VES-8000 Finite Element Analysis (FEA)

March 11, 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

The software described in this document is copyrighted by Vibrant Technology, Inc. or its suppliers and is protected by United States copyright laws and international treaty provisions. Unauthorized reproduction or distribution of this program, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the maximum extent possible under the law.

You may make copies of the software only for backup or archival purposes. No part of this manual may be reproduced or transmitted in any form or by any means for any purpose without the express written permission of Vibrant Technology.

Copyright © 1992-2025 by Vibrant Technology, Inc. All rights reserved. Printed in the United States of America.

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

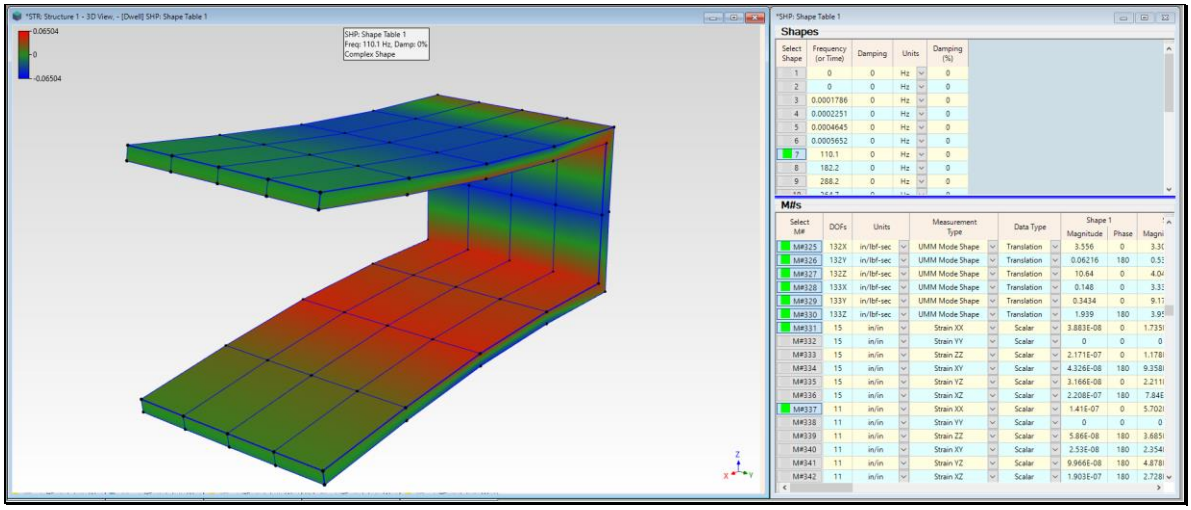
VES-8000 Finite Element Analysis (FEA) **6**
Additional Structure (STR) window Commands 6
FEA Objects 6
FEA Objects Used in FEA Models **7**
FEA Mass **8**
Using the FEA Assistant to Add Masses to a Model 8
FEA Spring **8**
FEA Damper **8**
Using the FEA Assistant to Add Springs & Dampers to a Model 8
FEA Rod **8**
Using the FEA Assistant to Add Rods to a Model 9
FEA Bar **9**
Cross Section Orientation Point 9
Using the FEA Assistant to Add Bars to a Model 9
FEA Triangle **9**
Using the FEA Assistant to Add FEA Triangles to a Model 9
FEA Quad **10**
Using the FEA Assistant to Add FEA Quads to a Model 10
Plate Element Stiffness Multiplier 10
FEA Tetra, FEA Prism & FEA Brick **10**
Using the FEA Assistant to Add FEA Tetras, FEA Prisms & FEA Bricks to a Modal 10
Physical Properties of FEA Objects 11
FEA | FEA Materials **12**
File | Import 12
File | Print Spreadsheet 12
Edit | Add 12
Edit | Delete 12
FEA | FEA Properties **12**
File | Import 13
File | Print Spreadsheet 13
Edit | Add 13
Edit | Delete 13
FEA | FEA Objects List **14**
FEA | FEA Objects | Objects Menu **14**
FEA | FEA Assistant **14**

Adding FEA Objects with the FEA Assistant....14
Add FEA Objects Tab....15
FEA | Calculate FEA Modes....16
Number of Modes....16
Rigid Body Offset (Hz)....16
Maximum Iterations....16
Include Damping Check Box....17
Computing A & B from Modal Frequency & Damping....17
Normal versus Complex Mode Shapes....17
FEA | Calculate Stress or Strain....18
Setting Up the Deflection and Stress-Strain Display....18
FEA | FEA Model Updating....19
Command Requirements....19
Best to Worst Solutions....20
Upper Spreadsheet....21
Lower Spreadsheet....21
FEA Properties That Can Be Updated....21
Solution Space....22
Solution Space Examples....22
Calculating Solutions....22
Error Function....22
Before Starting a Calculation....22
Upper Spreadsheet....22
Lower Spreadsheet....23
Solution | Calculate....23
Solution | Stop Calculation....23
Solution Scroll Bar....24
Display | Split....24
Display | Spreadsheets....24
Display | Bar Charts....24
Solution | Save Solution Mode Shapes....25
FEA | Point Matching....25
Before Using This Command....25
Creating FEA & EMA Substructures....26
Creating an FEA Substructure....26
Creating an EMA Substructure....26
Place the two Substructures Together in the Same Window....26

Aligning the Substructures 26
Point Matching Steps....26
FEA | Point Matching Command....27
FEA | Export the FEA Model....27

VES-8000 Finite Element Analysis (FEA)

If the **VES-8000 FEA Option** is authorized by your MEscope license, the following commands are enabled in the Structure window. Execute **Help | License Manager** to verify the Options authorized by your MEscope license.



***FEA** Mode Shape of the Jim Beam.*

Additional Structure (STR) window Commands

- **FEA** | Materials List
- **FEA** | Properties List
- **FEA** | **FEA** Objects List
- **FEA** | **FEA** Assistant
- **FEA** | Calculate **FEA** Modes
- **FEA** | Calculate Stress
- **FEA** | Calculate Strain
- **FEA** | **FEA** Model Updating
- **FEA** | Point Matching
- **FEA** | Export the **FEA** Model

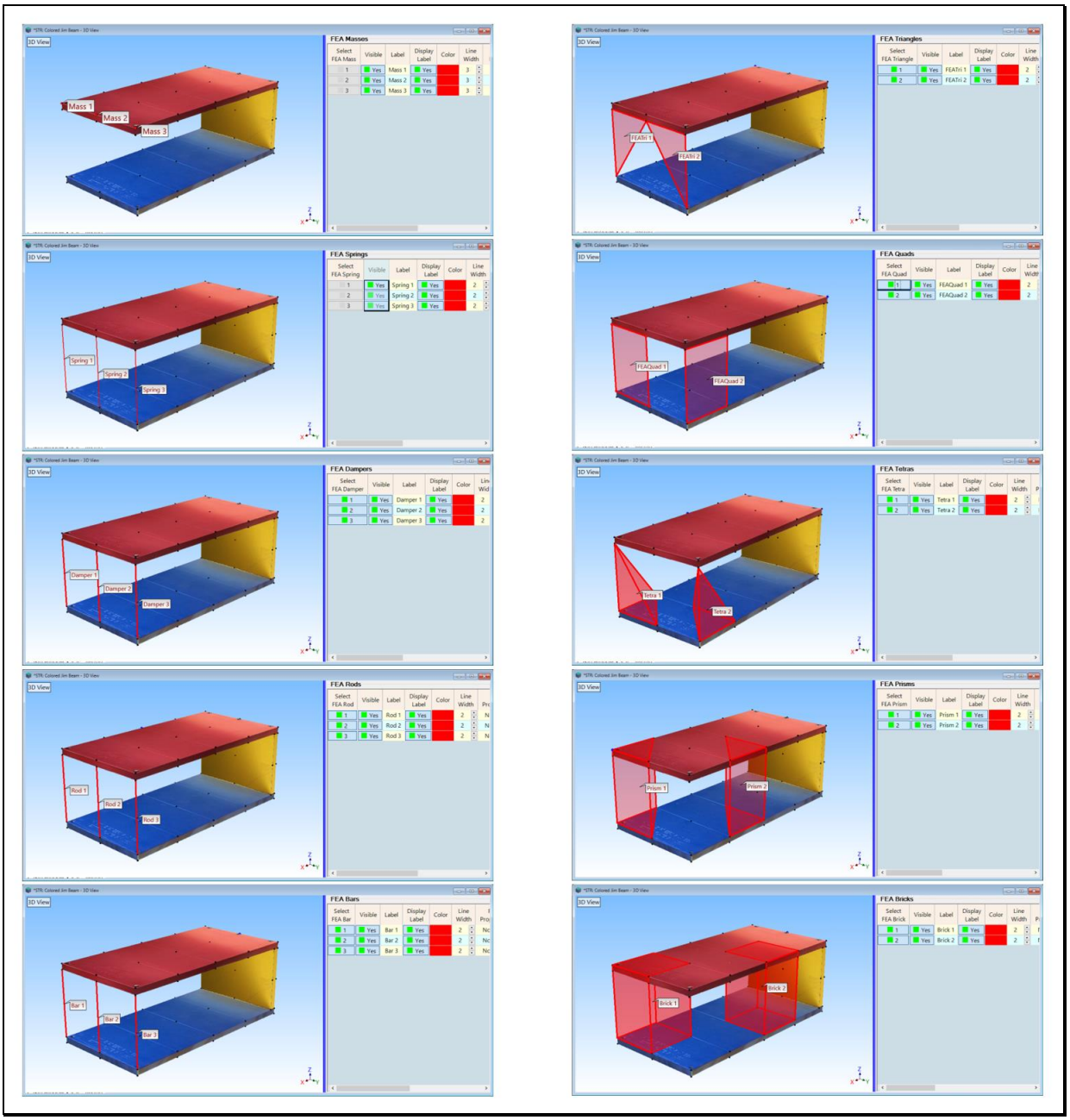
FEA Objects

- **FEA** Objects are added to the Current Objects list.

FEA Objects Used in FEA Models

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

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



*Types of **FEA** Objects used in **FEA** Models*

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 box
- 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

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

Stiffness is applied between two Points either ***axially*** (along the element axis), ***translationally*** in 3 directions or ***rotationally*** in 3 directions.

Stiffness can be constrained to specific directions by making selections from the **Orientation** columns of the end Points in the **FEA Spring** spreadsheet.

- Spring stiffness is defined on the **Springs** 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

FEA Damper

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

Damping are applied between two Points either ***axially*** (along the element axis), ***translationally*** in 3 directions or ***rotationally*** in 3 directions.

Damping can be constrained to specific directions by making selections from the **Orientation** columns of the end Points in the **FEA Damper** spreadsheet.

- Damping is defined on the **Dampers** tab in the **FEA | FEA Properties** dialog
- 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 Rods** 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 box
- 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

Cross Section Orientation Point

The cross section of an **FEA** Bar is oriented by selecting an **Orientation Point** for the Bar in the **FEA Bars** spreadsheet.

- The **Orientation Point** is defined by its ***row number*** in the Points spreadsheet
- The **Orientation Point** ***row number*** must be entered into the **Orient. Point** cell for each **FEA** Bar in the **FEA Bars** spreadsheet

The **Orientation Point** must be located in a ***plane that is normal to (at right angle to)*** the plane of the **FEA** Bar cross section.

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

An **FEA** Triangle is a ***linear plate element***, also called a ***membrane element***.

An **FEA** Triangle 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

- Plate thicknesses are defined on the **Plates** tab in the **FEA | FEA Properties** dialog box
- Plate material properties are defined in the **FEA | FEA Materials** dialog box
- The properties of each **FEA** Triangle must be chosen in its **FEA Property cell** in the **FEA Triangles** spreadsheet

Using the FEA Assistant to Add FEA Triangles to a Model

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

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

FEA Quad

An **FEA** Quad is a ***linear plate element***, also called a ***membrane element***.

An **FEA** Quad 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** Quad is attached between ***four Points*** on the structure model

- Plate thicknesses are defined on the **Plates** tab in the **FEA | FEA Properties** dialog box
- Plate material properties are defined in the **FEA | FEA Materials** dialog box
- The properties of each **FEA** Quad must be chosen in its **FEA Property cell** in the **FEA Quads** spreadsheet

Using the FEA Assistant to Add FEA Quads to a Model

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

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

Plate Element Stiffness Multiplier

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.

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

- **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

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 box
- Tetra, Prism & Brick material properties are defined in the **FEA | FEA Materials** dialog box
- Each **FEA** Tetra, **FEA** Prism & **FEA** Brick must have its property chosen in the **FEA Property cell** in the **FEA Tetra, FEA Prisms FEA Bricks** spreadsheet

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 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

Physical Properties of FEA Objects

The **FEA** Objects require the following physical properties.

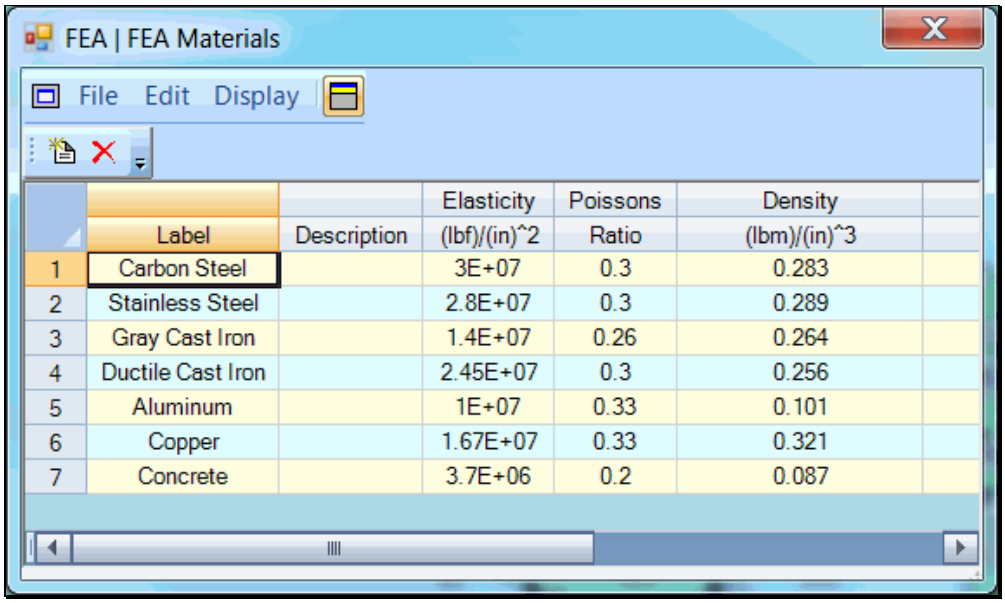
FEA Object	Physical Properties
Mass	Mass
Spring	Stiffness
Damper	Damping
Rod	cross sectional Area Elasticity, Density
Bar	cross sectional Area X inertia, Y inertia, XY inertia Elasticity, Density
Triangular Plate	Thickness, Elasticity Poisson's Ratio, Density
Quad Plate	Thickness, Elasticity Poisson's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

FEA | FEA Materials

Opens the **FEA** Materials window.

The **FEA** Materials contains ***material properties*** that are ***referenced*** by elements in the **FEA Properties** window.

- Material units are chosen on the **Units** tab in the **File | MEscope Options** box
- The **FEA** Materials 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 in a PDF file.

Edit | Add

Adds a new Material to the Materials List.

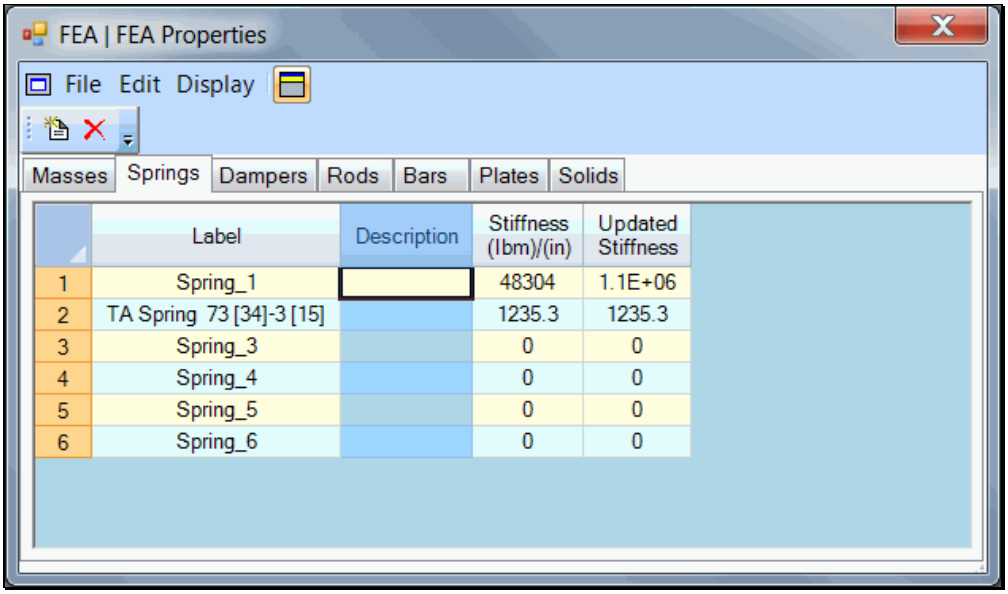
Edit | Delete

Deletes the currently selected Material from the Materials List.

FEA | FEA Properties

Opens the **FEA** Properties window.

- The **FEA** Properties 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**
- The **FEA** Properties is saved in the ***currently open*** **Project** file



***FEA** Properties 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
- Each **FEA Bar** requires a ***cross sectional area***, ***inertias***, and a ***material*** from the **FEA** Materials
- Each **FEA Plate** requires a ***thickness*** and a ***material*** from the **FEA** Materials
- Each **FEA Solid** requires a ***material*** from the **FEA** Materials

File | Import

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

File | Print Spreadsheet

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

Edit | Add

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

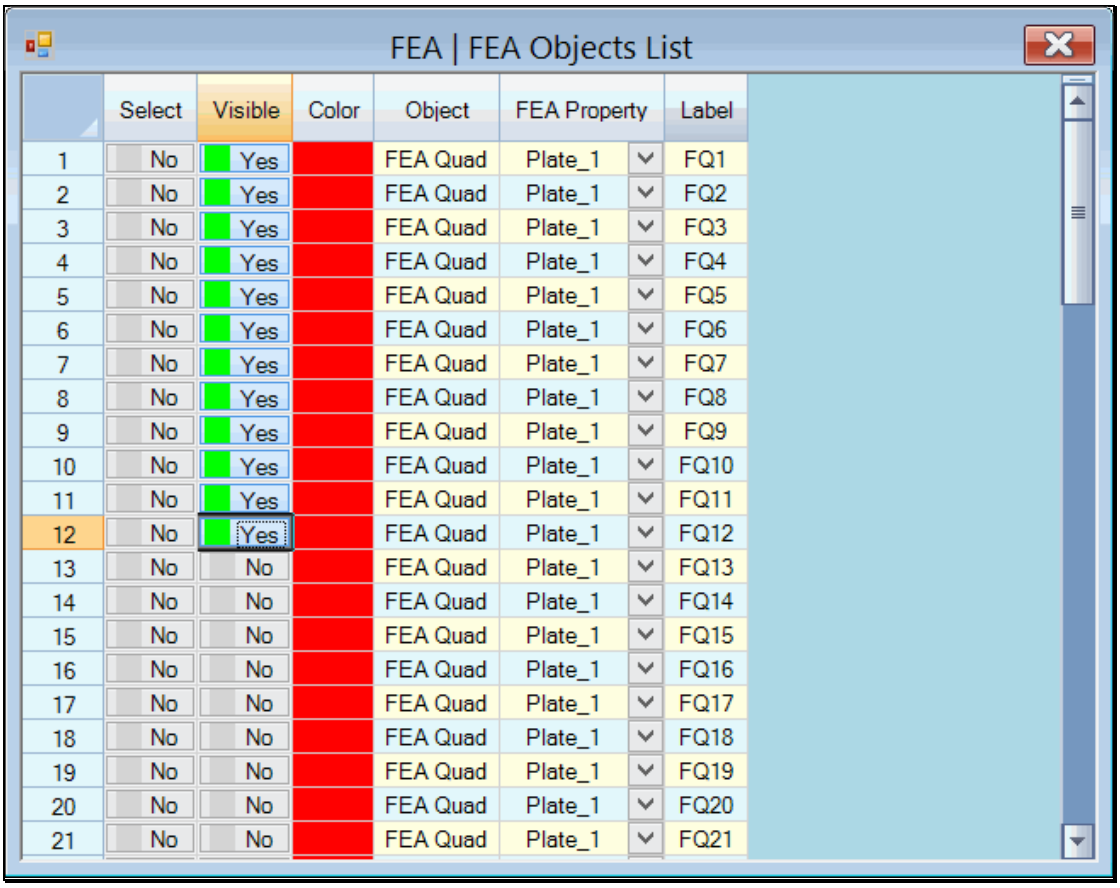
Edit | Delete

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

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



	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

FEA | FEA Assistant

The **FEA** Assistant allows you to ***add multiple* FEA Objects** to a structure model.

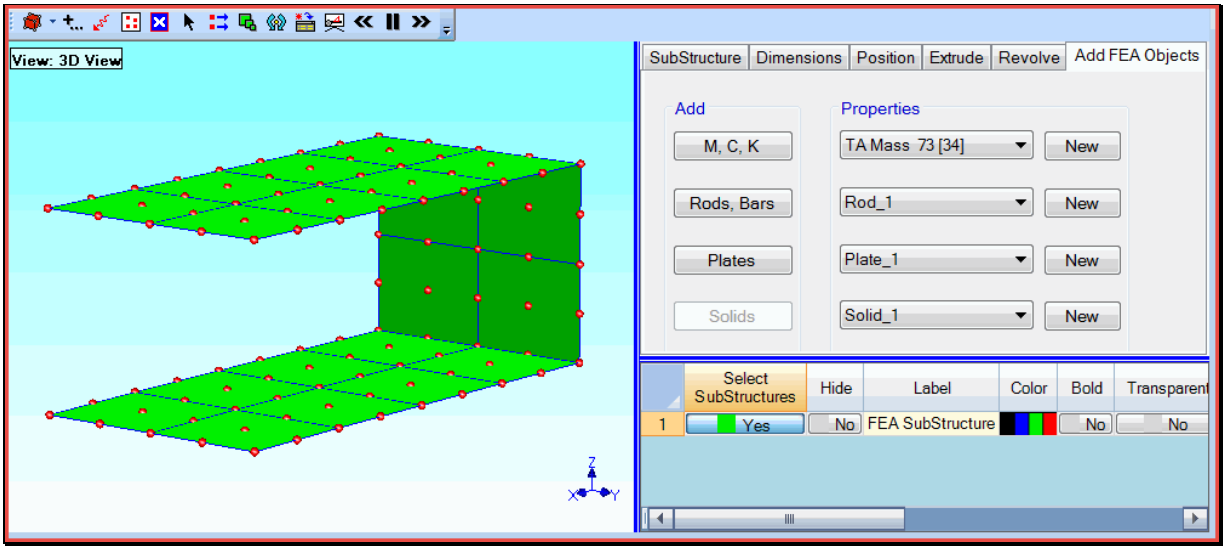
The **FEA** Assistant adds the **Add FEA Objects** tab to the Drawing Assistant tabs. See **Draw | Drawing Assistant** for details on using the Drawing Assistant tabs.

Adding FEA Objects with the FEA Assistant

With the **FEA** Assistant you can quickly create an **FEA** model by adding multiple **FEA** objects to a structure model.

- An **FEA Mass** can be added to each ***visible selected* Point** on the structure model
- An **FEA Spring, FEA Damper**, **FEA Rod** or **FEA Bar** can be added wherever there is a ***visible selected* Line** on the structure model

- An **FEA Triangle** can be added wherever there is a ***visible selected*** **Surface Triangle** on the structure model
- An **FEA Quad** can be added wherever there is a ***visible selected*** **Surface Quad** on the structure model
- **FEA Tetras, FEA Prisms & FEA Bricks *can only be added*** to a ***selected*** **Editable Substructure**, or to any **3D Substructure** that was created using the **Extrude** or **Revolve** command



*Structure (**STR**) window Showing **FEA** Assistant Tabs.*

Add FEA Objects Tab

Adds **FEA** Objects of the same type to the ***selected*** **Substructure**.

The **FEA** Assistant only operates on ***one selected Substructure*** at a time.

Before using the **FEA** Assistant, use **Draw | Substructures | Add Objects to Substructure** to ***create as many Substructures as necessary*** with only Points, Lines, or Surfaces in so that when they are selected, the desired **FEA** Objects will be added to the structure model.

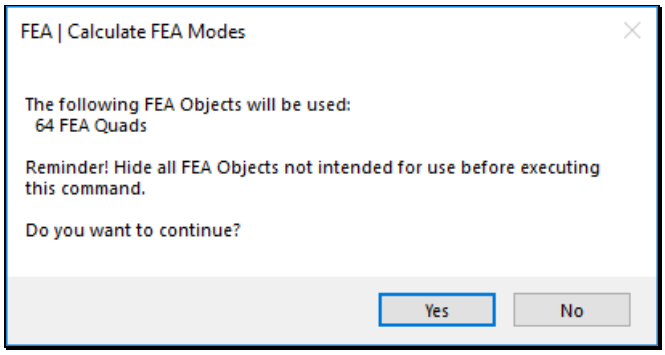
To add **FEA** Objects to a ***selected*** Substructure using the **Add FEA Objects** tab,

- Choose an **FEA** property from the **Properties** list next to the (**M, C, K**), (**Rods**, **Bars**), **Plates** or **Solids** button
- If the **FEA | Properties List** is empty, ***press*** the **New** button to add a new property to the **FEA** Properties list
- ***Press*** the ***enabled*** (**M, C, K**), (**Rods, Bars**), **Plates** or **Solids** button to add the **FEA** Objects to the ***selected*** Substructure

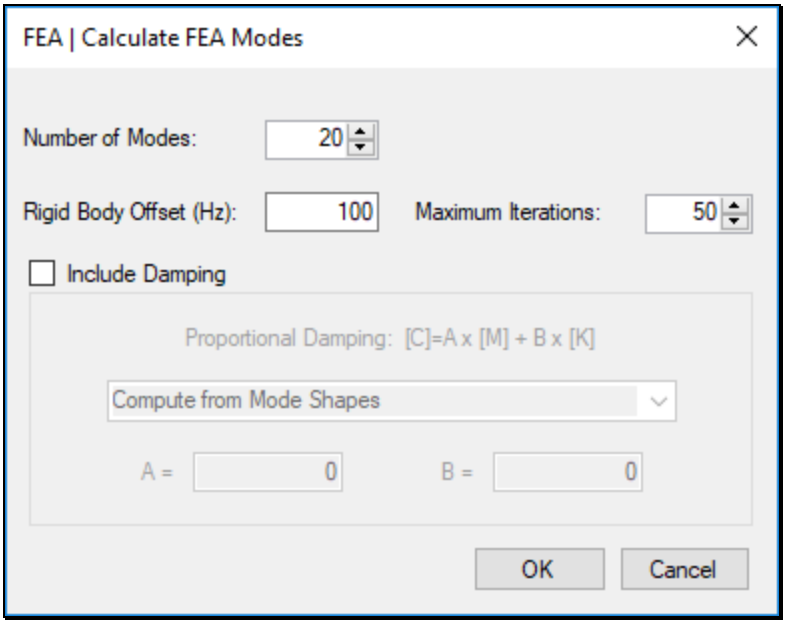
FEA | Calculate FEA Modes

Calculates the mode shapes of the **FEA** model represented by the visible **FEA** Objects on the model in the Structure window.

When this command is executed, the following dialog box will open listing the **FEA** objects that will be used to calculate **FEA** modes shapes,



The next dialog box (shown below) is where the **FEA** mode shape calculation is setup.



*Calculate **FEA** Modes Dialog Box.*

Number of Modes

- Enter the ***total number of modes*** desired in the Solution. (**10** to **50** is typical)

Rigid Body Offset (Hz)

- Enter a frequency ***less than the expected lowest frequency*** **flexible-body mode shape**, (**20 to 100 Hz** is typical)
- If the Structure model has ***free-free boundary conditions,*** **six (6) rigid-body mode shapes** will be calculated as part of the Solution

Maximum Iterations

- Enter the maximum number of expected iterations required to converge on a Solution (**20** to **50** is typical)

Include Damping Check Box

If this box is ***checked***, a ***Proportional Damping matrix*** is added to the equations of the **FEA** model.

Proportional Damping matrix ➔ A x [Mass Matrix] + B x [Stiffness Matrix]

(A>=0) & (B>=0) ➔ proportionality constants

Computing A & B from Modal Frequency & Damping

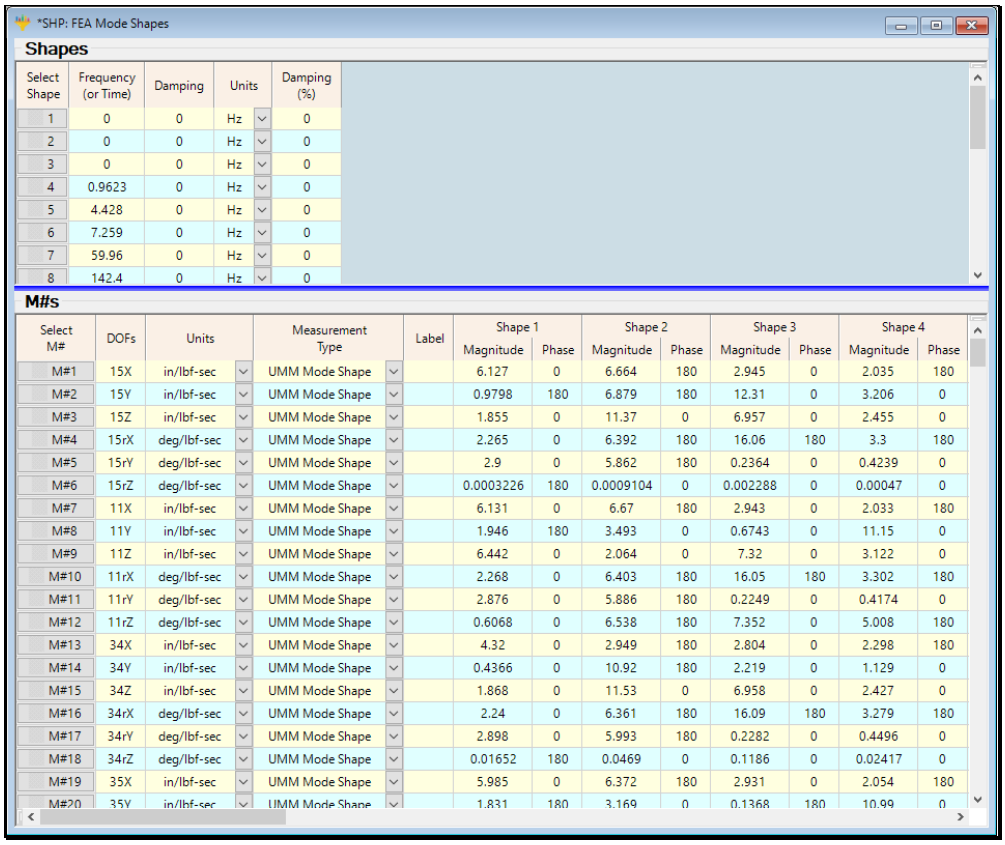
If a **Shape Table** with modal frequency & damping in it is chosen from the list in the dialog box, **A** & **B** values will be calculated from the modal frequency & damping values.

Using experimental modal frequency & damping values typically gives realistic A & B estimates.

Normal versus Complex Mode Shapes

When the **FEA** modes have been calculated, the Shape Table selection dialog box will open allowing you to save the **FEA** mode shapes into a Shape Table (**SHP**).

- If **FEA Dampers** are attached to the model, complex mode shapes are calculated when this command is executed
- If **no FEA Dampers** are attached to the model and **Proportional Damping *is not included***, normal mode shapes are calculated when this command is executed
  - Normal mode shapes ***have no modal damping***
  - Normal mode shapes are ***real-valued***. Each mode shape component has a ***phase angle of 0 & 180 degrees***
  - Complex modes ***have modal damping*** and are ***complex-valued***. Each mode shape component has ***arbitrary phase angles***



Shape Table Window Showing Normal Mode Shapes.

FEA | Calculate Stress or Strain

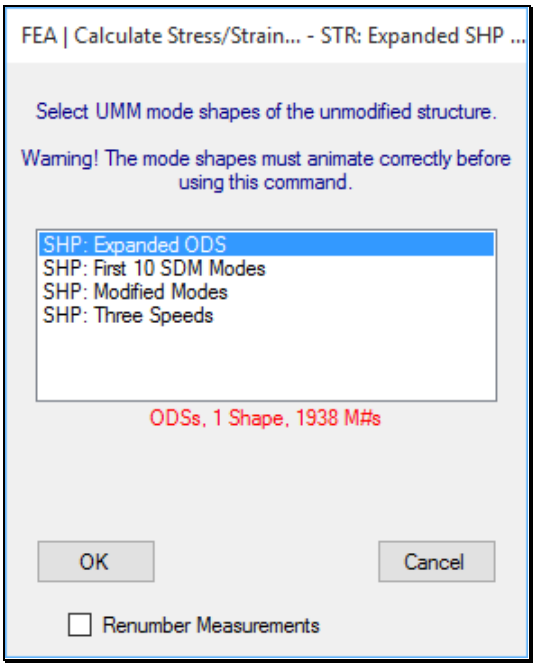
This command uses an ODS or mode shape to deflect an **FEA** model that contains **FEA** Solid Objects.

This command is used in a Structure (**STR**) window with an **FEA** model containing **FEA Solid elements**, (**FEA Tetras**, **FEA Prisms**, or **FEA Bricks**).

An ***ODS or mode shape*** can be used to deflect the **FEA** model.

The stress or strain of the deflected **FEA** model are displayed using color contours.

When this command is executed a dialog box will open from which to select a Shape Table (**SHP**) with ODS or Mode Shapes in it, as shown below.



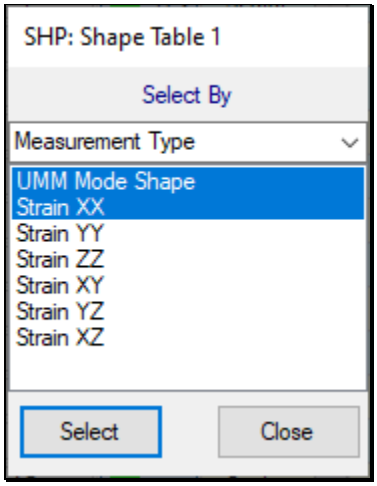
When this calculation has completed, stress or strain values ***are added*** as ***Measurement Type*** ➔ **Scalar** to the **DOFs** of each shape in the Shape Table (**SHP**) used to deflect the **FEA** model.

- Save the shapes with stress or strain added to them into a Shape Table (**SHP**)

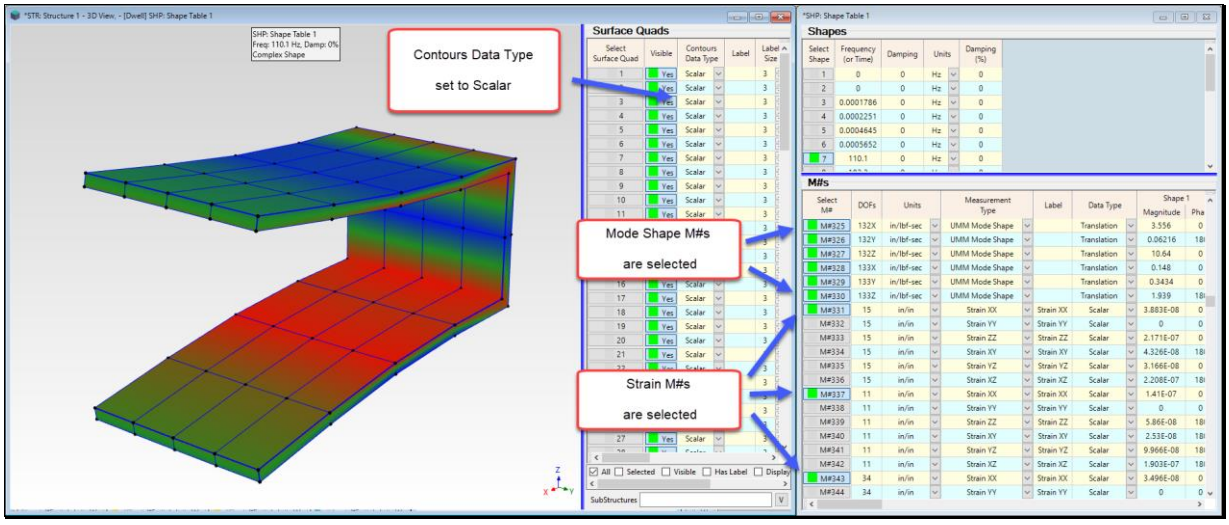
Setting Up the Deflection and Stress-Strain Display

Several steps are necessary to display the deflection of an ODS or mode shape together with the stress or strain caused by the ODS or mode shape.

1. Set **Contours Data Type** ➔ **Scalar** ***for all*** **Surfaces** in the Structure (**STR**) window containing the **FEA** model
2. **Check Deflection | Contours | Contour Colors** in the same Structure (**STR**) window
3. Execute **M#s | Select | Select By** in the Shape Table (**SHP**) window containing both shapes and the stress-strain values
4. In the dialog box that opens, ***hold down*** **the Ctrl key** and select **Translation** and ***one of the*** **stress-strain *values*** as the **Measurement Types** to display in animation, as shown below
5. Execute **Animate | Animate a Shape**



Measurement Type ➔ *Translation & Measurement Type* ➔ *Strain XX*



Settings to Display Defection and Stress-Strain Together.

FEA | FEA Model Updating

FEA Model Updating ***changes*** the ***Properties of selected FEA Objects*** on an **FEA** model so that its modal parameters ***more closely match*** a set of ***experimental*** (**EMA**) modal parameters.

This command uses the **SDM command** to solve for new mode shapes using **FEA** properties from a ***user-defined Solution Space*** of **FEA** Properties.

The **FEA** property ***Solution Space*** is defined by the **Minimum, Maximum,** & **Number of Steps** of each **FEA** property.

Command Requirements

The following items are required to perform **FEA** Model Updating,

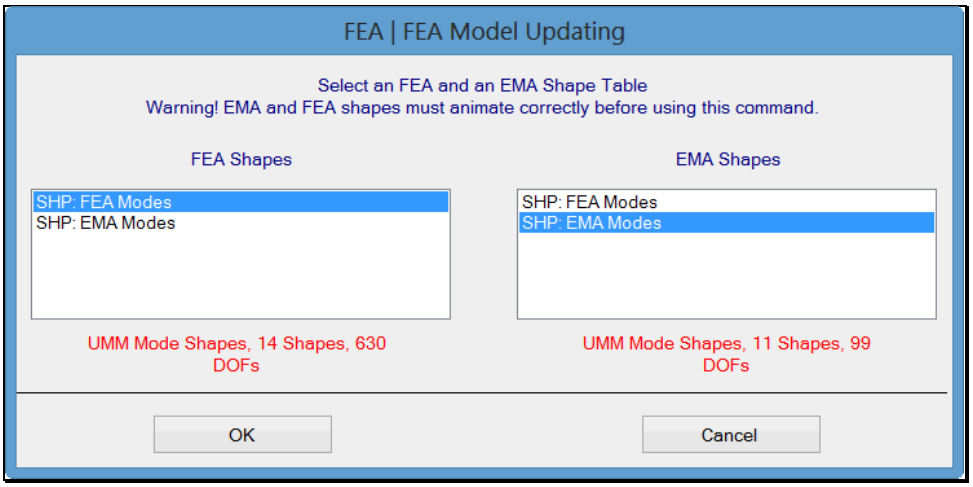
- An **FEA model** containing the **FEA** Objects with ***current*** **properties** to be updated
- **FEA mode shapes** for the ***current*** **FEA** model
- **EMA modal parameters**

Best to Worst Solutions

All Solutions are ordered from **best** to **worst**.

The ***best* Solution** is the one that ***minimizes*** an **Error Function** between the **Solution** parameters and **Target EMA** modal parameters.

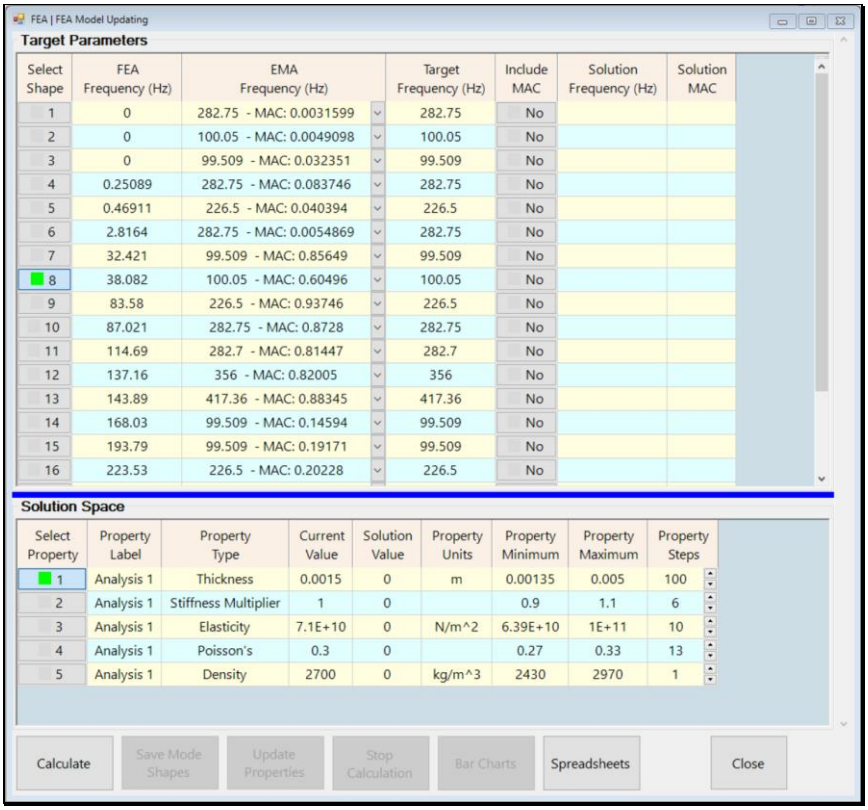
When this command is executed, the following dialog box is opened.



Shape Table Selection Dialog Box.

- Choose a Shape Table containing the **FEA Mode Shapes** on the left, and a Shape Table containing the **EMA Mode Shapes** on the right

When **OK is *pressed***, the **FEA** Model Updating window will open, as shown below



***FEA** Model Updating Showing One Selected Shape Pair & One Selected Property for Updating.*

Upper Spreadsheet

The ***upper*** **spreadsheet** lists the modal frequencies and **MAC** values of all matching **FEA-EMA mode pairs**.

Each **FEA-EMA** mode shape pair is the pair with the ***maximum*** **MAC** ***value*** between all pairs of **FEA** & **EMA** mode shapes.

- The **FEA** modal frequencies are listed in the **FEA Frequency (Hz)** column
- The **EMA** modal frequencies are listed in the **EMA Frequency (Hz)** column
- The **MAC** value for each matching **FEA-EMA mode shape pair** is also listed in the **EMA Frequency (Hz)** column
- The **EMA** modal frequencies of each matching **FEA-EMA pair** are also listed in the **Target Frequency (Hz)** column
- If **FEA-EMA mode pairs** ***are selected***, only those ***selected*** **pairs** are used to calculate the **Error function**

If ***no*** **FEA-EMA mode pairs** ***are selected***, ***all*** **FEA-EMA mode pairs** are used to calculate the **Error function**.

Lower Spreadsheet

The ***lower*** spreadsheet lists the **FEA** Properties of ***all*** **visible FEA Objects** attached to the structure model.

The ***Solution Space*** for each Property is defined by its ***user-defined*** **Minimum, Maximum & Steps**.

Minimum & Maximum define the ***lower*** & ***upper*** bounds of the Property **Solution Space**

Steps is the number of property steps evaluated between the Minimum & Maximum bounds of the **Solution Space**

If **Properties** ***are selected***, only those ***selected*** **properties** are used to define the **Solution Space**

If ***no*** **Properties** ***are selected***, the **Minimum, Maximum & Steps** of ***all*** properties are used to define the **Solution Space.**

FEA Properties That Can Be Updated

Below is a Table of the Properties that can be updated using **FEA** Model Updating.

FEA Object	Property to Update
Spring	stiffness
Mass	mass
Rod	cross sectional area, material (Elasticity, Poisson's ratio, Density)
Bar	cross sectional area & inertias material (Elasticity, Poisson's ratio, Density)
Plate	thickness material (Elasticity, Poisson's ratio, Density)
Solid	material (Elasticity, Poisson's ratio, Density)

FEA *Properties That Can Be Updated Using* ***FEA*** *Model Updating.*

Solution Space

The **Minimum**, **Maximum & Steps** of ***all*** (or ***selected***) Properties in the ***lower*** **spreadsheet** define the **Solution Space** over which Solutions are calculated.

The Property **Minimum & Maximum** define the ***lower & upper bounds*** of the **FEA** Property in the Solution Space

Steps is the number of property steps evaluated between the Minimum & Maximum bounds of the **Solution Space**

The **total number of Solutions** is the **product of the Steps** of ***all (or selected)*** **FEA** Properties

Solution Space Examples

- If Property Minimum ➔ 1, Maximum ➔ 10, Steps ➔ 10, **10 Solutions** are calculated for Property values **(1,2,3,4,5,6,7,8,9,10)**
- If ***three*** **FEA** Properties **each have 10 Steps**, **1000 Solutions** are calculated using Property values in the **Solution Space**

Calculating Solutions

When **Solution | Calculate** is executed, **FEA** Model Updating Solutions are calculated over the **Solution Space** of ***all*** (or ***selected***) **FEA** Properties in the ***lower*** **spreadsheet**.

All combinations of property values in the **Solution Space** are used, so the **best Solution** will always be found.

Error Function

An **Error function** is used to order the Solutions.

The Error function is the ***summation of the differences*** between the **Solution & Target** frequency of ***each selected*** **mode shape pair**.

If **Include MAC** is ***depressed*** for a ***selected*** **pair**, a second term **(1 - MAC)** is added to the **Error function**, as shown below.

The formula for the **Error function** is:

$$\text{Error} = \sum_{\text{modes}} \left[\frac{\|\text{Fup} - \text{Fema}\|}{\text{Fema}} + (1 - \text{MACup}) \right]$$

Fup ➔ Frequency of the **Updated** mode shape

Fema ➔ Frequency of the **EMA** (**Target**) mode shape

MACup ➔ **MAC** value between the **Updated FEA & Target** mode shapes

Before Starting a Calculation

Upper Spreadsheet

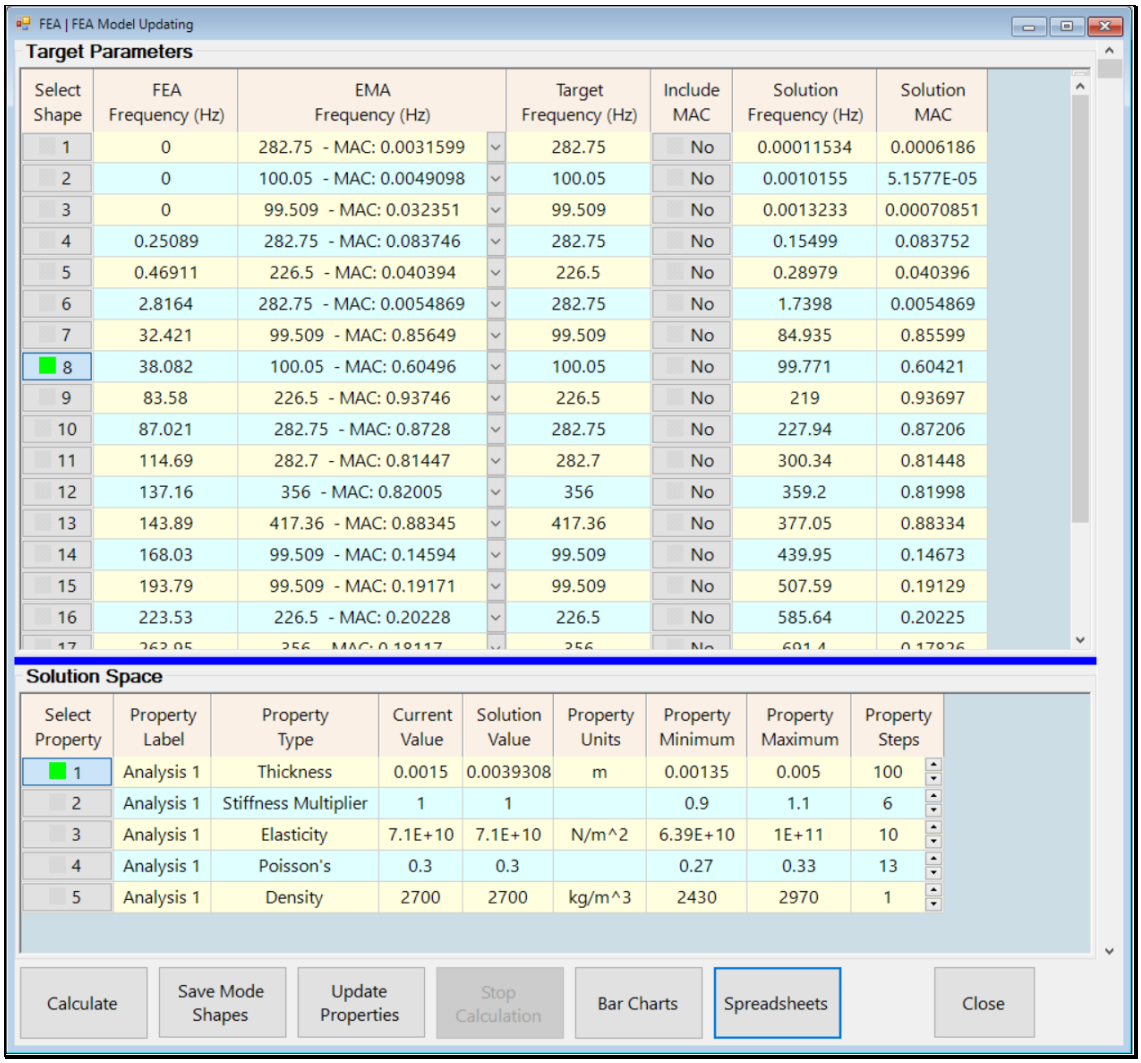
- ***Change*** the **Select Pair** button ➔ Yes ***each mode shape pair*** to be included in the Error function
- Enter a **Target frequency** into the ***upper*** **spreadsheet** for each ***selected*** **mode pair** to be used in the **Error** function

If no **Target modal parameters** are entered for a ***selected mode pair***, that pair will be excluded from the **Error function**.

Lower Spreadsheet

- ***Depress*** the **Select Property** button in the ***lower*** **spreadsheet** to ***select*** **each property** to be used in the Modal Sensitivity calculations
- Enter Property **Minimum, Maximum & Steps** to define the **Solution Space** for each **FEA** Property

If ***no*** **Properties** ***are selected***, the **Minimum, Maximum & Step** of ***all*** **FEA Properties** will be used to define the Solution Space.



FEA *Model Updating Window After Solutions are Calculated.*

Solution | Calculate

Calculates the Solutions for ***all*** (or ***selected***) **FEA** Properties in the ***Solution Space*** defined in the ***lower*** spreadsheet.

Solution | Stop Calculation

Stops the calculation and displays the Solutions that have already been calculated.

Solution Scroll Bar

If the Solution is stopped or all Solutions have been calculated, a **Solution Scroll Bar** is displayed on the ***right side*** of the window.

Parameters of each Solution corresponding to the **Solution Scroll Bar** position are displayed in both the ***upper*** & ***lower*** spreadsheets

The ***best*** Solution is displayed when the Solution Scroll Bar is in ***its top position***.

The ***worst*** Solution is displayed when the Solution Scroll Bar is in ***its bottom position***.

If ***no*** **mode shape pairs** ***are selected*** in the ***upper*** spreadsheet, Solutions are displayed in the order in which they were calculated.

Display | Split

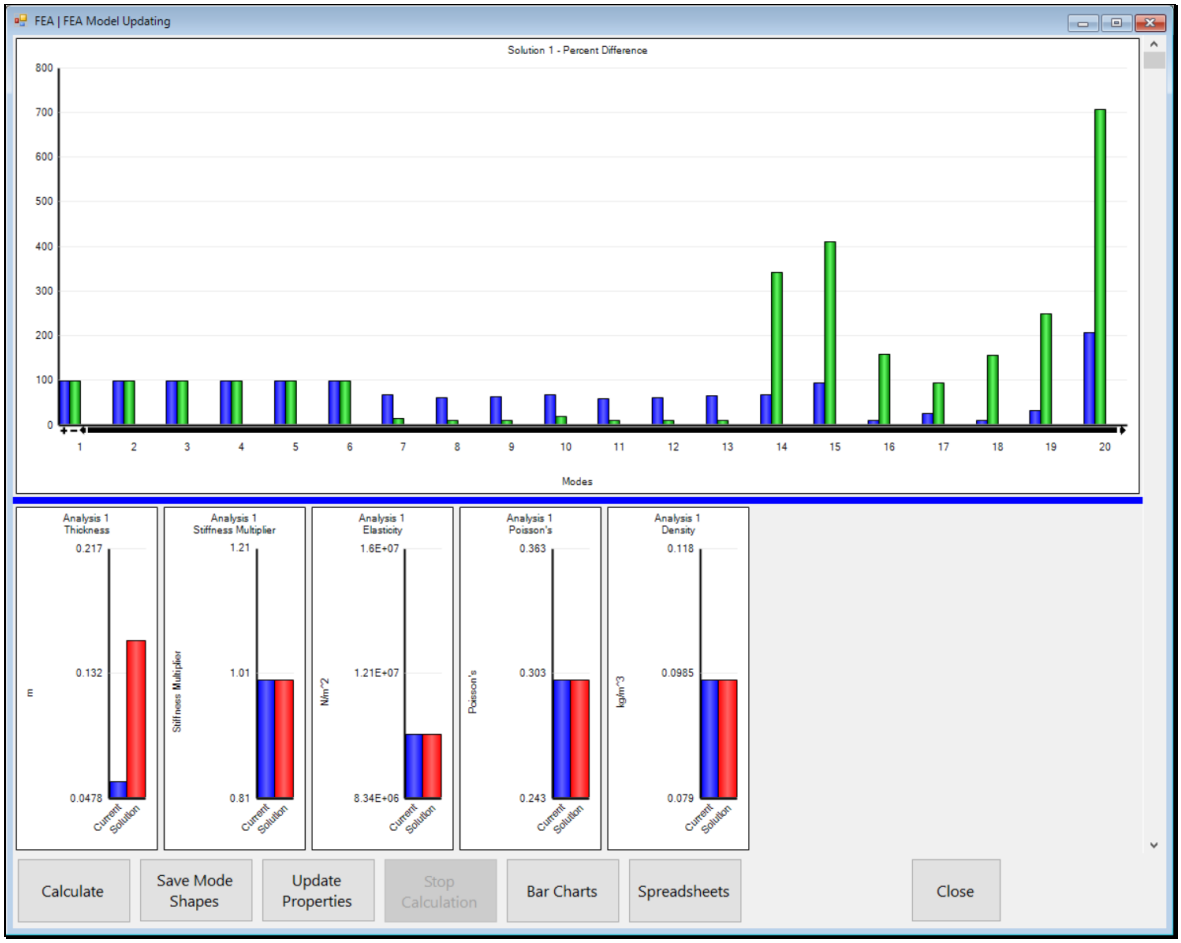
Toggles the spreadsheet display between ***Upper-Lower*** and ***Left-Right*** format.

Display | Spreadsheets

When this command is ***checked***, the Solutions are displayed using ***two spreadsheets***.

Display | Bar Charts

When this command is ***checked***, the Solutions are displayed graphically, as shown below.



Bar Chart Display.

The ***upper left*** bar chart shows two bars for ***each*** **mode shape pair**.

The left (**blue**) bar is the **Percent Difference** between the **FEA** & **Target** frequency
The right (**green**) bar is the **Percent Difference** between the **Solution** & **Target** frequency

- ***Hover*** the mouse pointer over each bar on a bar chart to display its values

If **MAC** values are used in the Error function, the ***upper right*** **bar chart** displays **MAC** values using two bars for ***each*** **mode shape pair**

The left (**blue**) bar is the **MAC** value between the **FEA** & **EMA** mode shapes
The right (**red**) bar is the **MAC** between the **Solution** & **EMA** mode shapes

The ***lower*** bar chart displays the **Current** & **Solution** values of the **FEA** Properties.

- ***Hover*** the mouse pointer over each bar to display its numerical values

Solution | Save Solution Mode Shapes

Saves the mode shapes into a Shape Table that result from the ***current*** **FEA Properties** in the ***lower*** spreadsheet.

- Before executing this command ***scroll*** the **Solution Scroll Bar** on the ***right-hand*** side to display the ***desired*** **Solution**

FEA | Point Matching

This command carries out the following steps in order.

1. Numbers all Un-numbered **FEA** Structure Points.
2. Finds the **EMA** structure Point that is ***geometrically closest*** to each **FEA** structure Point.
3. Re-numbers the **EMA** structure Points using **FEA** structure Point numbers.
4. Replaces the **FEA** Measurement Axes with the **EMA** Measurement Axes.
5. Re-defines the **FEA** mode shapes to match the new **FEA** Measurement Axes.
6. Replaces **FEA** mode shape **DOFs** with new **FEA** Measurement Axis labels

To compare **FEA** & **EMA** modes shapes using **MAC** or **SDI,** some **FEA** mode shape components ***must have*** **DOFs** ***that match the*** **DOFs** of **EMA** mode shape components.

Point matching ***is required*** if,

1. The **Display | MAC** command is executed between **FEA** & **EMA** mode shapes in two Shape (**SHP**)Tables
2. **MAC** is displayed with **Animate | Animate a Pair** between **FEA** & **EMA** mode shapes
3. **MAC** is used in the **Error function** in **FEA** Model Updating

There can be several differences between **FEA** & **EMA** mode shapes,

1. **FEA** mode shapes usually have ***many more*** **DOFs** (point numbers & directions) than **EMA** mode shapes
2. Points are ***numbered differently*** on an **FEA** model than on an **EMA** model
3. Directions ***are different*** at each point (node) on an **FEA** model compared to directions used during a modal test

Before Using This Command

The following steps are required before executing the Point Matching command.

1. Create a Substructure for the **FEA** model
2. Create a Substructure for the **EMA** model
3. Copy or Paste the two Substructures together in the same Structure (**STR**) window
4. ***Geometrically align*** the **FEA model** with the **EMA model**

Creating FEA & EMA Substructures

Both the **FEA** & **EMA** Substructures must be contained in the ***same*** Structure **(STR)** window.

The **FEA model *must precede*** the **EMA model** in the Substructure spreadsheet.

Creating an FEA Substructure

- ***Select*** all Points on the **FEA** model
- Execute **Draw | Substructures | Add Objects**
- ***Press*** the **New Substructure File** button in the Substructure selection box, enter ***"FEA Model"*** into the next dialog box, and click on **OK**

Creating an EMA Substructure

- ***Select*** all Points on the **EMA** model
- Execute **Draw | Substructures | Add Objects**
- ***Press*** the **New Substructure** button in the Substructure selection box, enter ***"EMA Model"*** into the next dialog box, and click on **OK**

Place the two Substructures Together in the Same Window

- ***Select*** the **FEA** Model and execute **Edit | Copy Objects to File** in its window
- ***Press*** the **New Structure** button and save the **FEA Model** into the new Structure (**STR**) file
- ***Select*** the **EMA** Model and execute **Edit | Copy Objects to File** in its window
- ***Select*** the Structure (**STR**) file with the **FEA Model** in it and ***click*** on the **Add To** button

Aligning the Substructures

In the Structure (**STR**) window containing both the **FEA** & **EMA** Substructures,

- Execute **Draw | Drawing Assistant** to open the Drawing Assistant tabs
- ***Select*** the **FEA** Model
- Use the controls on the **Dimensions** and **Position** tabs to align the Points of the two Substructures ***geometrically as close as possible***

When the two Substructures are aligned, they are ready for Point Matching.

Point Matching Steps

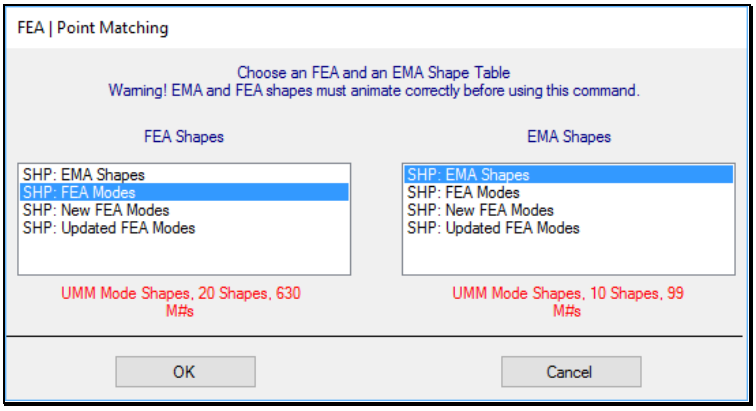
After an **FEA** model has been ***geometrically aligned*** with an **EMA** model in the same **STR** window, **FEA | Point Matching** carries out the following steps,

1. ***Re-numbers*** each Point on the **EMA model** using the Point number of the ***closest* Point *geometrically*** on the **FEA model**
2. ***Re-numbers*** the **DOF** of ***each* EMA mode shape** component using the ***re-numbered* Point** on the **EMA** model
3. ***Copies the* Measurement Axes** of each Point on the **EMA** model to its ***matching* Point** on the **FEA** model
4. ***Transforms each* FEA mode shape** component into the **Measurement Axis coordinates** of the ***matching* Point** on the **EMA** model

FEA | Point Matching Command

- Make sure that the two Shape (**SHP**) Tables with **FEA** & **EMA** mode shapes are open
- ***Select both*** the **FEA** & **EMA** Models
- Execute **FEA | Point Matching**

The Point Matching dialog box will open as shown below



- ***Select*** the **FEA modes** in the ***left list***, the **EMA modes** in the ***right list***, and ***click*** on **OK**

When Point Matching is completed, a series of dialog boxes will open, allowing you to save the two ***point-matched*** **Models** and the two ***point-matched*** **Shape Tables** into new files.

- The ***point-matched FEA & EMA Shape Tables*** can now be used for **MAC** calculations

FEA | Export the FEA Model

Exports an **FEA** model into a disk file using the **NASTRAN** bulk data deck format.