VES-5000 Structural Dynamics Modification

March 7, 2025

Notice

Information in this document is subject to change without notice and does not represent a commitment on the part of Vibrant Technology. Except as otherwise noted, names, companies, and data used in examples, sample outputs, or screen shots, are fictitious and are used solely to illustrate potential applications of the software.

Warranty

Vibrant Technology, Inc. warrants that (a) the software in this product will perform substantially in accordance with the accompanying documentation, for a period of one (1) year from the date of delivery, and that (b) any hardware accompanying the software will be free from defects in materials and workmanship for a period of one (1) year from the date of delivery. During this period, if a defect is reported to Vibrant Technology, replacement software or hardware will be provided to the customer at no cost, excluding delivery charges. Any replacement software will be warranted for the remainder of the original warranty period or thirty (30) days, whichever is longer.

This warranty shall not apply to defects resulting from improper or inadequate maintenance by the customer, customer supplied software or interfacing, unauthorized modification or misuse, operation outside of the environmental specifications for the product, or improper site preparation or maintenance.

If the software does not materially operate as warranted above, the sole remedy of the customer (and the entire liability of Vibrant Technology) shall be the correction or detour of programming errors attributable to Vibrant Technology. The software should not be relied on as the sole basis to solve a problem whose incorrect solution could result in injury to a person or property. If the software is employed in such a manner, it is at the entire risk of the customer, and Vibrant Technology disclaims all liability for such misuse.

NO OTHER WARRANTY IS EXPRESSED OR IMPLIED. VIBRANT TECHNOLOGY SPECIFICALLY MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANT ABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

THE REMEDIES PROVIDED HEREIN ARE THE CUSTOMER'S SOLE AND EXCLUSIVE REMEDIES. VIBRANT TECHNOLOGY SHALL NOT BE LIABLE FOR ANY DIRECT, INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES IN CONNECTION WITH THE FURNISHING, PERFORMANCE, OR USE OF THIS PRODUCT, WHETHER BASED ON CONTRACT, TORT, OR ANY OTHER LEGAL THEORY.

Copyright

Vibrant Technology, Inc.

13275 East Fremont Place
Suite 200
Centennial, CO 80112 USA

phone: (831) 430-9045

fax: (831) 430-9057

E-mail: support@vibetech.com

http://www.vibetech.com

Table of Contents

VES-5000 Structural Dynamics Modification (SDM) 3
- Additional Structure Window Commands 3
- Modeling Structural Dynamics Changes 3

What is SDM? 3
- FEA Objects Used for Modeling Modifications 4

FEA Mass 5
- Using the FEA Assistant to Add Masses to a Model 5

FEA Spring & FEA Damper 5
- Using the FEA Assistant to Add Springs & Dampers to a Model 5

FEA Rod 5
- Using the FEA Assistant to Add Rods to a Model 6

FEA Bar 6
- Using the FEA Assistant to Add Bars to a Model 6

FEA Triangle & FEA Quad 6
- Plate Element Stiffness Multiplier 7
- Using the FEA Assistant to Add FEA Triangles & FEA Quads to a Model 7

FEA Tetra, FEA Prism & FEA Brick 7
- Using the FEA Assistant to Add FEA Tetras, FEA Prisms & FEA Bricks to a Modal 7

What is a Modal Model? 8

Validating a Modal Model 9
- Round Trip SDM Validation 9

SDM | Calculate New Modes 9

SDM | Modal Sensitivity 9
- Upper Spreadsheet 10
- Lower Spreadsheet 11
- Display | Split 11
- Display | Spreadsheets 11
- Display | Bar Graphics 11
- Solution | Calculate 12
- Solution Space 12
  - Solution space Examples 12
- Error Function 13
- Solution Scroll Bar 14
- Solution | Stop Calculation 14
- Solution | Save Mode Shapes 14

Solution | Update Properties 14
SDM | Add Tuned Absorbers 14
Absorber Spring....................14
Before Adding an Absorber....................15
Absorber Calculate New Modes....................15
FEA | FEA Materials List 16
File | Import 16
File | Print Spreadsheet 16
Edit | Add....................16
Edit | Delete 16
FEA | FEA Properties List 17
File | Import 17
File | Print Spreadsheet 17
Edit | Add....................17
Edit | Delete 17
FEA | FEA Objects List 18
FEA | FEA Objects | Objects Menu....................18

VES-5000 Structural Dynamics Modification (SDM)

NOTE: If the **VES-6000 SDM** option is authorized by your MEscope license, the following commands are enabled in MEscope. Execute **Help | License Manager** to verify the Options authorized by your MEscope license.

Additional Structure Window Commands

- SDM | Calculate New Modes
- SDM | Modal Sensitivity
- SDM | Add Tuned Absorbers
- FEA | Materials List
- FEA | Properties List
- FEA | Current Objects List
- FEA | FEA Objects | Objects menu

Modeling Structural Dynamics Changes

Modal analysis is used to characterize and further understand noise & vibration problems in operating machinery and structures.

- There are only two options for reducing the ***resonant vibration*** in a machine or structure,
  - ***Isolate*** the structure from its excitation forces
  - ***Physically modify*** the structure to reduce its resonant vibration levels
- When ***tuned absorbers*** or other ***physical modifications*** such as rib stiffeners are attached to a structure, its ***resonant vibration*** will change
- When ***two or more structural components*** are coupled together, its ***resonant vibration*** will change
- If the ***boundary conditions*** of a structure are changed, its ***resonant vibration*** will change

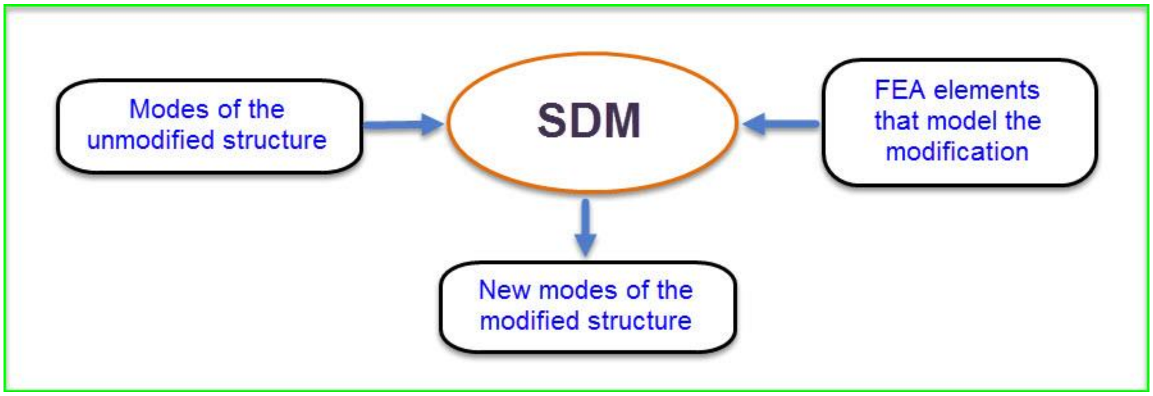
What is SDM?

The **Structural Dynamics Modification (SDM)** method allows you to model ***additions (or subtractions) to physical parts*** to a structure and calculate the new modal parameters that result from those structural modifications.

SDM requires two inputs,

- A ***Modal Model*** of the unmodified structure
- ***Finite elements which model the modifications*** to the structure

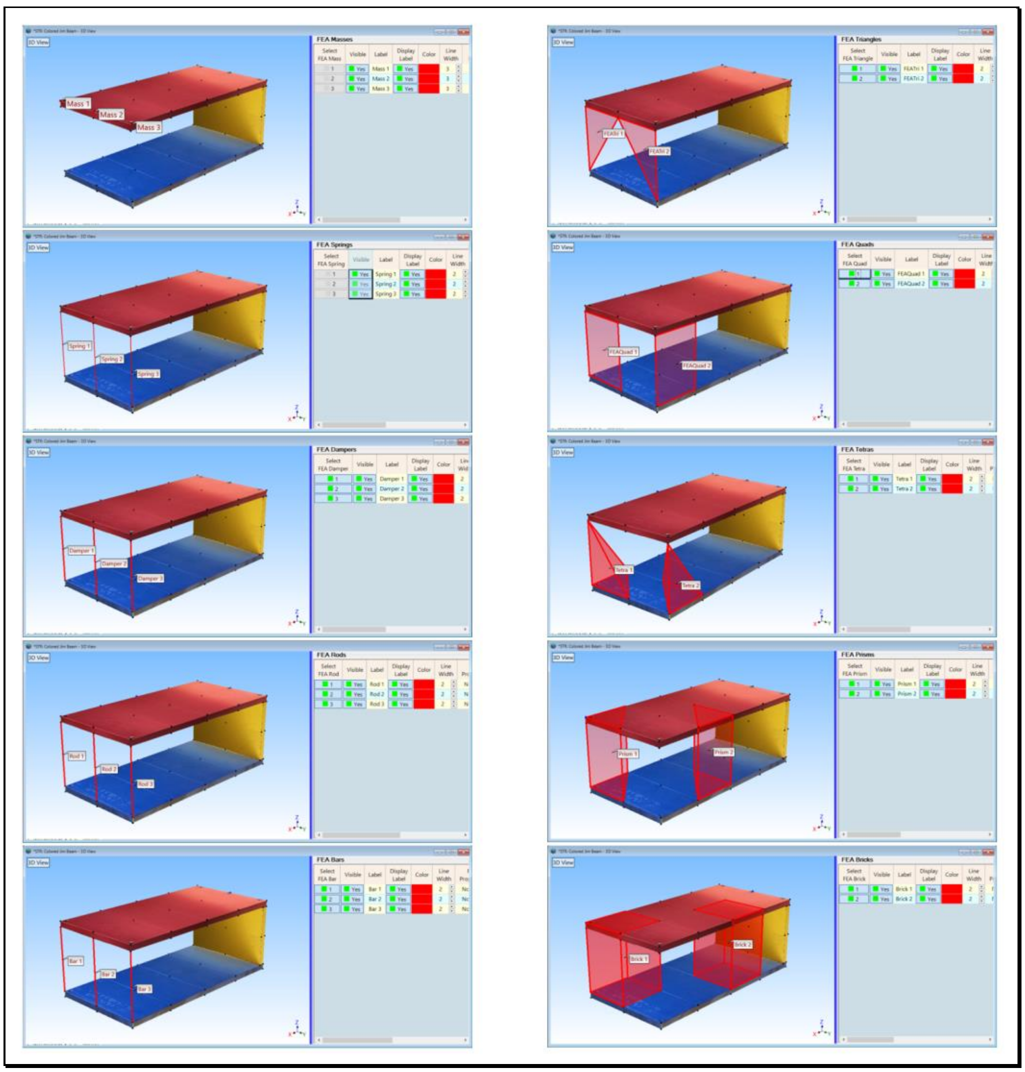
SDM converts all structural modifications internally to ***changes in the mass, stiffness & damping properties*** of the structure. The mass, stiffness, & damping changes are used together with the modes of the unmodified structure to calculate the new modes of the modified structure.



FEA Objects Used for Modeling Modifications

The **SDM** commands use ***industry-standard finite elements*** to model structural modifications. Finite elements are called **FEA Objects** in MEscope.

The following types of FEA Objects can be used for modeling structural modifications.



Types of FEA Objects used by SDM

FEA Mass

An FEA Mass Object ***adds translational*** or ***rotational mass*** (***or inertia***) to ***a Point*** on a structure model.

- Inertia can be constrained to specific directions by making the appropriate selections in the Orientation column of the FEA Masses spreadsheet
    - Mass properties are defined on the Masses tab in the **FEA | FEA Properties** dialog
    - The property of each FEA Mass must be chosen in its **FEA Property cell** in the FEA Masses spreadsheet

Using the FEA Assistant to Add Masses to a Model

Using the FEA Assistant, an FEA Mass is added to ***each selected*** **Point** on a structure model.

- Multiple FEA Masses can be added to multiple Points by creating a Substructure of Points and selecting the Substructure

FEA Spring & FEA Damper

An FEA Spring adds ***linear stiffness between two Points*** on a structure model.

An FEA Damper adds ***linear damping between two Points*** on a structure model.

- Stiffness & damping are applied between two Points either axially (along the element axis), translationally in 3 directions or rotationally in 3 directions.
- Stiffness & damping can be constrained to specific directions by making selections from the Orientation columns of the end Points in the FEA Spring or FEA Damper spreadsheet.
    - Spring stiffness is defined on the Springs tab in the **FEA | FEA Properties** dialog
    - Damping is defined on the Dampers tab in the **FEA | FEA Properties** dialog
    - The stiffness of each FEA Spring must be chosen in its **FEA Property cell** in the FEA Springs spreadsheet
    - The Damping of each FEA Damper must be chosen in its **FEA Property cell** in the FEA Dampers spreadsheet

Using the FEA Assistant to Add Springs & Dampers to a Model

Using the FEA Assistant, an FEA Spring or FEA Damper is added to a structure model wherever there is a ***selected*** **Line.**

- Multiple FEA Springs or FEA Dampers can be added to a model by creating a Substructure of Lines and selecting the Substructure

FEA Rod

An FEA Rod is a ***linear*** element that ***applies translational force*** along its axis ***between two Points*** on a structure model.

- Cross sectional areas of rods are defined on the Rods tab in the **FEA | FEA Properties** dialog
    - Elasticity & density of rods are defined in the **FEA | FEA Materials** dialog
    - The properties of each FEA Rod must be chosen in its **FEA Property cell** in the FEA Rod spreadsheet

Using the FEA Assistant to Add Rods to a Model

Using the FEA Assistant, an FEA Rod is added to a structure model wherever there is a ***selected*** **Line.**

- Multiple FEA Rods can be added to a model by creating a Substructure of Lines and selecting the Substructure

FEA Bar

An FEA Bar is a ***long slender, linear element*** that applies ***translational force*** along its axis between two Points and ***bending force*** to the structure at its two end-Points.

- Cross sectional areas & cross-sectional inertias (X Inertia, Y Inertia, XY Inertia) for bars are defined on the Bars tab in the FEA | FEA Properties dialog
- The cross section is oriented by pointing the X-Axis of the cross section to an **Orientation Point** in the structure window
  - The **Orientation Point** is defined by its row number in the Points spreadsheet
  - The **Orientation Point** row number must be entered into the **Orientation** cell for each FEA Bar in the FEA Bars spreadsheet
  - Material properties for bars are defined in the **FEA | FEA Materials** dialog
  - The properties of each FEA Bar must be chosen in its **FEA Property cell** in the FEA Bars spreadsheet

Using the FEA Assistant to Add Bars to a Model

Using the FEA Assistant, an FEA Bar is added to a structure model wherever there is a ***selected*** **Line.**

- Multiple FEA Bars can be added to a model by creating a Substructure of Lines and selecting the Substructure

FEA Triangle & FEA Quad

FEA Triangles & FEA Quads are ***linear plate elements***, also called ***membrane elements***.

FEA Triangles & FEA Quads should be used to model parts of a structure that are ***relatively thin compared to their width & height dimensions***. Otherwise, solid elements should be used.

- An FEA Triangle is attached between ***three Points*** on the structure model
- An FEA Quad is attached between ***four Points*** on the structure model
  - Plate thicknesses are defined on the Plates tab in the **FEA | FEA Properties** dialog
  - Plate material properties are defined in the **FEA | FEA Materials** dialog
  - The properties of each FEA Triangle must be chosen in its **FEA Property cell** in the FEA Triangles spreadsheet
  - The properties of each FEA Quad must be chosen in its **FEA Property cell** in the FEA Quads spreadsheet

Plate Element Stiffness Multiplier

The Stiffness Multiplier is used to increase or decrease the ***bending stiffness*** of FEA Triangles or FEA Quads.

- The **FEA Triangle** & **FEA Quad** stiffness calculation assumes that the plate ***cross section*** consists of a ***uniform distribution of one material***

In cases where a plate cross section consists of ***two or more dissimilar materials***, its bending stiffness could be greater or less than the stiffness calculated with a single uniform material.

- **Stiffness Multiplier = 1 ➔** bending stiffness is based on thickness & material properties
- **Stiffness Multiplier > 1 ➔** increases the bending stiffness of an FEA Triangle or FEA Quad
- **Stiffness Multiplier < 1 ➔** decreases the bending stiffness of an FEA Triangle or FEA Quad

Using the FEA Assistant to Add FEA Triangles & FEA Quads to a Model

Using the FEA Assistant, an FEA Triangle is added to a model wherever there is a ***selected*** **Surface Triangle**.

Using the FEA Assistant, an FEA Quad is added to a model wherever there is a ***selected*** **Surface Quad**.

- Multiple FEA Triangles can be added to a model by creating a **Substructure of Surface Triangles** and selecting the Substructure
- Multiple FEA Quads can be added to a model by creating a **Substructure of Surface Quads** and selecting the Substructure

FEA Tetra, FEA Prism & FEA Brick

FEA Tetras, FEA Prisms & FEA Bricks are ***3-dimensional solid finite elements***.

FEA Tetras, FEA Prisms & FEA Bricks should be used to model parts of a structure that ***have approximately the same width, height, & length dimensions***.

- An **FEA Tetra** is attached between ***four Points*** on the structure model
- An **FEA Prism** is attached between ***six Points*** on the structure model
- An **FEA Brick** is attached between ***eight Points*** on the structure model
  - Tetra, Prism & Brick properties are defined on the Solids tab in the **FEA | FEA Properties** dialog
  - Each FEA Tetra, FEA Prism & FEA Brick must have its property chosen in the **FEA Property cell** in the FEA Tetra, FEA Prism or FEA Brick spreadsheet
  - Tetra, Prism & Brick material properties are defined in the **FEA | FEA Materials** dialog

Using the FEA Assistant to Add FEA Tetras, FEA Prisms & FEA Bricks to a Modal

Using the FEA Assistant, multiple FEA Tetras, FEA Prisms & FEA Bricks can be added to a selected Substructures.

- FEA Tetras, FEA Prisms & FEA Bricks can be added to any ***selected*** **3D Editable Substructure**
- FEA Tetras, FEA Prisms & FEA Bricks can be added to any ***selected*** **2D Substructure** that has been **Extruded** or **Revolved** into a 3D Substructure

The FEA Objects require the following physical properties.

FEA Object	Physical Properties
Mass	Mass
Spring	Stiffness
Damper	Damping
Rod	cross sectional Area Elasticity, Density
Bar	cross sectional Area X inertia, Y inertia, XY inertia Elasticity, Density
Triangular Plate	Thickness, Elasticity Poisson's Ratio, Density
Quad Plate	Thickness, Elasticity Poisson's Ratio, Density
Tetrahedron	Elasticity, Poisson's Ratio, Density
Prism	Elasticity, Poisson's Ratio, Density
Brick	Elasticity, Poisson's Ratio, Density

What is a Modal Model?

A Modal Model is a set of mode shapes that has been scaled to preserve the mass, stiffness & damping properties of a structure.

- MEscope uses two different Modal Models, **Residue** mode shapes and **UMM** mode shapes
- **Residue** mode shapes are obtained by curve fitting a set of ***calibrated FRFs***
  - **Residue** mode shapes can also be obtained by re-scaling **UMM** mode shapes
  - *see* **Tools | Scaling | UMM to Residue Shapes** in the Data Block (BLK) chapter
- **UMM** mode shapes are obtained by re-scaling **Residue** mode shapes
  - *see* **Tools | Scaling | Residues to UMM Shapes** in the Data Block (BLK) chapter
- **UMM** mode shapes are also obtained by solving for the modes of an FEA model

Validating a Modal Model

A **Modal Model** of the ***unmodified structure*** must be validated before using it with SDM commands.

There are several ways to validate a Modal Model

- Synthesize FRFs using the mode shapes of the Modal Model and compare the synthesized and experimental FRFs using the **Tools | Data Block Correlation** or **Tools | M# Pairs Correlation** command
- Calculate the **Participation** of each mode in an FEA Modal Model with each mode of an EMA or OMA modal model
    - If each EMA or OMA mode shape is ***scaled to Unit Modal Masses (UMM)***, the Participation of each FEA shape in each EMA or OMA shape ***will be close to "1"***
- Perform a ***round trip*** SDM validation, explained below

Round Trip SDM Validation

- Add an **FEA Mass** to the structure model
- Execute **SDM | Calculate New Modes**, and choose the Modal Model of the ***unmodified structure***
- Save the new modes calculated by SDM in a Shape Table named ***"new mode shapes"***
- In the **FEA Properties** list, change the FEA Mass value to ***"minus" the previous value***
- Execute **SDM | Calculate New Modes** again, and choose **SHP: new mode shapes** as the Modal Model of the ***unmodified structure***
- Execute **Display | SDI** and compare the ***original*** **UMM** mode shapes with the **UMM** mode shapes from the previous step
    - All SDI values should be ***close to "1"***

SDM | Calculate New Modes

Calculates new modes for a structure using the **SDM** method

- The following are required before using this command,
    - A Shape Table containing a **Modal Model** of the ***unmodified structure***
    - **FEA Objects** attached to the structure that ***model the structural modifications***
    - The Force & Length units on the **Units** tab in the **File | Structure Options** box ***must match*** the Force & Length units of the Modal Model
    - The mode shapes of the **Modal Model** ***must animate correctly*** on the structure model

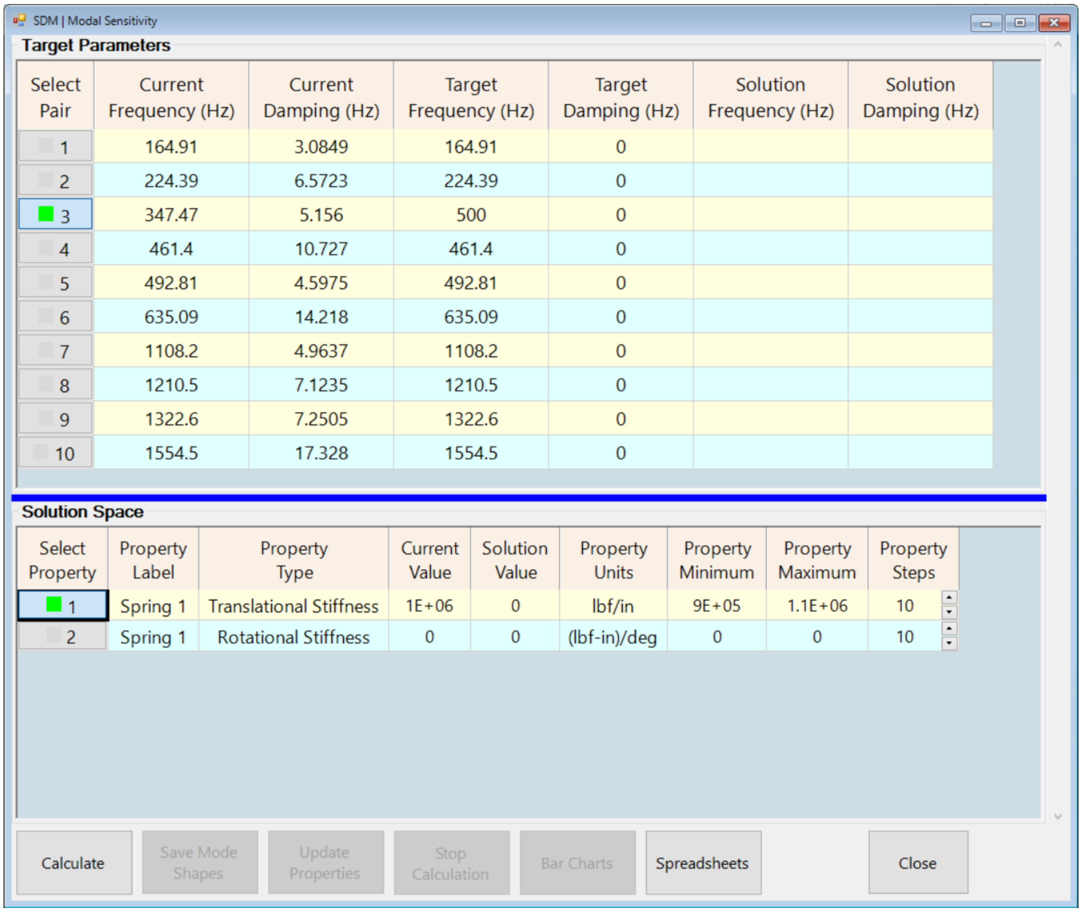
SDM | Modal Sensitivity

Calculates ***how much modification*** is necessary to change the modal properties of a structure so that they ***match target modal parameter values***.

- Multiple SDM solutions are calculated over a ***user-defined Solution space*** of FEA Object properties
    - The ***Solution space*** is defined by the **Minimum, Maximum** and **Number of Steps** of each FEA property to be changed
    - The solutions are ordered from ***best to worst***
    - The ***best solution*** is the one that ***minimizes the percent difference*** between Target & Solution modal parameters

Calculation all SDM solutions in a ***user-defined Solution space*** guarantees that the ***best solution*** will always be found.

- When this command is executed, the Shape Table selection dialog box will open from which to choose a Shape Table (SHP) file containing the **Modal Model** of the ***unmodified structure***
- The **Modal Sensitivity** window will open next, displaying two spreadsheets



Modal Sensitivity Spreadsheets (in Upper Lower Display format).

Upper Spreadsheet

The current, target, & solution modal parameters are listed in the upper spreadsheet

- **Current** parameters are from the ***unmodified structure***
- **Target** parameters are ***user-defined values***
- **Solution Frequency** & **Solution Damping** are the results of ***the SDM calculations***
- If ***one or more* Pairs** are ***selected***, then only the **Solution** parameters are compared with the **Target** parameters for those modes to ***order the solutions from best to worst***

If ***no* Target** parameters are entered, the **Solution** parameters are listed in the order they are calculated.

Lower Spreadsheet

The **Current** & **Solution** FEA properties are listed in the ***lower spreadsheet***.

The following Table lists the FEA properties that can be used to ***define a Solution space*** for Modal Sensitivity.

FEA Object	Property	Target Parameter
Spring	stiffness	frequency
Damper	damping	damping
Mass	mass	frequency, damping
Rod	cross sectional area, (Elasticity, Poisson's ratio, Density)	frequency, damping
Bar	cross sectional area & inertias, (Elasticity, Poisson's ratio, Density)	frequency, damping
Plate	thickness, (Elasticity, Poisson's ratio, Density)	frequency, damping
Solid	(Elasticity, Poisson's ratio, Density)	frequency, damping

FEA Properties and Target Parameters.

Display | Split

Toggles the spreadsheet display between ***Upper Lower*** and ***Left Right*** format.

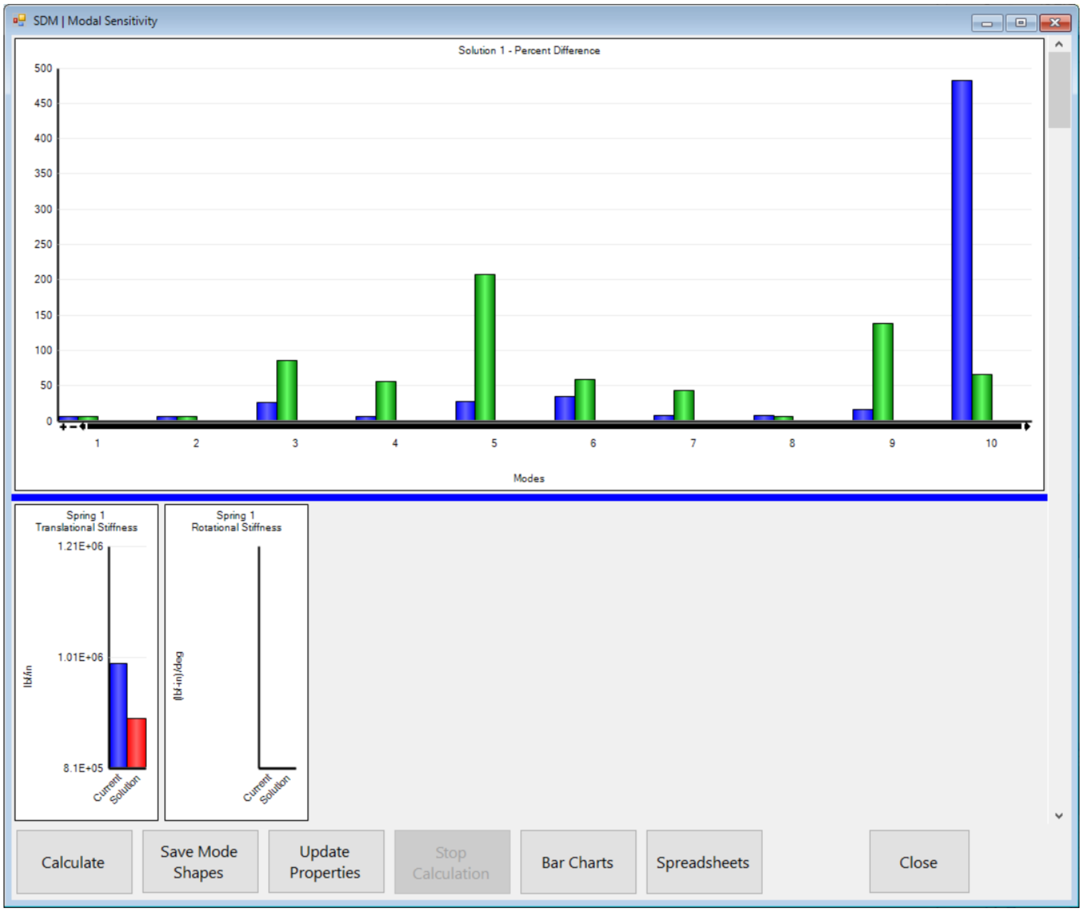
Display | Spreadsheets

When ***checked***, the ***solutions are displayed in* spreadsheets**.

Display | Bar Graphics

When ***checked***, the ***solutions are displayed using bar graphics***, as shown below.

- The ***upper-left*** bar graph shows the ***percent difference*** between the **Solution** & **Target** parameters
- The ***lower*** bar graph displays the **Current** & **Solution** values of ***all*** (or ***selected***) FEA properties
- ***Hover*** the mouse pointer over ***each bar*** on a bar graph to display its values



Solution Bar Chart Display

Solution | Calculate

Calculates SDM solutions using ***all*** (or ***selected***) **FEA Properties** in the ***Solution space*** defined in the ***lower spreadsheet.***

Solution Space

SDM solutions are calculated for ***all values*** in the ***user-defined Solution space***.

The **Minimum**, **Maximum**, & **Steps** properties ***define the Solution space***.

- The **Minimum**, **Maximum**, & **Steps** of each FEA property must be entered into a row in the ***lower spreadsheet***
    - The **Minimum** & **Maximum** values define the ***lower & upper bounds*** of a FEA property
    - **Steps** is the ***number of values*** between the **Minimum & Maximum** values to be used by SDM to calculate Solutions
    - The ***total number of SDM solutions*** is the ***product of the*** **Steps** of ***all*** (or ***selected***) FEA properties

Solution space Examples

- If **Minimum = 1**, **Maximum = 10** and **Steps = 10**, then **10 Solutions** are calculated using property values (**1, 2, 3, 4, 5, 6, 7, 8, 9, 10**)
- If **three FEA properties** each have **10 Steps**, then **1000 Solutions** are calculated

Error Function

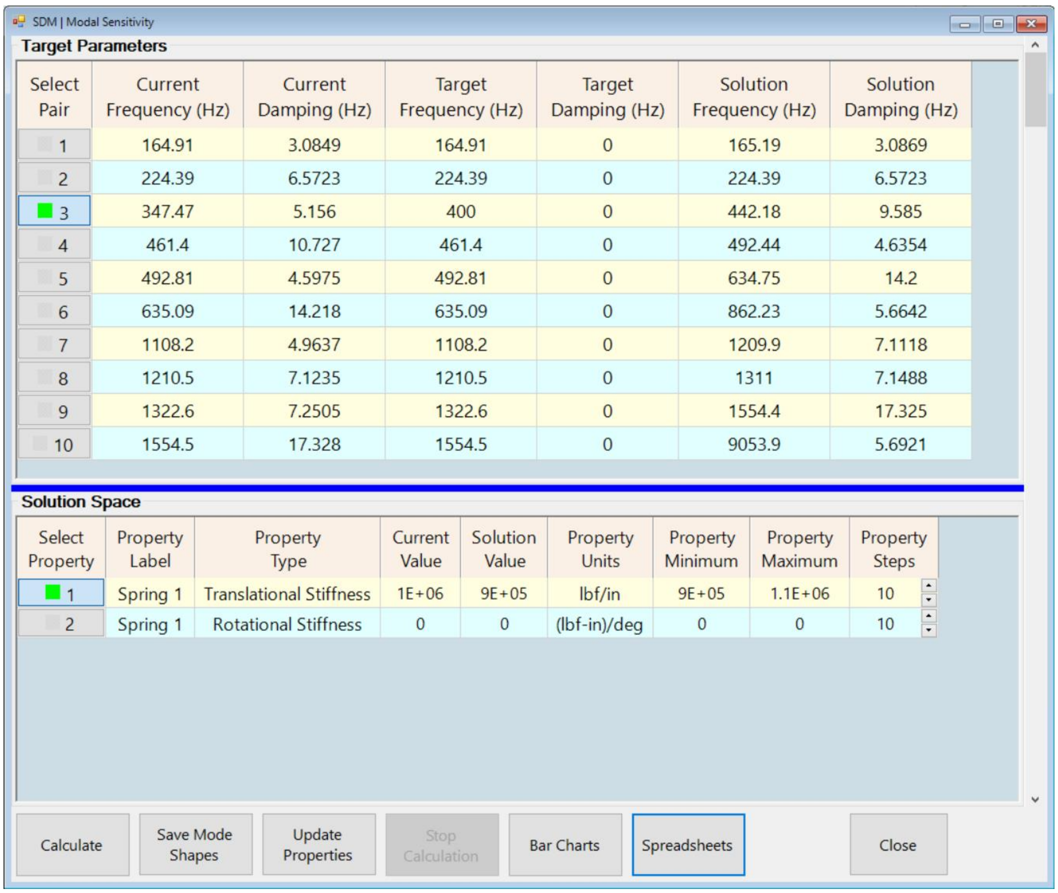
If mode **Pairs** are ***selected*** in the ***upper spreadsheet***, an **Error function** is calculated using the **Target & Solution** parameters for those Pairs

$$\text{Error} = \sum_{\text{modes}} \left[\frac{\|Fs - Ft\|}{Ft} + \frac{\|Ds - Dt\|}{Dt} \right]$$

Fs = **Solution** frequency, Ft = **target** frequency

Ds = **Solution** damping, Dt = **target** damping

- All Solutions are ordered according to the value of the Error function
  - The ***best*** solution has the ***minimum* Error value**
  - The ***worst*** solution has the ***maximum* Error value**



Select Pair	Current Frequency (Hz)	Current Damping (Hz)	Target Frequency (Hz)	Target Damping (Hz)	Solution Frequency (Hz)	Solution Damping (Hz)
1	164.91	3.0849	164.91	0	165.19	3.0869
2	224.39	6.5723	224.39	0	224.39	6.5723
3	347.47	5.156	400	0	442.18	9.585
4	461.4	10.727	461.4	0	492.44	4.6354
5	492.81	4.5975	492.81	0	634.75	14.2
6	635.09	14.218	635.09	0	862.23	5.6642
7	1108.2	4.9637	1108.2	0	1209.9	7.1118
8	1210.5	7.1235	1210.5	0	1311	7.1488
9	1322.6	7.2505	1322.6	0	1554.4	17.325
10	1554.5	17.328	1554.5	0	9053.9	5.6921

Select Property	Property Label	Property Type	Current Value	Solution Value	Property Units	Property Minimum	Property Maximum	Property Steps
1	Spring 1	Translational Stiffness	1E+06	9E+05	lbf/in	9E+05	1.1E+06	10
2	Spring 1	Rotational Stiffness	0	0	(lbf-in)/deg	0	0	10

Best Solution Shown in Both Spreadsheets.

Solution Scroll Bar

When the SDM calculations are completed or stopped, a **Solution Scroll Ba**r is displayed on the ***right side*** of the Modal Sensitivity window.

- Solutions are displayed in the ***upper*** & ***lower*** **spreadsheets** by scrolling the Solution Scroll Bar
  - The ***best*** **solution** is displayed when the scroller is ***at the top*** of the **Solution Scroll Bar**
  - The ***worst*** **solution** is displayed when the scroller is ***at the bottom*** of the **Solution Scroll Bar**
  - If ***no mode Pairs are selected***, the solutions are displayed in the order they were calculated

Solution | Stop Calculation

Stops the SDM solution calculations and displays the solutions that have already been calculated.

Solution | Save Mode Shapes

Saves the mode shapes from the ***current*** **Solution**.

- ***Scroll*** to the desired solution before executing this command.

Solution | Update Properties

Updates the FEA Properties List with properties from the ***current*** **Solution**.

SDM | Add Tuned Absorbers

Models the addition of ***one or more tuned vibration absorbers*** to a structure.

- A tuned absorber is used to ***suppress the amplitude*** of a resonance
- If the frequency of the tuned absorber is ***close to*** the frequency of a structural resonance, the tuned absorber will ***"split"*** the original resonance into ***two*** resonances with ***lower*** amplitudes
- SDM calculates the new modes that result from attaching tuned absorbers to a structure
- SDM is used to ***attach each tuned absorber mass*** to the structure ***using a spring***
- A tuned absorber is defined by the following parameters,
  - Absorber **mass** & direction of motion
  - Absorber **frequency**, chosen ***"close to"*** the frequency of the mode ***to be suppressed***
  - Absorber **damping**, as a ***percent of critical damping***, is optional

Absorber Spring

The tuned absorber **FEA mass** is ***attached to a Point & direction*** of a structure model using an **FEA spring**.

- The **FEA spring stiffness** is calculated using the formula,

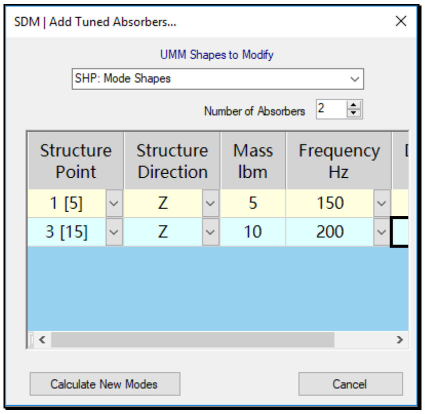
$$\textbf{Absorber spring stiffness} = \textbf{(Absorber Mass) x (Absorber Frequency)}^2$$

Before Adding an Absorber

The following items are required before using this command,

- A Shape Table (SHP) containing a **UMM Modal Model** of the ***unmodified structure***
- The mode shapes of the **UMM Modal Model *must animate correctly*** on the structure model

When this command is executed, the following dialog box will open



Add Tuned Absorbers Dialog Box.

- Select the Shape Table (SHP) containing the **UMM Modal Model** of the ***unmodified structure***
- Select the **Number of Absorbers** to be attached to the structure
- For each Absorber,
  - Select a **Structure DOF** for attaching the Absorber
  - Enter a **Mass** value into the spreadsheet
  - Enter a **Frequency** value into the spreadsheet
  - Enter a **Damping** value into the spreadsheet (optional)

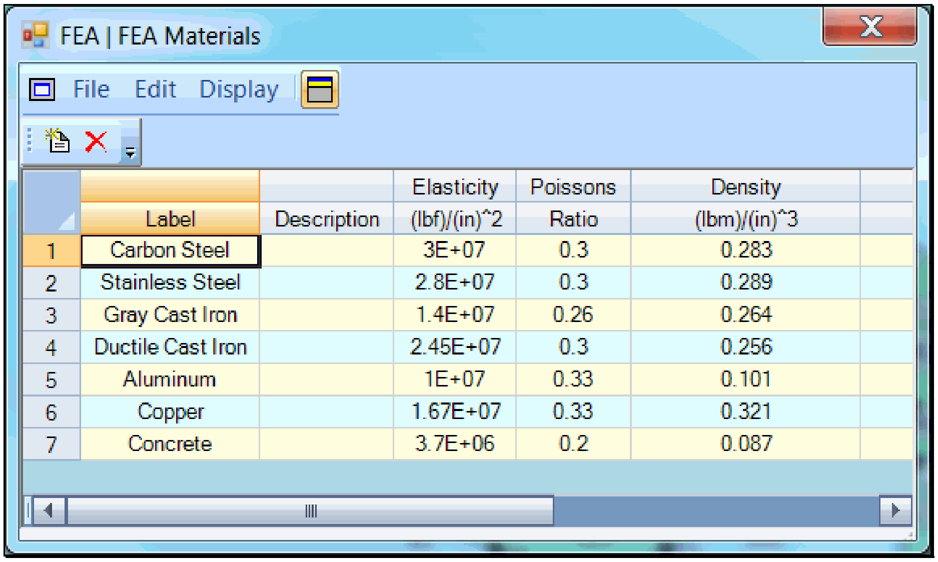
Absorber Calculate New Modes

- ***Press* Calculate New Modes** to calculate the new modes with the Absorbers attached
- The Shape Table selection box will open allowing you to ***save the new mode shapes*** with the Tuned Absorbers attached to the structure

FEA | FEA Materials List

Opens the FEA Materials List window.

- The FEA Materials List contains ***material properties*** that are ***referenced*** by elements in the **FEA Properties** window
- Material units are chosen on the **Units** tab in the **File | Structure Options** box
- The FEA Materials List is saved in the ***currently open*** **Project** file



	Label	Description	Elasticity (lbf)/(in)^2	Poissons Ratio	Density (lbm)/(in)^3
1	Carbon Steel		3E+07	0.3	0.283
2	Stainless Steel		2.8E+07	0.3	0.289
3	Gray Cast Iron		1.4E+07	0.26	0.264
4	Ductile Cast Iron		2.45E+07	0.3	0.256
5	Aluminum		1E+07	0.33	0.101
6	Copper		1.67E+07	0.33	0.321
7	Concrete		3.7E+06	0.2	0.087

FEA Materials Window.

File | Import

Imports a Materials spreadsheet from another MEscope Project file.

File | Print Spreadsheet

Prints the Materials spreadsheet on the system printer or into a PDF file.

Edit | Add

Adds a new Material to the Materials List.

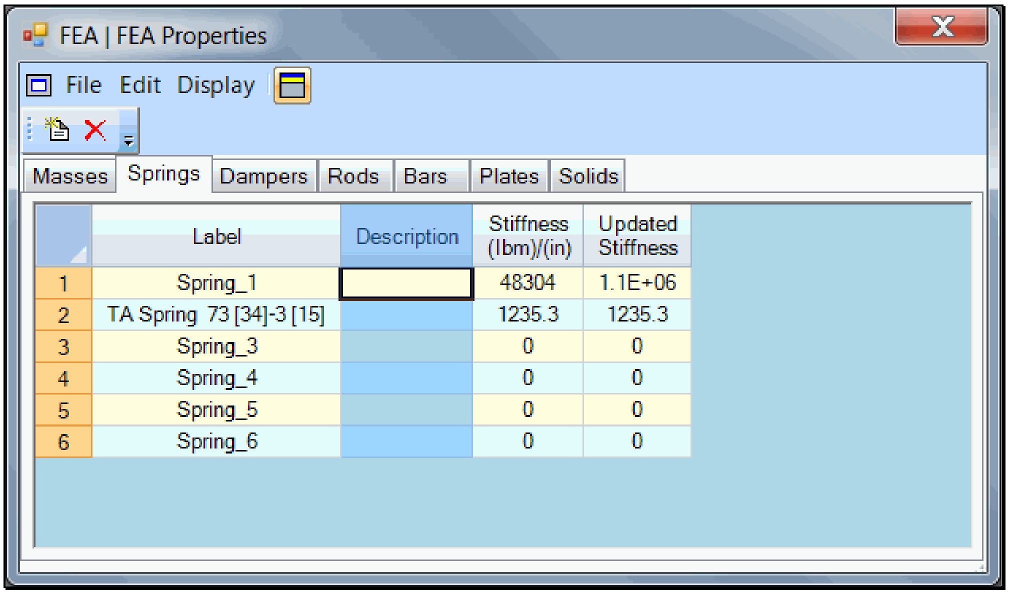
Edit | Delete

Deletes the currently selected Material from the Materials List.

FEA | FEA Properties List

Opens the FEA Properties List window.

- The FEA Properties List contains properties that are ***referenced* by *each FEA Object*** in the Objects spreadsheets
- FEA Properties used for **Rods, Bars, Plates, & Solids** also reference properties from the **FEA Materials List**
- The FEA Properties List is saved in the ***currently open* Project** file



FEA Properties List Window.

- Each **FEA Mass** requires a ***mass*** property
- Each **FEA Spring** requires a ***stiffness*** property
- Each **FEA Damper** requires a ***damping*** property
- Each **FEA Rod** requires a ***cross sectional area*** and a ***material*** from the **FEA Materials List**
- Each **FEA Bar** requires a ***cross sectional area***, ***inertias,*** and a ***material*** from the **FEA Materials List**
- Each **FEA Plate** requires a ***thickness*** and a ***material*** from the **FEA Materials List**
- Each **FEA Solid** requires a ***material*** from the **FEA Materials List**

File | Import

Imports a **FEA Properties List** from another MEscope Project file.

File | Print Spreadsheet

Prints the **FEA Properties List** on the system printer or a PDF file.

Edit | Add

Adds a ***new*** Property to the **FEA Properties List**.

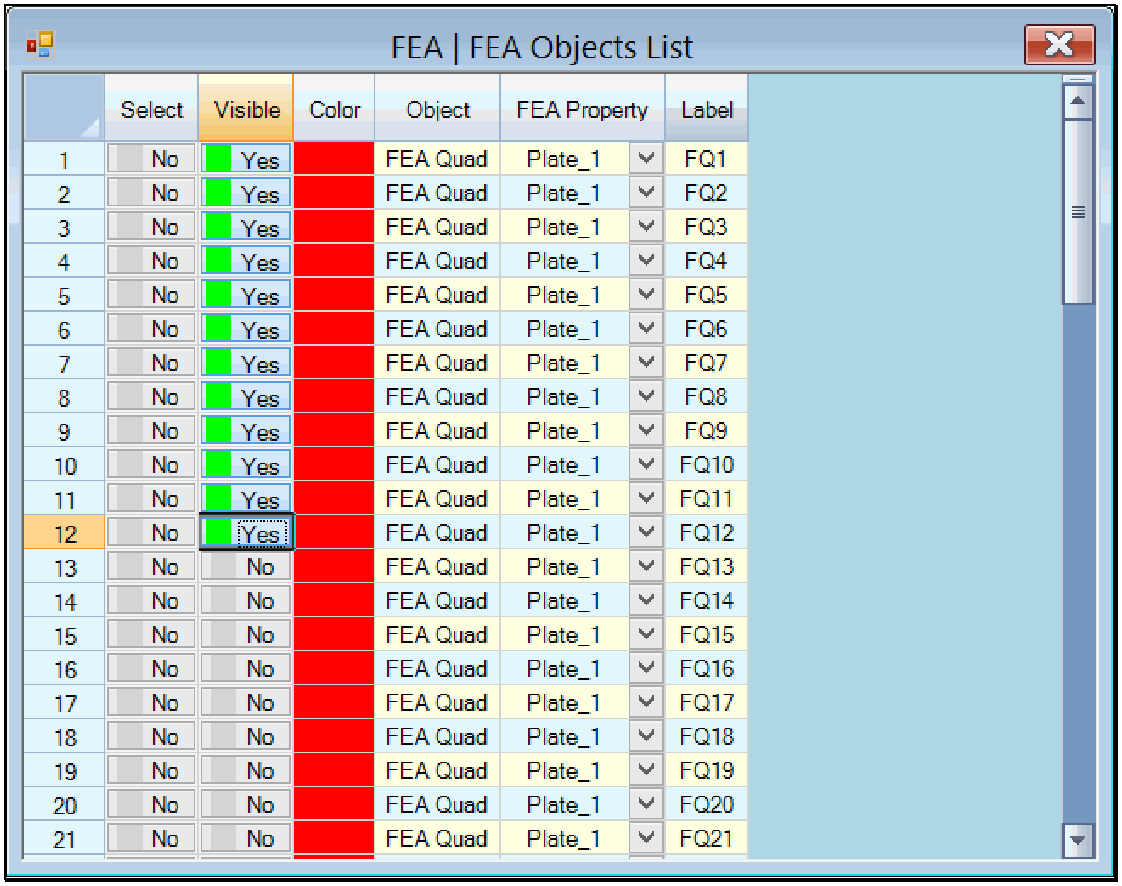
Edit | Delete

Deletes the *currently selected* Property from the FEA Properties List.

FEA | FEA Objects List

Opens the FEA Objects window.

- ***All FEA Objects*** currently attached to the structure model are listed
- Only ***visible* FEA Objects** are used by the **SDM** & **FEA** commands


FEA | FEA Objects List

	Select	Visible	Color	Object	FEA Property	Label
1	No	Yes		FEA Quad	Plate_1	FQ1
2	No	Yes		FEA Quad	Plate_1	FQ2
3	No	Yes		FEA Quad	Plate_1	FQ3
4	No	Yes		FEA Quad	Plate_1	FQ4
5	No	Yes		FEA Quad	Plate_1	FQ5
6	No	Yes		FEA Quad	Plate_1	FQ6
7	No	Yes		FEA Quad	Plate_1	FQ7
8	No	Yes		FEA Quad	Plate_1	FQ8
9	No	Yes		FEA Quad	Plate_1	FQ9
10	No	Yes		FEA Quad	Plate_1	FQ10
11	No	Yes		FEA Quad	Plate_1	FQ11
12	No	Yes		FEA Quad	Plate_1	FQ12
13	No	No		FEA Quad	Plate_1	FQ13
14	No	No		FEA Quad	Plate_1	FQ14
15	No	No		FEA Quad	Plate_1	FQ15
16	No	No		FEA Quad	Plate_1	FQ16
17	No	No		FEA Quad	Plate_1	FQ17
18	No	No		FEA Quad	Plate_1	FQ18
19	No	No		FEA Quad	Plate_1	FQ19
20	No	No		FEA Quad	Plate_1	FQ20
21	No	No		FEA Quad	Plate_1	FQ21

FEA Objects List Box.

FEA | FEA Objects | Objects Menu

Displays the Objects spreadsheet for ***all FEA Objects of the type checked*** in this menu.

- ***Select*** an FEA Object type in this menu to display its **Object** spreadsheet