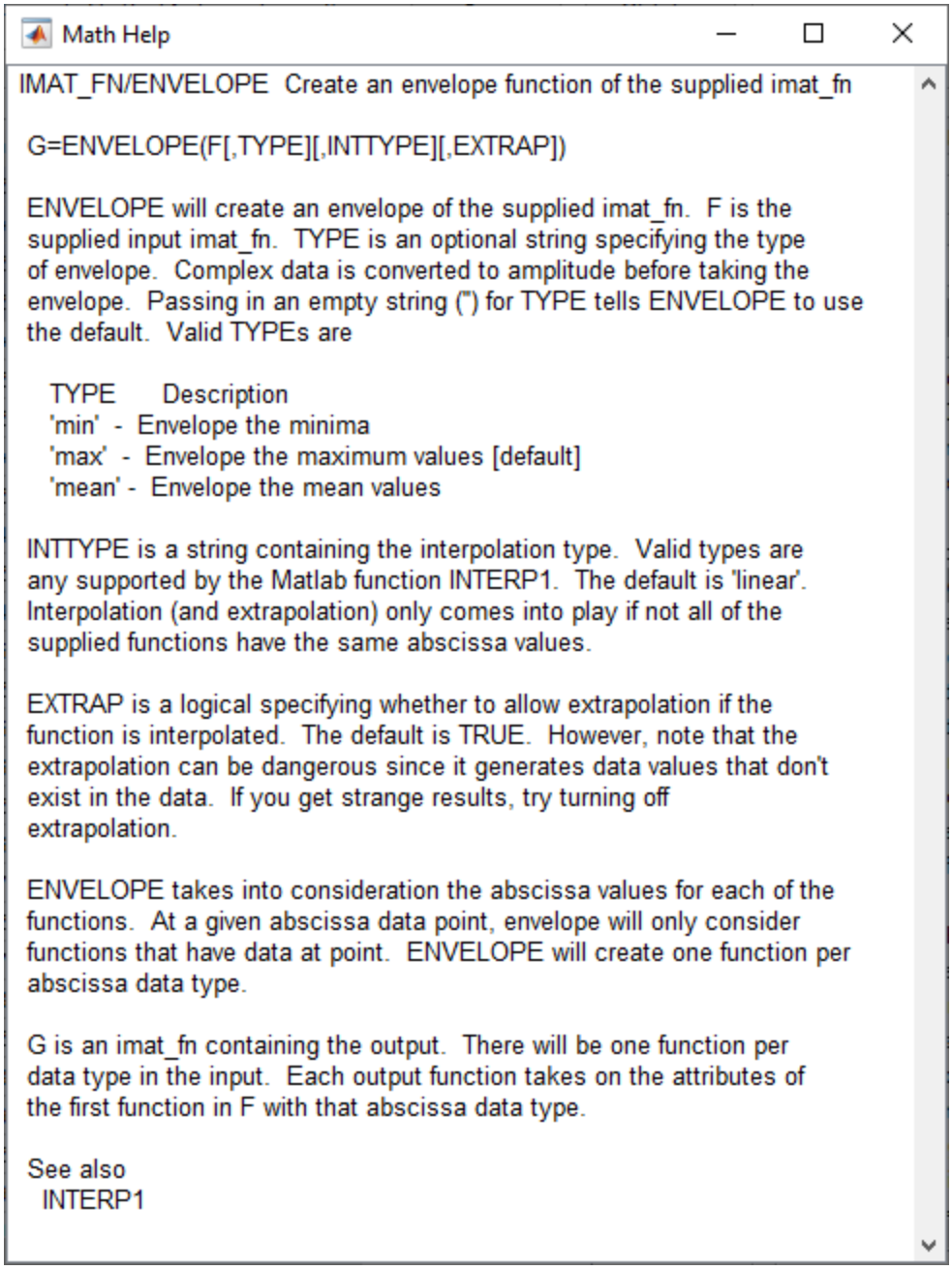
When operators are added to the expression box, placeholders will appear showing what additional arguments are needed for the expression. Optional arguments appear in square brackets (`[]`). The nomenclature for these arguments is explained below.

Code	Meaning
f	Function input. This will be filled in with the functions selected in the Data table.
g	Additional input functions
A	Numeric input
S	String input

Data that is referenced in an expression (such as in inputs `f` and `g` above) has a specific syntax. It begins with a "`#`" and then lists a letter corresponding to the data type, followed by the index, or range of indexes, of the desired data from the table. The index of each data item is listed in the first column of the table. For example, to reference function 10 in the table, "`#f10`" would be used. To reference functions 16 through 20, "`#f16:20`" would be used. To reference a noncontiguous list of items and ranges, the list of items is comma delimited and surrounded by "`<`" and "`>`," such as in `<#f1,#f5,#f13:15>`. Shape 10 in the table would be referenced by "`#s10`." Result 10 would be referenced by "`#r10`". When math expressions are created by highlighting items in the table and double-clicking math operations, this indexing will be handled automatically.

To evaluate the expression, press the **Evaluate** button. A message will appear after the operation has been performed successfully. If an error occurs during the evaluation, an error message will appear with the approximate location in the expression where the error occurred.

For more information about any of the available functions, or to study the required syntax, simply select the math operator and press the **Help** button. An example help window is shown below.



The Math Help window is a convenient way to learn about a math operation.

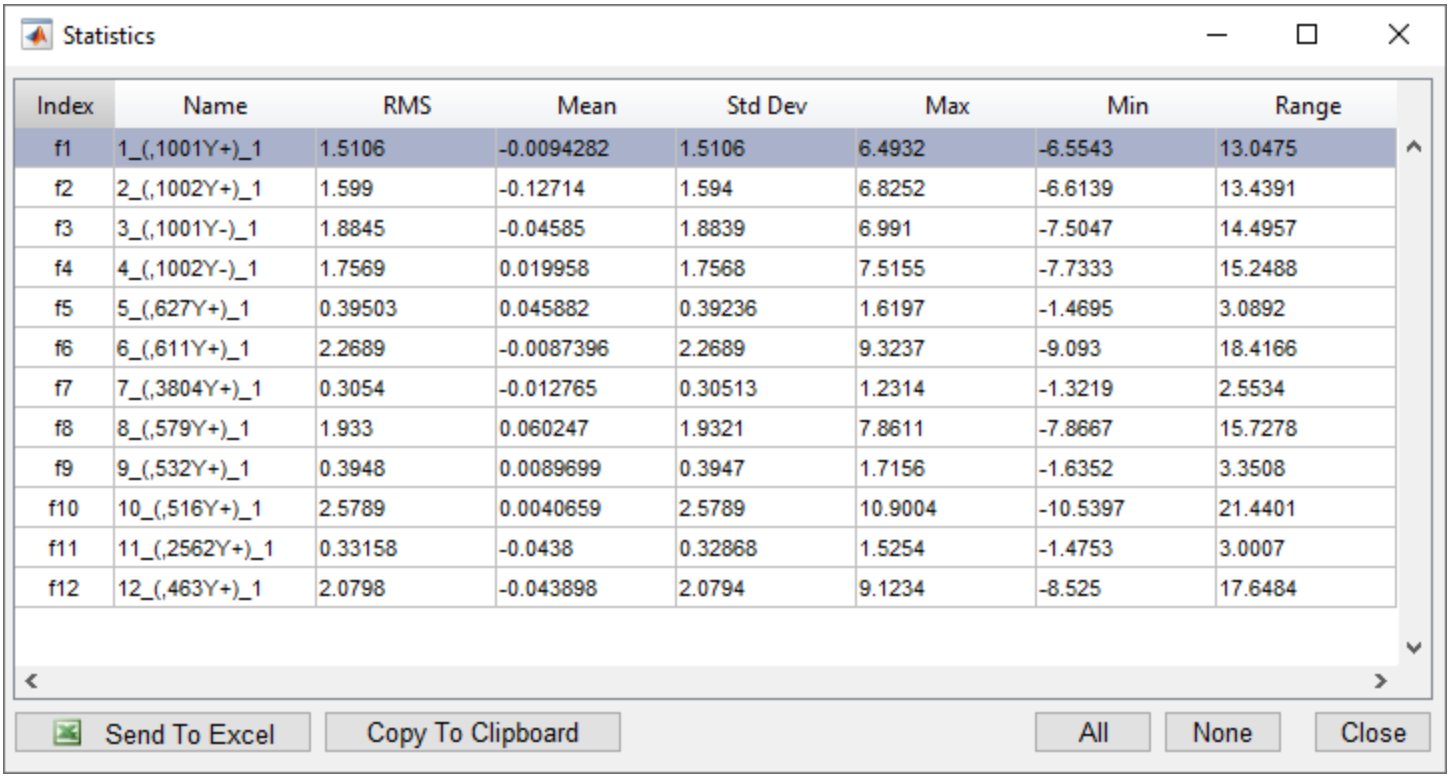
History

A List of previously evaluated math expressions can be accessed by clicking the **History** button. If the previous expression created valid output it will appear in a black font, while expressions causing errors will be in a red font. If desired, a previous expression can be selected and the expression will then be placed into the edit box. The list of expressions will be reset if IMAT^{4}XL is closed.

Statistics

In addition to creating new data with math operations, IMAT^{4}XL can also provide statistics on existing functions. After highlighting functions in the data table, clicking the **Statistics** button will open a window showing the RMS, Mean, Standard Deviation, Maximum, Minimum, and Range of the data. On complex data many of the functions are calculated on the magnitude of the data; the minimum is often 0 giving a range equal to the magnitude of the maximum.

From the Statistics window the data can be copied to the clipboard, or output directly to Excel. If sent to excel it can't be read back in, as the statistical information doesn't fully define a function.

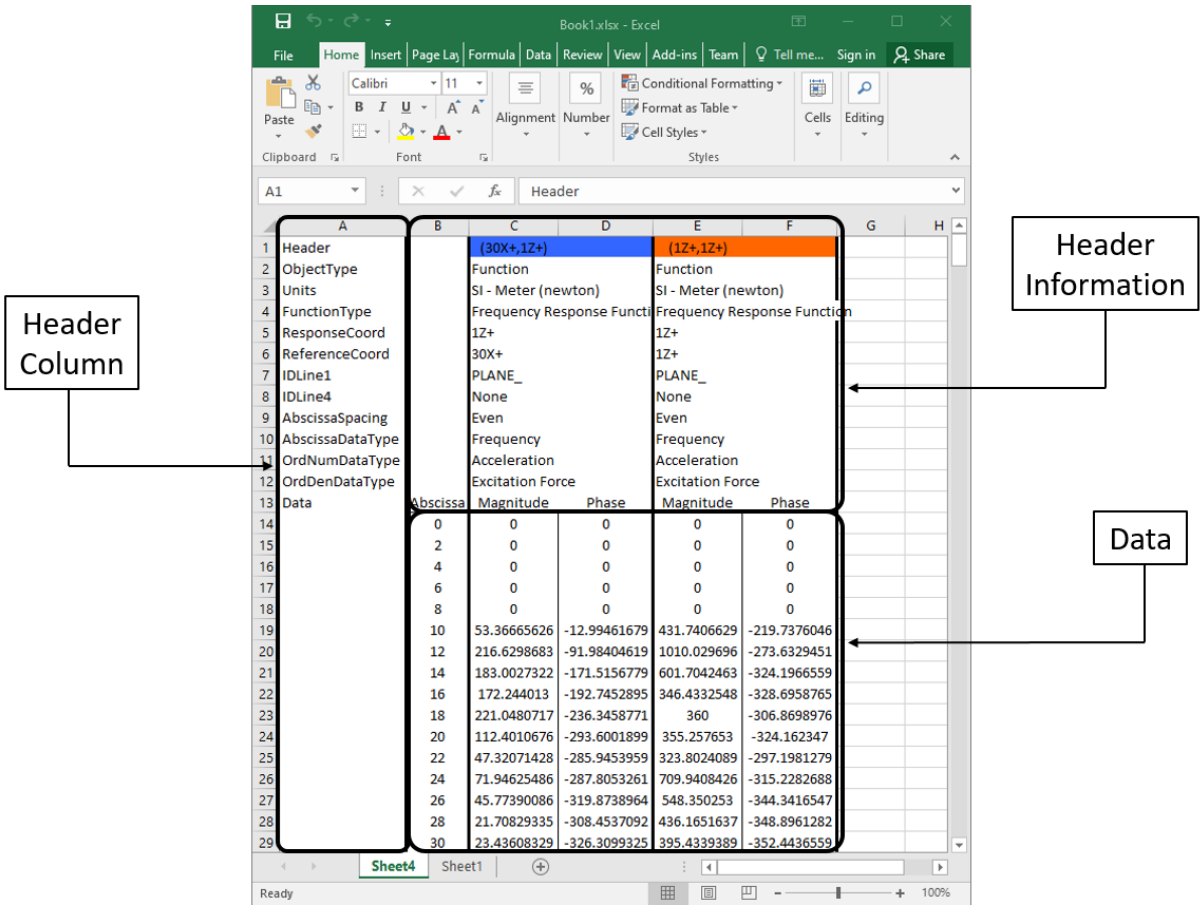


The statistics window allows for easy inspection of multiple function's statistics

Excel Result Format

Data sent from IMAT^{4}XL to Excel has a specific format. This format is intuitive for the user to manipulate and is easy for IMAT^{4}XL to understand. When manipulating data that has been sent to Excel, it is important to understand the format so that the data can be successfully re-imported.

The format of an exported worksheet is divided into three areas: the header column, the header information, and the data itself. These areas are labeled below.



Header Column

The header column defines the type of information the corresponding rows are describing. The header column must always be the first column in the worksheet. The first two rows of the header column must be “Header” and “ObjectType,” respectively. The last item in the header column must be “Data.” This defines the row where the data table begins. All data header information must be in this row.

All rows between "ObjectType" and "Data" must be valid attributes for the object defined in "ObjectType". They are not required, but help define the object when re-importing into IMAT^{4}XL.

Header Information

The header information is used to help define each data item. The range of data that is pared for each data item is defined as the range between subsequent labels in the "ObjectType" row. Because of this,

the *ObjectType* and all other header information must be placed in the first abscissa column or the first ordinate column for each data item. Notice that in the figure above, all header information is placed in columns B and E, which are the first abscissa and first ordinate columns of the two data items. The data in cells B2:B11 could also be placed in cells C2:C11, as column C is the first data column for the first function.

Each row of header information must either be blank, "none" without quotes, or contain a valid value for the corresponding attribute defined in that row. A blank cell will assume the same value as the function before it received for that attribute. Entering "none" without quotes will skip setting that attribute, which will either leave it empty or get the default value. Incorrect values for an attribute will result in an error when the data is imported into IMAT^{4}XL. If you are unsure of the correct value either enter "none" to skip setting it, or use the *IMAT Help* menu item or Attributes button on the main IMAT^{4}XL form to view the valid values for each attribute.

The colors in the first row of data, as well as any formatting or plots in the worksheet, are not considered when the worksheet is imported into IMAT^{4}XL.

Data

The Data area contains the analysis information for the data item. The analysis information comes in two parts: the "Abscissa" and "Ordinate." The "Abscissa" information (such nodes or elements) describes where the "Ordinate" data came from. In the case of stress results, the "Abscissa" would be considered the node, element, and layer a particular stress result came from. The "Ordinate" data would be all of the stress components at the location the "Abscissa" described. The specific format for each data type is described further in the following table.

	"Abscissa"	"Ordinate"
Function	The function "abscissa" columns contain the abscissa data for the function, which refers to the X-axis values of the function. This is always one column labeled "Abscissa."	The function "ordinate" columns contain the ordinate data, or Y values, of the function. This can be one or two columns. If the data is complex, then either "Magnitude" and "Phase" or "Real" and "Imaginary" are the valid "ordinate" columns. If the data is real-only, then only the "Real" column is needed.
Shape	The shape "abscissa" data is a single column labeled "Node." These are the nodes that the corresponding displacements are at.	The shape "ordinate" data is comprised of either three or six columns depending on how many DOF are in the displacement shape. These are labeled as X, Y, Z, RX, RY, and RZ. If the data item contains only three DOF, then only X, Y, and Z will be present. If the shape data is complex, there will be an "Imag" and "Real" portion for each DOF. For the three-DOF case this would be: "Real X," "Imag X," "Real Y," "Imag Y," "Real Z," "Imag Z." They must be in this order and must all must be present.
Result	The result "abscissa" columns can take on two of forms depending on the type of result data. For nodal and elemental data such as dis-	The result "ordinate" data columns forms match those of the result. The "ordinate" columns specify the data components and the node and/or element locations. For nodal data, there are three or six "ordinate" columns labeled X, Y, Z, and RX, RY, and

	placement and energy, this is a single column labeled "Node" or "Element." This is the node/element the analysis information is located at. For elemental data at nodes such as stress, this is three columns "Element," "Node and "Layer." This is the element and node the result data is located at. The layer is used for shell and composite results to indicate which layer the data is referring to.	RZ (only required if the data has rotational DOF). These correspond to the force in each direction at the corresponding node. For data-at-nodes-on-elements (DANOE) results, the "ordinate" columns depend on the type of data. For stress results they will be labeled XX, XY, YY, XZ, YZ, and ZZ. For force DANOE results, they will be labeled X, Y, Z, RX, RY, and RZ. Each of these columns corresponds to a data value at the corresponding element, node, and layer. As with shapes, complex data will have a "Real" and "Imag" portion for each of the required ordinate columns. They must be in the correct order and all must be present.
Matrix	The matrix "abscissa" column is unlike the other data types. It consists simply of one column of DOF that correspond to each row of the matrix. If there are no corresponding DOF, this column will be empty.	The matrix "ordinate" data columns form the columns of the matrix. Like the "abscissa" columns, the headers for the "ordinate" data are the DOF that correspond to each column. If there are no corresponding DOF, the header row will be empty.

It is important to note that the "Abscissas" and "Ordinates" can be organized in two separate ways. They can either be condensed if all of the exported data has the same "Abscissa" or can be explicitly written out for each data item. If they are condensed, then the "Abscissa" column only appears before the first data item. All data items will only have "Ordinate" columns, and they will all reference the "Abscissa" columns in front of the first data item. If each "Abscissa" is written out, each data item will be written independently, and each will have its own "Abscissa." It is important to note that these two structures do not mix – there is either a single "Abscissa" for all data items on a worksheet, or there is one for each data item. This does not apply to matrix data, because the "Abscissas" and "Ordinates" are the DOF for each matrix index.

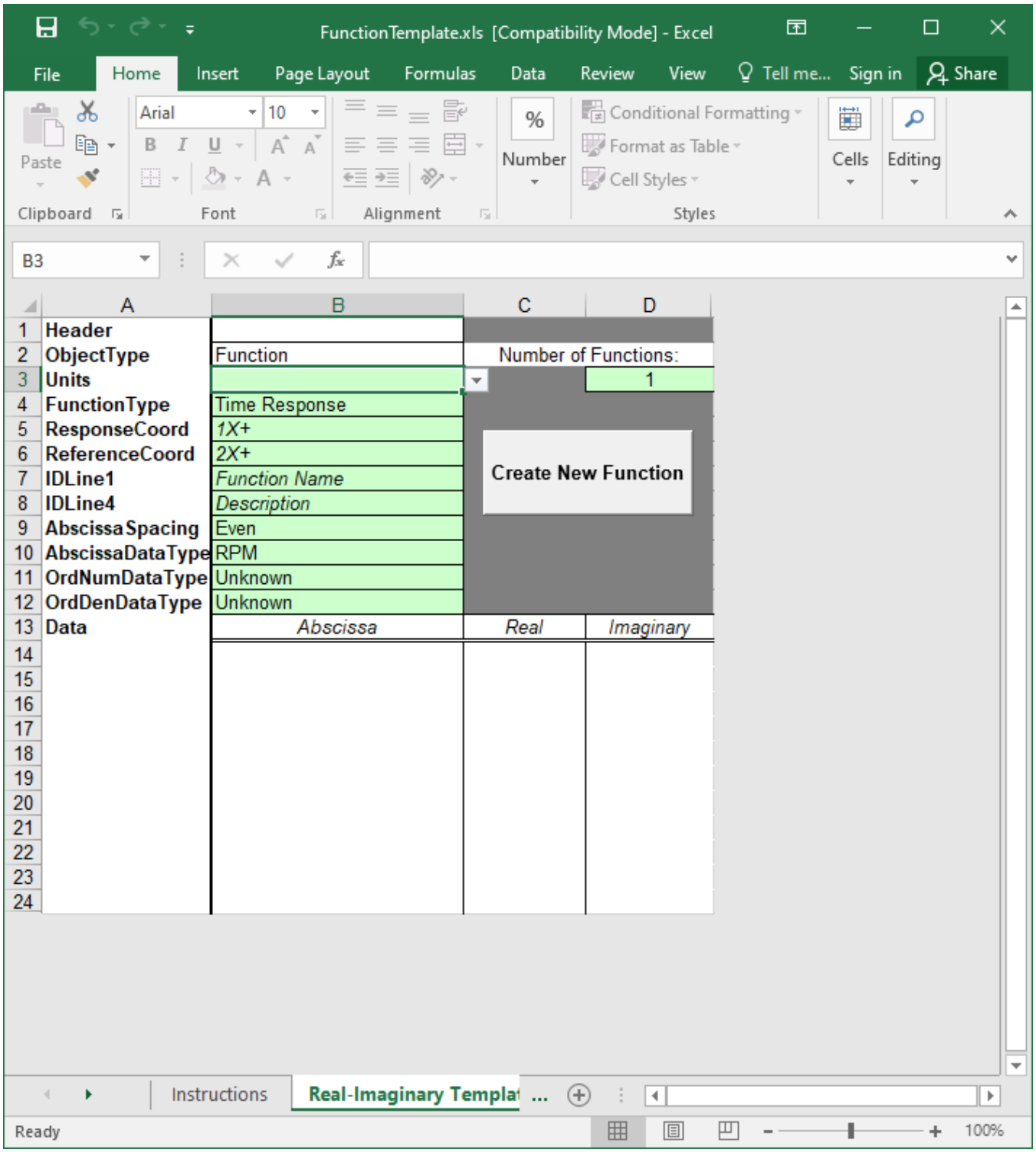
Inside each "Abscissa" and "Ordinate" there can be no blank spaces. If a component is meant to be 0, it must be set explicitly as 0. Blank cells embedded in the expected data range for a data item will cause unexpected results. The expected data range is from the row marked "Data" in the header column to the last row that contains data for that data item.

Templates

IMAT^{4}XL provides a number of Excel templates to aid in the creation of data to import into IMAT^{4}XL. These three Excel workbooks (*FunctionTemplate.xls*, *ShapeTemplate.xls* and *ResultTemplate.xls*) can be found in the installation folder of IMAT^{4}XL, or under the *Templates* menu on the main IMAT^{4}XL window.

Each workbook contains several worksheet templates. Each template contains a number cells that can be set with custom values. Only cells colored green contain values that can be changed. Cells in italic font expect a value to be entered. This may be a response coordinate, IDLine, or description. Cells in non-italic font are pulldowns with allowable values for that attribute. As values are changed, the data headers will change accordingly. Once all of these values are set as desired, press the **Create** button to create a new worksheet template with the allowed values.

Once the blank workbook is created, you can copy the blank template to another workbook and enter your desired data in each data column. From there, connect to the workbook and retrieve the worksheet to import that data into IMAT^{4}XL.



IMAT^{4}XL provides templates to assist in creating data in Excel to import.

Functions

The Function template workbook contains two templates: one for real-imaginary data entry and one for magnitude-phase data entry. On each template, you can set the *Function Type*, *Response Coordinate*, *Reference Coordinate*, *IDLine1*, *IDLine4*, *AbscissaDataType*, *OrdNumDataType* and *OrdDenDataType*.

Shapes

The Shape template contains a single template. The ordinate columns will change based on whether the *DOFType* is set to *"3DOF"* or *"6DOF"*. The other values that can be set are the *Frequency*, *Damping*, *IDLine1*, *IDLine3*, *OrdNumDataType* and *OrdDenDataType*.

Results

The Result template contains four templates: One for all defined result types ("Result Template"), and three for unknown result types.

For defined result types, changing the *DataLocation* will change the first columns to reflect the correct *Abscissa* for that *DataLocation*. Changing the *DataCharacteristic* will change the appropriate *Ordinate* columns. The other editable values are *Analysis Type*, *Result Type*, *IDLine1*, and *Name*.

The other three templates are for *Unknown* data types - one template for each *DataLocation*. Instead of changing the *Ordinate* column headers by setting the *DataCharacteristic*, the column headers can be changed directly using a pulldown in each header cell.

Examples

The following sections contain two examples demonstrating various parts of IMAT^{4}XL.

FRF Calculation

A frequent task for test engineers is to compute frequency response functions using a series of acceleration time histories. These functions are used to identify natural frequencies, damping ratios, and mode shapes of a structure. They are calculated using the following formula:

H = Output / Input = Y / X = GYY / GXY

Here, the Output, or GYY, is the power spectral density (PSD) of the responses of the system. The Input, or GXY, is the cross spectral density (CSD) between the inputs and outputs of the system.

The computation of the FRF involves the following steps:

1. Read in acceleration time histories.

2. Compute the power spectral densities.

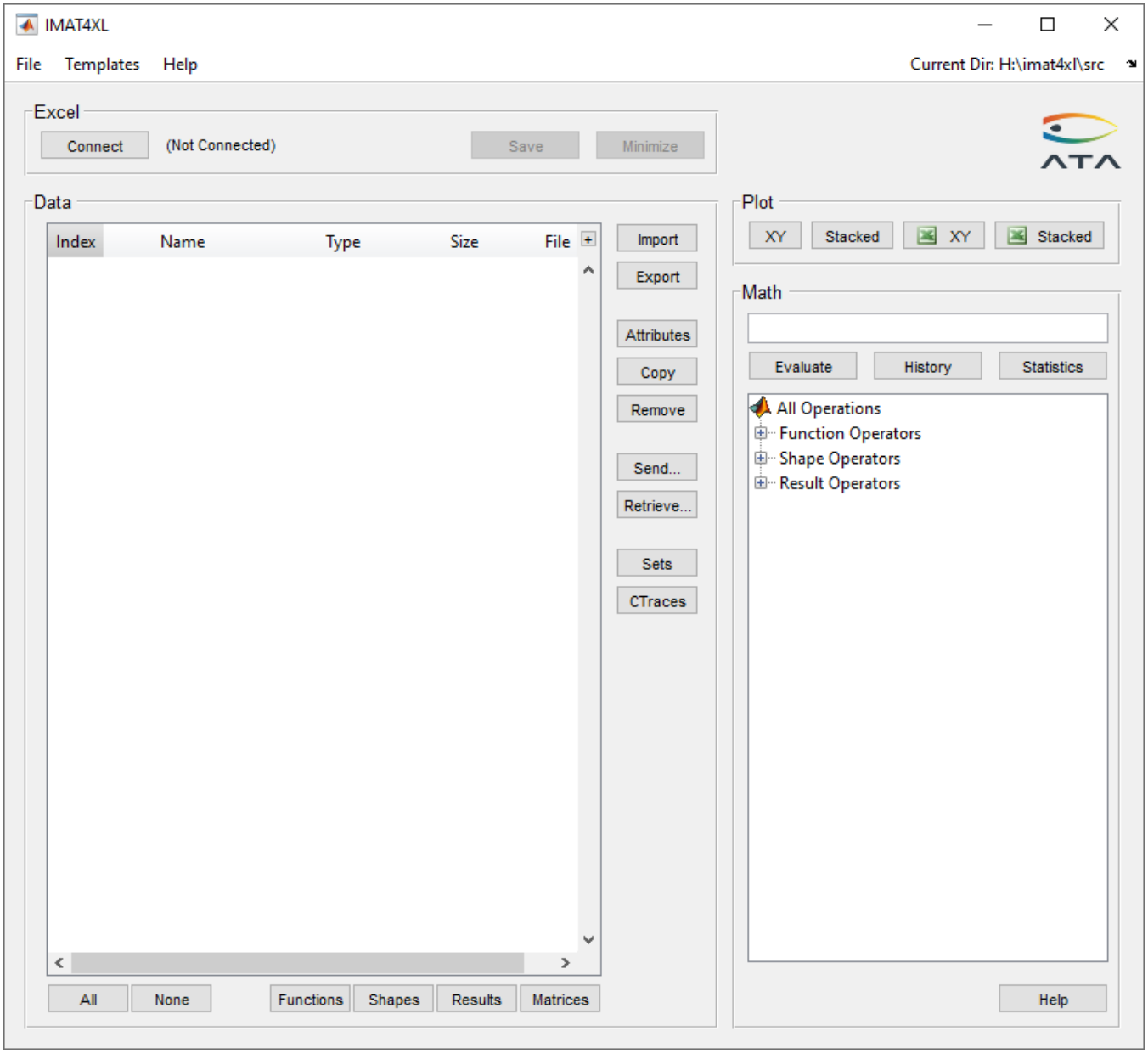
3. Compute the cross spectral densities.

4. Compute the frequency response functions (FRFs) from the PSDs and CSDs.

These steps can be performed in IMAT^{4}XL, resulting in FRFs that can be viewed in Excel, plotted, or exported into a variety of analysis file formats. The following sections show an example of this process and the specific steps to implement it in IMAT^{4}XL.

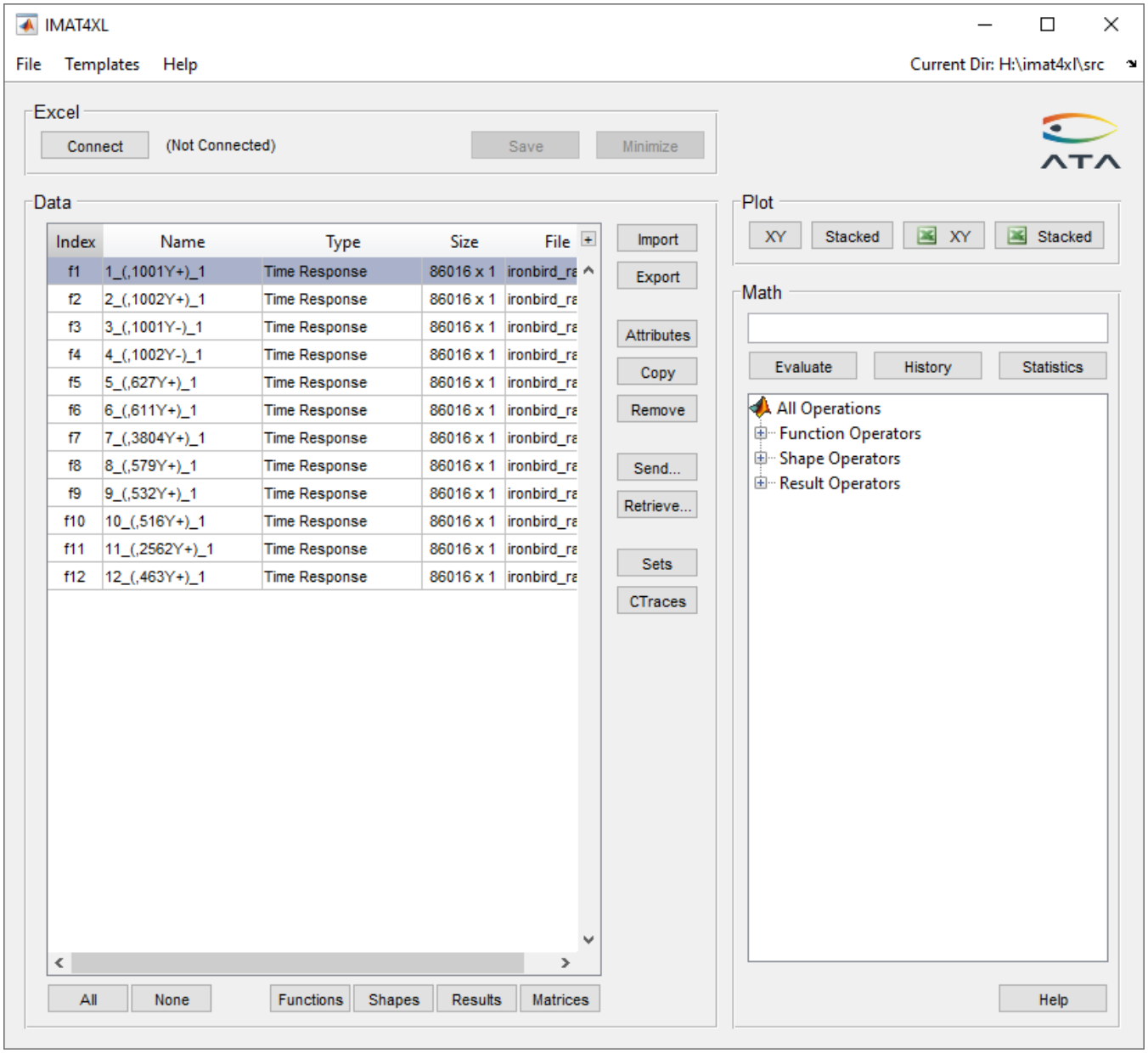
Opening IMAT^{4}XL

To open IMAT^{4}XL, select it from the *Start* menu. The application is found under *Start --> ATA Engineering -->IMAT4XL --> IMAT4XL 7.0.0*. After a short wait, the following window will appear.



Read in the time histories

To read in the time histories, click the **Import** button and then select the appropriate file. For this example, the file is named *ironbird_random.ati*. After reading in that file, the IMAT^{4}XL window will look like this:



Compute the power spectral densities (PSDs)

The PSDs can easily be calculated by using the math operations on the right side of the IMAT^{4}XL form. To create the math operation, select all of the functions in the table, and then double-click on the *Power Spectral Density* math operation. It is found under *Function Operators --> Time/Frequency*.

After double-clicking the math operation, the math edit box should show `psd(#f1:f12,A,[A])`. This indicates that the PSD operation is going to be performed on functions 1 through 12 in the table. The `A` and `[A]` arguments indicate that more arguments are required to successfully complete the math

operation. In general, `A` means that a numeric input is required. Brackets around an argument mean that the argument is optional.

By selecting *Power Spectral Density* again from the list, and pressing the **Help** button, the specific meanings of the two arguments can be found.

Math Help

IMAT_FN/PSD Compute PSDs from the supplied time history imat_fn

OUT = PSD(T,NBLK[,NAVG][,'overlap',OVPCT][,'istart',ISAMP][,'maximax',NOCT]
[,'silent'][,'peakhold'][,'noprogbar'])

PSD computes single-sided Power Spectral Density functions from the supplied time histories in the imat_fn T. All of the supplied time histories must have same length. NBLK is a required parameter indicating the window size. If it is a scalar it indicates the block size to be used in the PSD calculation. A Hamming window will be used. If NBLK is a vector or imat_fn, it should contain the window to be used. The length of the vector indicates the window size.

NAVG is an optional scalar specifying the number of averages to use. If you do not specify the number of averages or the overlap, the default for NAVG is the number of frames that can be used given the specified NBLK without overlapping, or 5, whichever is greater. If NAVG is specified, and OVPCT is not, PSD will automatically use overlap processing to achieve the requested number of averages.

If NAVG is a numeric vector, it will be treated as a vector of weighting coefficients VWEIGHT for the PSD average. The length of the vector determines the number of averages. The default is to evenly weight each frame, which is equivalent to passing in ONES(1,NAVG)/NAVG for the weighting vector. PSD normalizes the weighting vector such that SUM(VWEIGHT) = 1. This option is not valid with 'maximax'.

'overlap' is an optional input that must be followed by OVPCT to specify the amount of overlap desired. OVPCT is a numeric scalar less than 100 specifying the overlap percentage. If OVPCT is negative, that percentage of samples will be skipped between averages. For example, if OVPCT is -100, one blocksize of samples will be skipped between each block. This is useful for processing long time histories with fewer averages. If you specify OVPCT, but do not specify NAVG, PSD will calcalate NAVG to use as much of the time history as possible with the specified overlap.

'istart' is an optional input that must be followed by a numeric vector ISAMP that specifies the starting indices into the time histories for the frames to be averaged. If ISAMP does not contain integer indices, the time histories will be linearly interpolated to estimate the values of the time histories at the specified indices. You cannot specify an overlap if you specify ISAMP.

'peakhold' is an optional string that keeps the maximum values for each spectral line rather than averaging across all of the frames.

'maximax' is similar to 'peakhold' except that it does an Nth octave reduction before applying the peak-hold. It must be followed by a numeric scalar NOPT specifying the Nth octave reduction to be used. Passing in an empty value specifies the default, which is 6.

'silent' is an optional input string that suppresses output during the PSD calculation.

'noprogbar' is an optional input string that disables the progress bar.

OUT is an imat_fn of the same size as T containing the PSDs.

From above, the `A` argument is required and is the block size to be used for the PSD calculation. The `[A]` argument is optional and is the number of averages to use. For the purposes of this example, the following will be used: `psd(#f1:f12,4096,50)`.

After the correct values are entered, press **Evaluate** to evaluate the math operation. If the operation was successful, a message will be displayed and twelve new functions will have been added to the data table. Note that these new functions are blue, indicating that they were created by a math operation. The "File" column shows the specific math operation that was used to create them.

Compute the cross spectral densities (CSDs)

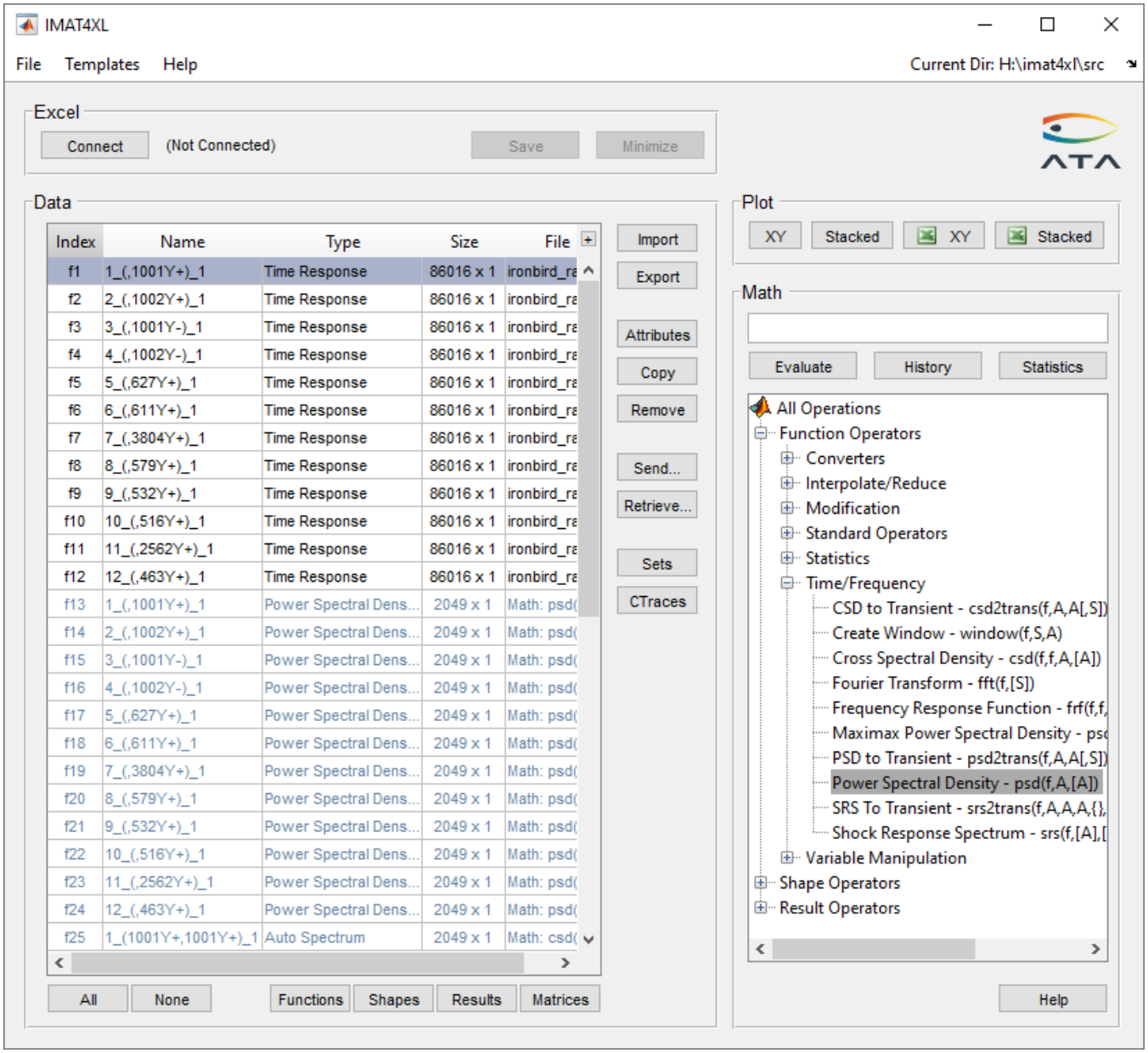
The CSDs are calculated in much the same way as the power spectral densities – the arguments are just slightly different. Again, select the time histories (rows 1 through 12) from the data table and double-click the *Cross Spectral Density* math operation. The math edit box will show `csd(#f1:f12,f,A,[A])`. This indicates that three additional arguments need to be specified: `f`, `A`, and `[A]`.

The **Help** button can again be used to determine the meaning of each argument:

- The first argument is the set of reference time histories.
- The second argument is the set of response time histories.
- The third argument is the number of blocks to use for the calculation.
- The fourth argument is the number of averages to use.

From this information, the math can be modified as follows. The reference time histories are the first two time histories, labeled `"Shaker left wing"` and `"Shaker right wing"`, so the first argument should be changed to `#f1:f2`. The response time histories are the remaining time histories, so the second argument should be `#f1:f12` (functions 1 and 2 are reference time histories and should be included because the fully populated CSD matrix is needed for the `frf` function.) For this file, the third and fourth arguments should be 4096 and 50, respectively, giving `csd(#f1:f2,#f1:f12,4096,50)` as the final equation

After the math operation, the window will look like this:



Twenty-four total functions are created by the cross spectral density function—twelve for each reference coordinate.

Compute the Frequency Response Functions from the PSDs and CSDs

Calculating the frequency response function is similar to calculating the PSD and CSD, with an added complexity: one of the arguments of the FRF function has to be reshaped. This will be explained in more detail below.

As with the other math operations, double-clicking on *Frequency Response Function* will add the basic syntax to the math edit box. The **Help** button will describe the requirements of each argument.

The general form for the FRF function is `frf(f,f)`. The help explains that the first argument is a spectral matrix of CSDs of size NCHAN x NREF (NCHAN is the number of channels and NREF is the number of references). Because IMAT^{4}XL stores all data in a single column, the reshape function is needed to create this matrix of functions.

The reshape function has three arguments:

1. The array of objects to reshape

2. The number of resulting rows

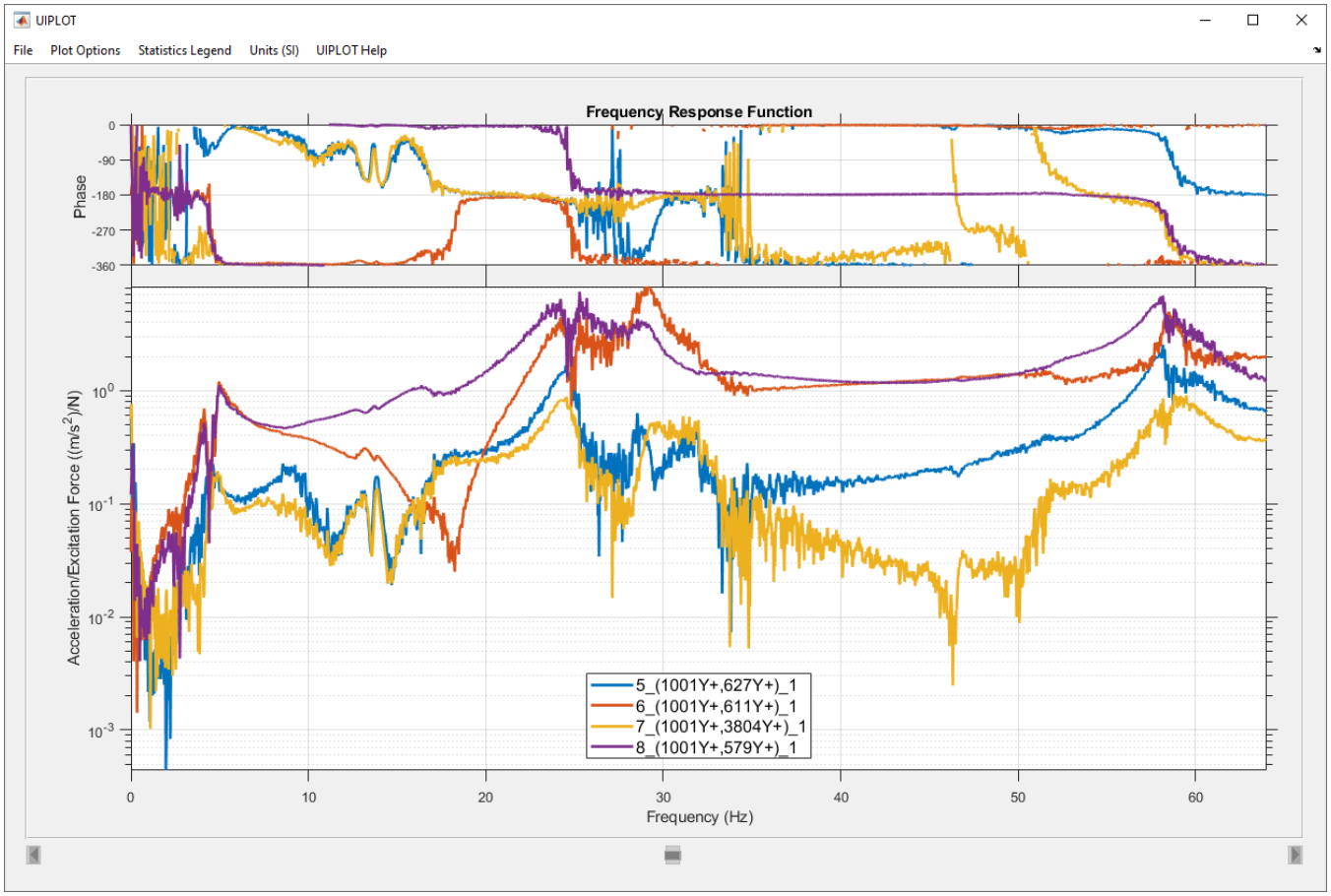
3. The number of resulting columns

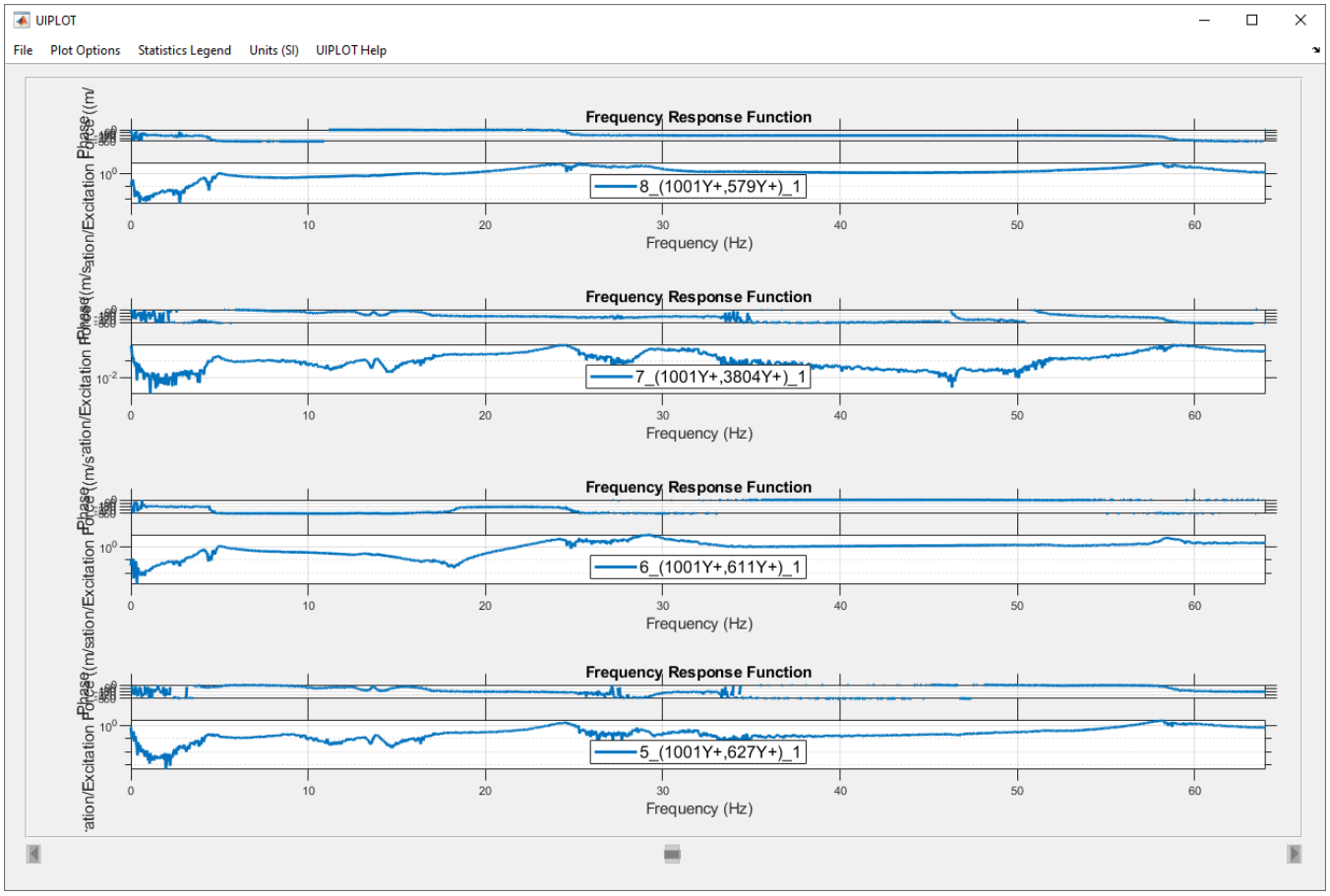
For this specific data, there are twelve channels and two references, so the correct syntax for the first argument is `reshape(#f25:f48,12,2)`. It follows that the full syntax for the FRF math operation is `frf(reshape(#f25:f48,12,2),#f13:f24)`.

After the operation has been successfully completed, 24 new frequency response functions will have been created from the original acceleration time histories.

Viewing the FRFs

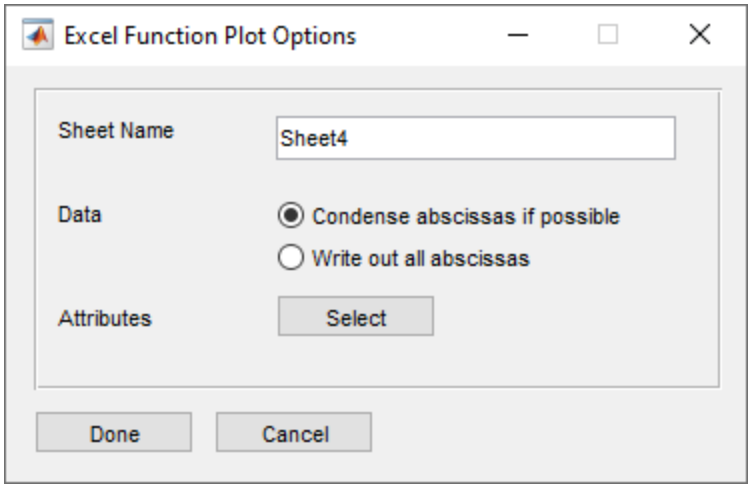
The FRFs can be viewed by selecting them from the table and pressing the **XY** or **Stacked** button. The plots shown below, in XY and stacked formats respectively, will be created if the four FRFs related to the left wing are plotted (functions 53-56).


UIPLOT
File
Plot Options
Statistics Legend
Units (SI)
UIPLOT Help
Frequency Response Function
Phase
0
-90
-180
-270
-360
Acceleration/Excitation Force ((m/s²)/N)
10⁰
10⁻¹
10⁻²
10⁻³
5_(1001Y+,627Y+)_1
6_(1001Y+,611Y+)_1
7_(1001Y+,3804Y+)_1
8_(1001Y+,579Y+)_1
0
10
20
30
40
50
60
Frequency (Hz)



To send the plot to Excel for better visualization, select *File* --> *Send Plot to Excel*. Both the raw data and the plot will be sent to Excel. The exact format of the UIPLOT window will be recreated in Excel.

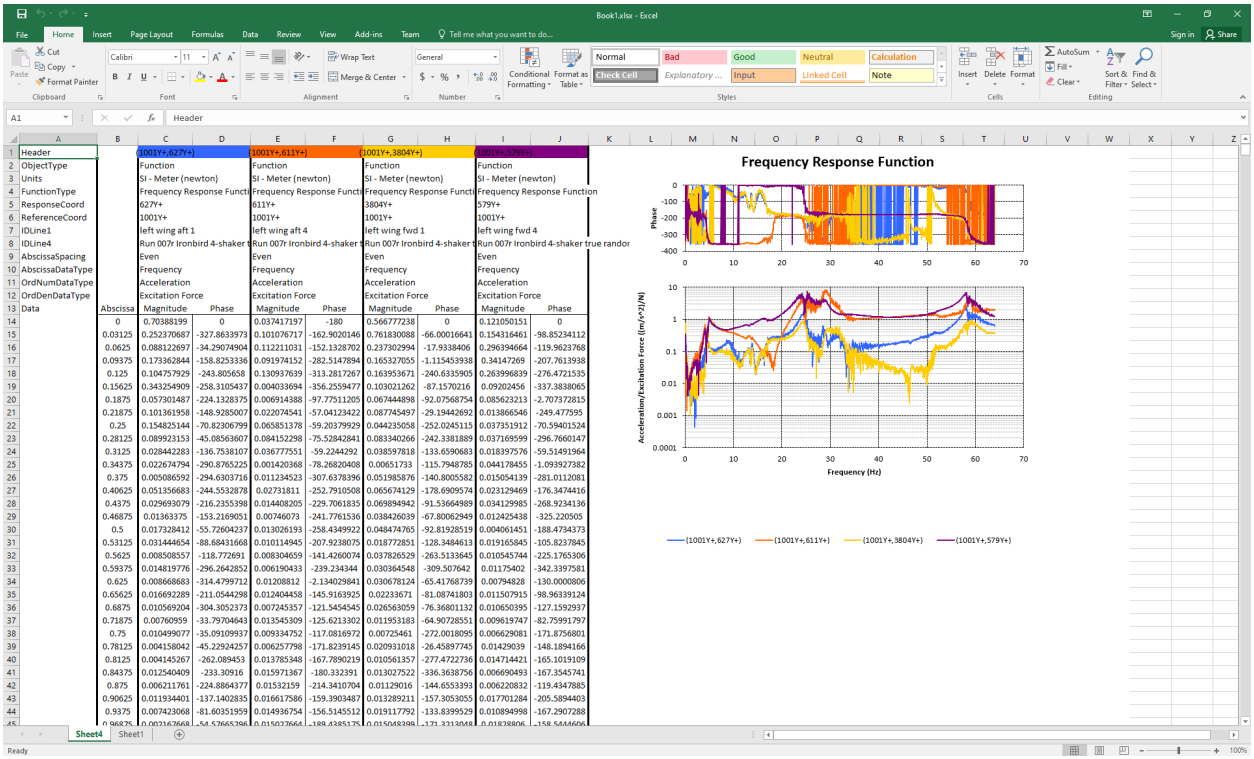
Once *Send Plot to Excel* is selected, the following form will be opened.



The Sheet Name edit box controls the name of the sheet that the plot and data will be placed on. It must be unique. The Data radio buttons control whether the X values (frequencies) will be written out for each

function, or only written out once per worksheet. The **Attributes** button selects which header information will be included on the worksheet.

After pressing **Done**, the worksheet will be populated with the data and the plots will be created. The final result is shown below.



Custom Stress Calculations

Often in engineering analysis, stress results need to be manipulated in a way that is difficult to do in a finite element analysis (FEA) program. However, this type of task can easily be accomplished in IMAT^{4}XL. Consider the case where an analyst would want to create a contour plot showing the results of an arbitrary margin calculation:

$$\text{Failure Margin} = 1 - \frac{F_{XX}}{\sqrt{F_{XY}^{\,2} + F_{YZ}^{\,2} + F_{XZ}^{\,2}}}$$

This can be accomplished in IMAT^{4}XL with the following steps:

1. Import analysis file with stresses.

2. Create a set to include only the elements of interest.

3. Send resulting stresses to Excel.

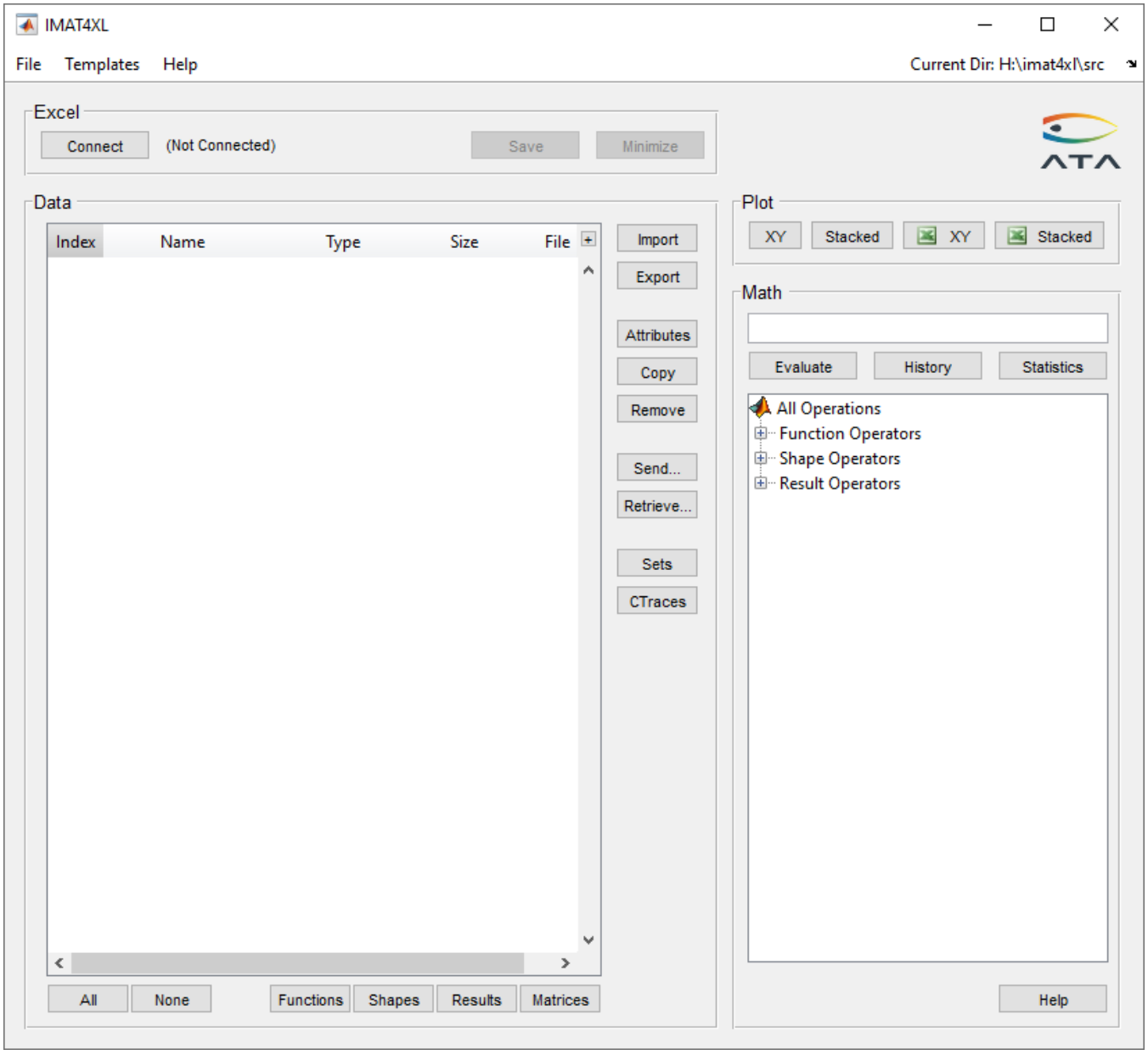
4. Calculate failure margin in Excel.

5. Import Excel worksheet back into IMAT^{4}XL.

6. Export new stress results to an analysis file.

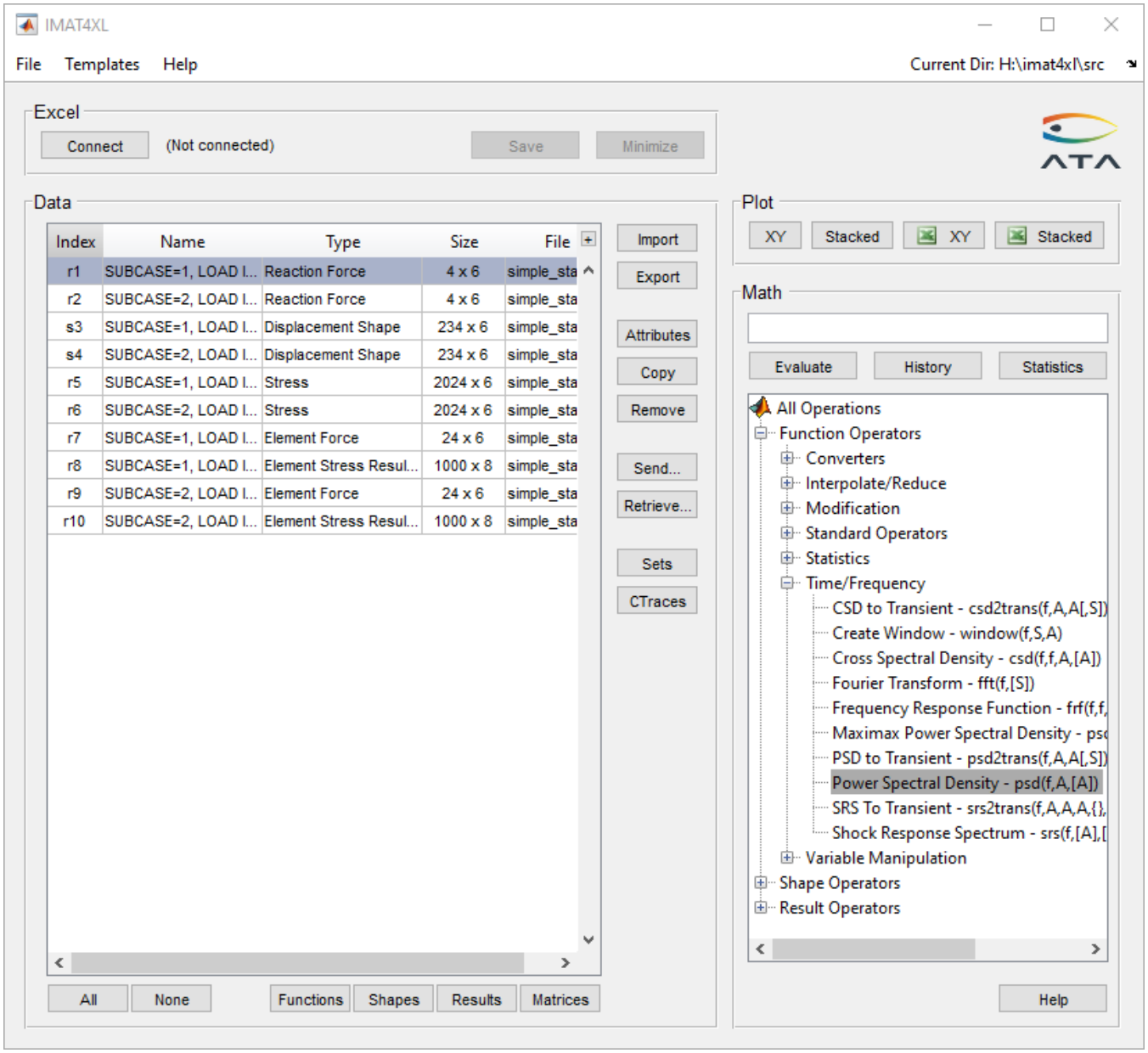
Open IMAT^{4}XL

To open IMAT^{4}XL, select it from the *Start* menu. The application is found under *Start --> ATA Engineering --> IMAT4XL --> IMAT4XL 7.0.0*. After a short wait, the following window will appear.



Import analysis file with stresses

To read in the stresses, click the **Import** button and then select the appropriate file. For this example, the file is named *simple_statics.op2*. After reading in that file, the IMAT^{4}XL window will look like this:

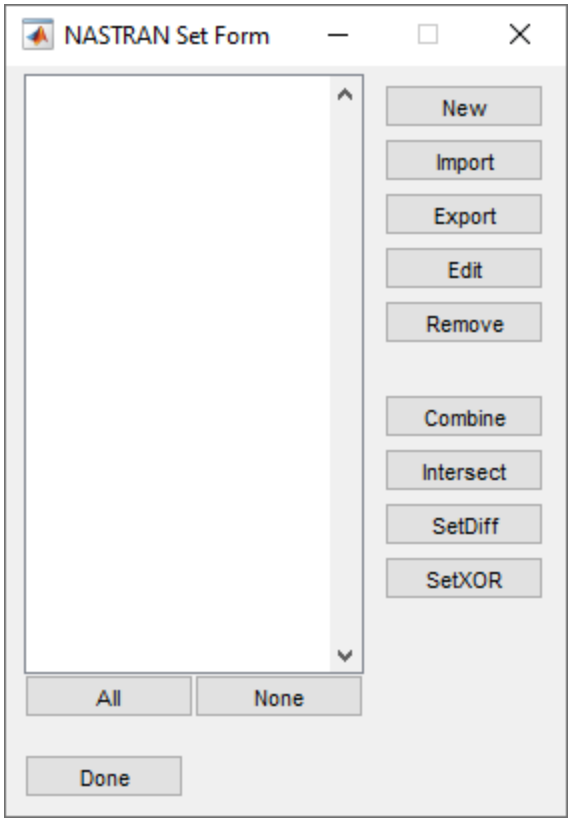


Stress results found in the file are labeled as "Stress" in the "Type" column. For this example, the two stress results are labeled as indexes "r5" and "r6."

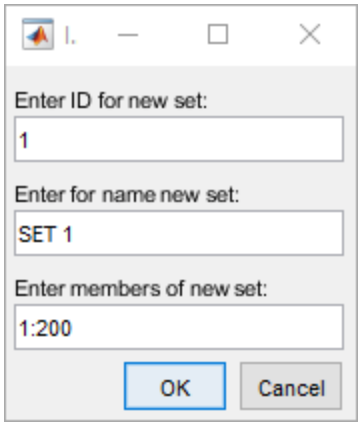
Create set to remove unneeded elements from results

The ability to use sets is an important feature in IMAT^{4}XL. A set is used when a result needs to be pared down to a smaller subset of the full result. This can be because the results are too large to view in Excel, or, as in this case, parts of the results are irrelevant to the operation to be performed.

To open up the Sets form, press the **Sets** button on the main IMAT^{4}XL window. The dialog below will be shown. This is the form that is used to create and modify sets.



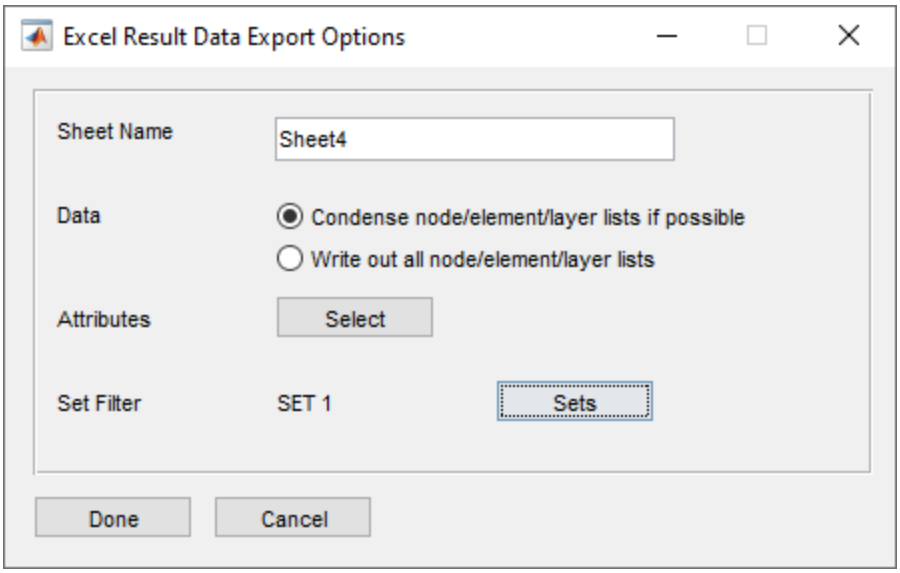
To create a new set, press the **New** button. This will create the form shown below. Default values are automatically entered for the ID and name of the set. Enter "1:200" into the third edit box to add elements 1 through 200 to the set. Press **OK** to save and close the form. "Set 1" will now be added to the set form and will be shown in the listbox. Press **Done** to save the changes.



Send stresses to Excel

To send the stresses to an Excel worksheet, first press the **Connect** button at the top of the window. This will bring up a dialog box asking whether a new Excel workbook should be created, or if an existing one should be connected. Select **New**.

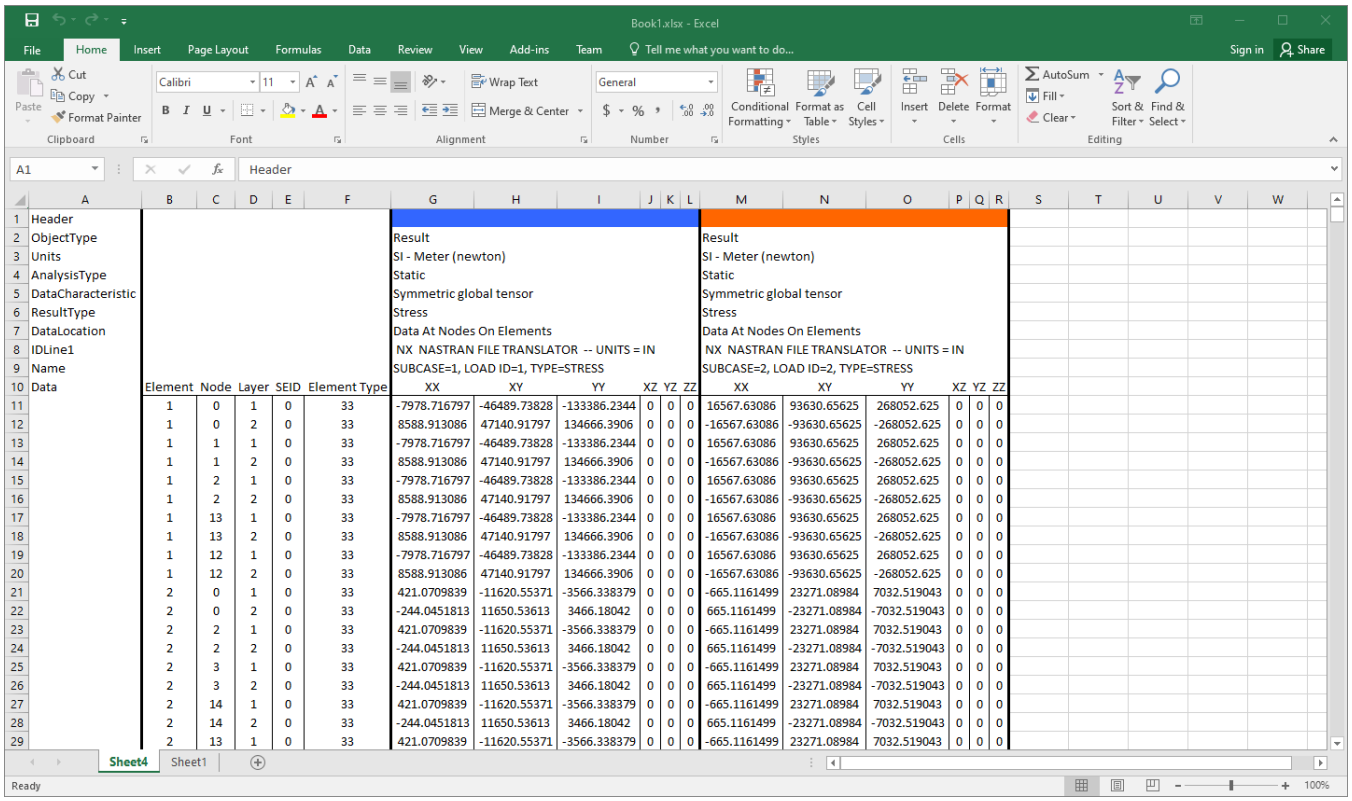
Next, select the stress results from the table and press the **Send** button. This will bring up the dialog box shown below.



The Sheet Name edit box controls the name of the sheet in which the stress data will be placed. It must be unique. The Data radio buttons control whether the nodes, elements, and layer information will be written out for each stress result, or only written out once per worksheet. The **Attributes** button selects which header information will be included on the worksheet. The **Sets** button brings up the set dialog box, which is used to pare down data with NASTRAN sets before sending to Excel.

To use the set that was created earlier, press the **Sets** button. On the resulting dialog box, select "Set 1" from the listbox and press **Done**. The text on the Export Options dialog will be updated to show that "Set 1" will be used as the set filter.

After pressing **Done**, the worksheet will be populated with the pared-down data. The result is shown below.



	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R
1	Header																	
2	ObjectType						Result						Result					
3	Units						SI - Meter (newton)						SI - Meter (newton)					
4	AnalysisType						Static						Static					
5	DataCharacteristic						Symmetric global tensor						Symmetric global tensor					
6	ResultType						Stress						Stress					
7	DataLocation						Data At Nodes On Elements						Data At Nodes On Elements					
8	IDLine1						NX NASTRAN FILE TRANSLATOR -- UNITS = IN						NX NASTRAN FILE TRANSLATOR -- UNITS = IN					
9	Name						SUBCASE=1, LOAD ID=1, TYPE=STRESS						SUBCASE=2, LOAD ID=2, TYPE=STRESS					
10	Data	Element	Node	Layer	SEID	Element Type	XX	XY	YY	XZ	YZ	ZZ	XX	XY	YY	XZ	YZ	ZZ
11		1	0	1	0	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0
12		1	0	2	0	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0
13		1	1	1	0	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0
14		1	1	2	0	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0
15		1	2	1	0	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0
16		1	2	2	0	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0
17		1	13	1	0	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0
18		1	13	2	0	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0
19		1	12	1	0	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0
20		1	12	2	0	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0
21		2	0	1	0	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0
22		2	0	2	0	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0
23		2	2	1	0	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0
24		2	2	2	0	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0
25		2	3	1	0	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0
26		2	3	2	0	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0
27		2	14	1	0	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0
28		2	14	2	0	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0
29		2	13	1	0	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0

Calculate failure margins in Excel

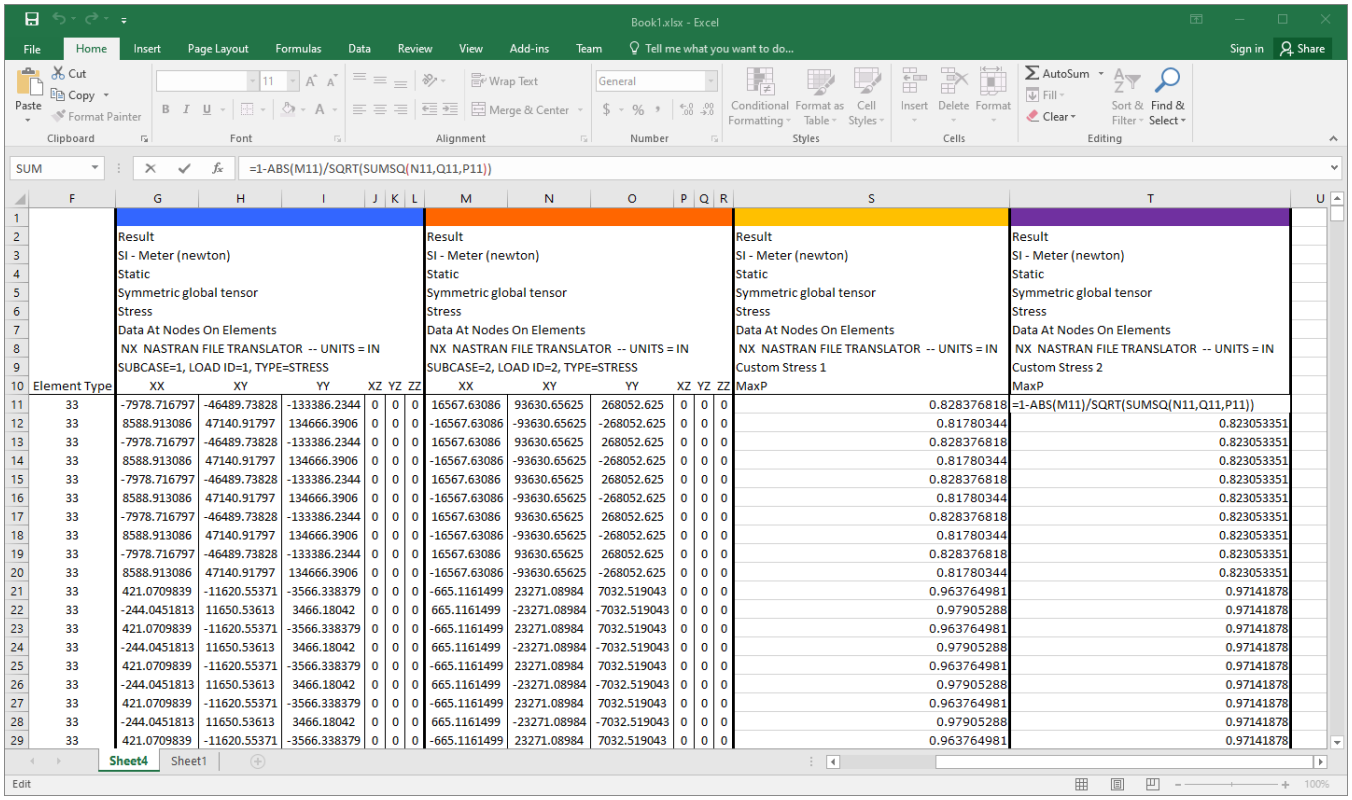
Now that the stress data is in an accessible form, Excel operations can be used to easily calculate the custom failure index by writing the formula once for a single element, and then "pulling down" for all of the other elements. In this case, the correct formula is:

=1-ABS(G11)/SQRT(SUMSQ(H11,K11,J11))

To create the new stress result in which to store the failure margin, simply copy and paste the header from the first result to the next available column. This essentially creates a blank result. Since only a single value is needed, one data header is needed. The text for this data header depends on the program that will be viewing the stress contour. For FEMAP, the data header should be labeled as an invariant such as "MaxP", "MinP", etc. For I-Deas and all others (Universal files), the data header should be labeled as "Scalar".

As an aside, this is because I-deas expects invariants to be written as scalar data sets, where FEMAP expects the invariant to be labeled as an invariant, and does not have a "scalar" result object, per se.

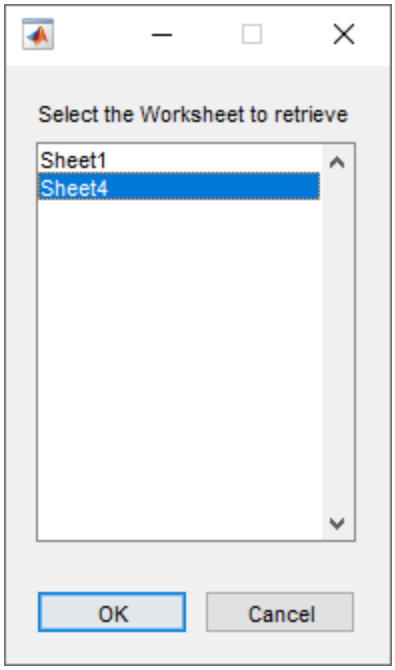
To compute the failure margins, place the formula under “MaxP” (or "Scalar") and pull down to calculate all of the margins.



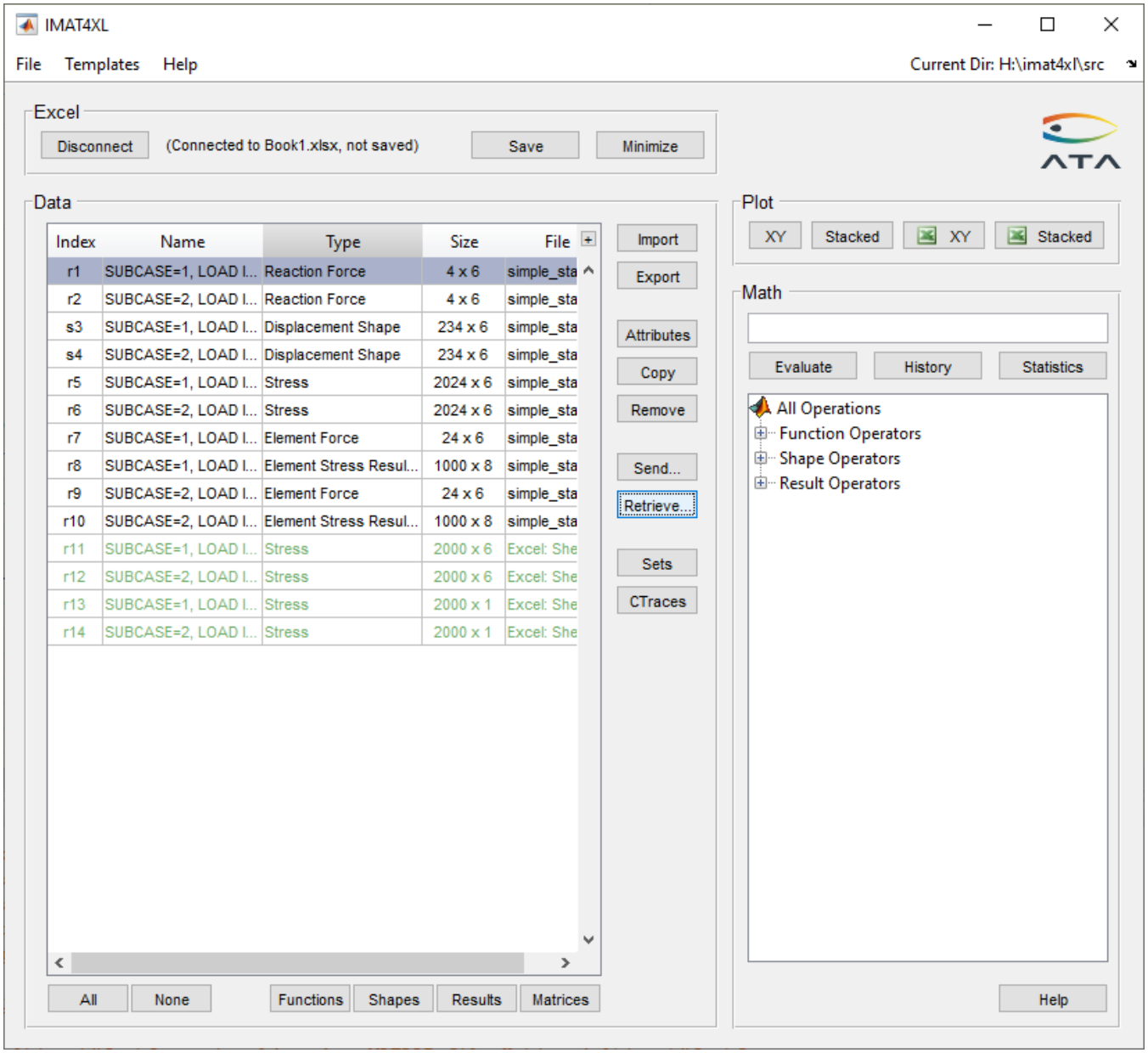
	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U
1																
2		Result						Result						Result	Result	
3		SI - Meter (newton)						SI - Meter (newton)						SI - Meter (newton)	SI - Meter (newton)	
4		Static						Static						Static	Static	
5		Symmetric global tensor						Symmetric global tensor						Symmetric global tensor	Symmetric global tensor	
6		Stress						Stress						Stress	Stress	
7		Data At Nodes On Elements						Data At Nodes On Elements						Data At Nodes On Elements	Data At Nodes On Elements	
8		NX NASTRAN FILE TRANSLATOR -- UNITS = IN						NX NASTRAN FILE TRANSLATOR -- UNITS = IN						NX NASTRAN FILE TRANSLATOR -- UNITS = IN	NX NASTRAN FILE TRANSLATOR -- UNITS = IN	
9		SUBCASE=1, LOAD ID=1, TYPE=STRESS						SUBCASE=2, LOAD ID=2, TYPE=STRESS						Custom Stress 1	Custom Stress 2	
10	Element Type	XX	XY	YY	XZ	YZ	ZZ	XX	XY	YY	XZ	YZ	ZZ	MaxP	MaxP	
11	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0	0.828376818	=1-ABS(M11)/SQRT(SUMSQ(N11,Q11,P11))	
12	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0	0.81780344	0.823053351	
13	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0	0.828376818	0.823053351	
14	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0	0.81780344	0.823053351	
15	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0	0.828376818	0.823053351	
16	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0	0.81780344	0.823053351	
17	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0	0.828376818	0.823053351	
18	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0	0.81780344	0.823053351	
19	33	-7978.716797	-46489.73828	-133386.2344	0	0	0	16567.63086	93630.65625	268052.625	0	0	0	0.828376818	0.823053351	
20	33	8588.913086	47140.91797	134666.3906	0	0	0	-16567.63086	-93630.65625	-268052.625	0	0	0	0.81780344	0.823053351	
21	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0	0.963764981	0.97141878	
22	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0	0.97905288	0.97141878	
23	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0	0.963764981	0.97141878	
24	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0	0.97905288	0.97141878	
25	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0	0.963764981	0.97141878	
26	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0	0.97905288	0.97141878	
27	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0	0.963764981	0.97141878	
28	33	-244.0451813	11650.53613	3466.18042	0	0	0	665.1161499	-23271.08984	-7032.519043	0	0	0	0.97905288	0.97141878	
29	33	421.0709839	-11620.55371	-3566.338379	0	0	0	-665.1161499	23271.08984	7032.519043	0	0	0	0.963764981	0.97141878	

Import Excel worksheet back into IMAT^{4}XL

Now that the failure index has been calculated, press the **Retrieve** button. This will bring up the dialog box shown below.



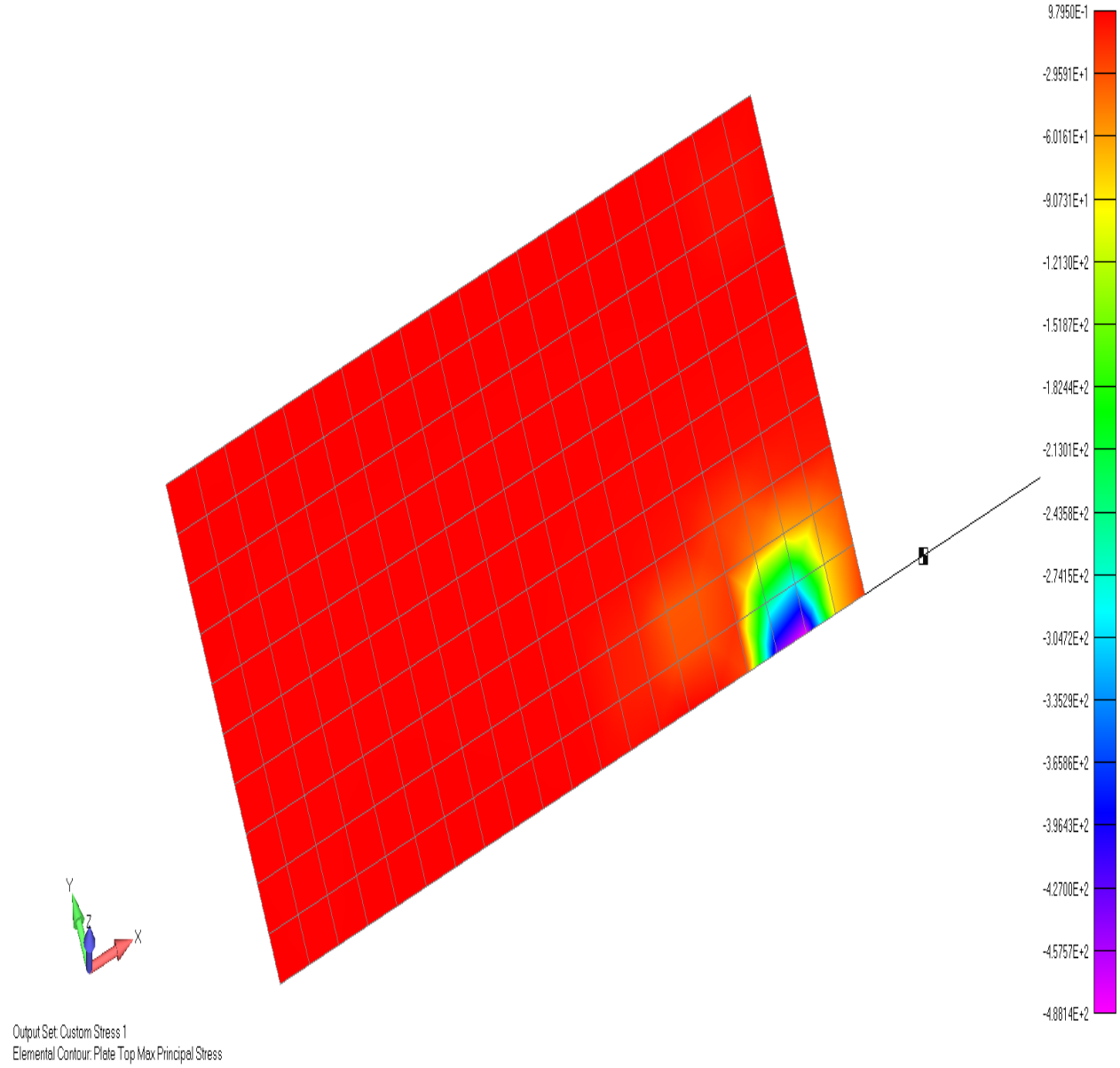
To retrieve the stress data on "Sheet4", select "Sheet4" from the listbox and press **Done**. The data will be retrieved and will appear in the data table as shown below. The newly retrieved stress data will appear in green, signifying that they were retrieved from Excel. The filename column indicates that they originated from "Sheet4".



Export new stress results to an analysis file

To export the newly created stress results with the calculated failure indexes to an analysis file, select them from the data table and press **Export**. This will bring up a file dialog where the filename and file type can be selected. The stress data will then be exported to that file. The two available file formats for stress data are I-deas Universal File and FEMAP Neutral file.

The resulting contour is shown below. The blue and magenta areas show regions with the most negative margins.



Output Set: Custom Stress 1
Elemental Contour: Plate Top Max Principal Stress