PlateMesh© Basic Manual

Document Version 1.1 (4/7/2020)

This **Basic Manual** consists of the basic instructions and information. The **Extended Manual** provides additional details about the inner workings of the program. There are also two additional documents (total of 4). They are for **Validation Cases** and **Convergence Studies**.

ABOUT

PlateMesh© is a pre/post processer that determines the Eigenvalue for a rectangular plate. The program uses the open source CalculiX solver (http://www.dhondt.de). It is a feature rich program for plate buckling analysis, addressing nearly all scenarios. The major capabilities are:

- Mesh density control and the option to use 4-noded and 8-noded quad elements

- Any combination of Nx, Ny, Nxy and also in-plane moments

- Any combination of edge supports (4 edges) that can be free, simple, or clamped

- Isotropic, Orthotropic, and Composites materials are supported

- Transverse shear stiffness effects

- Curvature may be defined in both directions

- Visual display of the buckled mode shape

INSTALLATION

1. Macros must be enabled for the *PlateMesh©* Excel program. Use the "Test" button in the Excel file to determine if macros are enabled. If they are enabled, a message box will state that they are.

2. Unpack the "PlateMesh_X.zip" package to the "C:\StructuralFEA" directory (default). Be sure that the directories "C:\StructuralFEA\PlateMesh" and "C:\StructuralFEA\PlateMesh\CalculiX" were created after unpacking.

3. From the PlateMesh download section on www.structuralfea.com, download the "CalculiX_2_16.zip" file and extract the contents to "C:\StructuralFEA\PlateMesh\CalculiX" (this directory should have been created in Step 2)

4. (Optional) — On the "Main" sheet, change the default paths. Pay attention to the comments and restrictions.

BASIC USE

STEP 1. On the "Main" Sheet, input the values on the left hand section first (vertical section). Only the material properties for the chosen type are used. Curvatures may be blank.

STEP 2. Input the edge conditions (free, simple, clamped) for each edge and the applied loading (the section underneath the 3 images).

STEP 3. Click "2 - Run/Update" to perform the Eigenvalue analysis.

SETP 3 (Optional). Click "2 - Run/Update and Plot" to perform the Eigenvalue analysis and plot the mode shape.

STEP 4 (Optional). After performing either of the Run/Update options, you may choose "3 - Show Deflection" or "3 - Show Fringe" or "4 - View in CGX PostProcessor".

ACCEPTABLE INPUTS VALUES

- Width, Height: Must be numerical values greater than zero. Can not be blank.

- QUAD4 or QUAD8: Input must be "4" or "8".

- Full(F) or Reduced(R): Input must be "F" or "R".

- Elements on short edge: An *even* integer that is great than 6 or greater if QUAD8 is used or 10 or greater if QUAD4 is used.

- Allow stretch?: "Y" for Yes or "N" for No.

- Curvature: A blank entry indicates no curvature. A numerical value indicates the radius of curvature.

- Material (Choose Option): Enter one of "A", "B", or "C". "A" for Isotropic, "B" for Orthotropic, or "C" for Composite.

-Option A: If "A" is chosen, all 3 numerical values must be entered.

-Option B: If "B" is chosen, all 7 numerical values must be entered.

-Option C: If "C" is chosen, all 7 numerical values must be entered for Mat 1. Mat 2, Mat 3, and Mat 4 are optional. At least 1 ply must be entered. For each ply, the corresponding material must be entered. For each ply, a theta value (in degrees), must be entered. Theta is defined in the Nx, Ny, Nxy loading figure.

- Edge Conditions: Each edge must be have an input of either "F", "S", or "C", corresponding to free, simply supported, and clamped, respectively. A free edge has no boundary conditions. A simply supported edge is restrained in the Z-direction. A clamped edge is restrained in the Z-direction and also rotation (X-rotation for Top/Bottom edges and Y-rotation for Left/Right edges).

- Loading: Values may be empty. If the loading causes a negative Eigen value, the value is not reported. For example, a +Nx loading (without any other loads), will not cause buckling but will have an associated negative Eigen value (not reported).

- Scale Factor: A numerical value that may be positive or negative. A default value of 10 is expected to show a "reasonable" amount of deformation.

For many of these input values, there are internal checks for valid entries. If the user enters an invalid entry, the program will explain how the entry should be properly entered. However, some entries are not checked for validity.

QUAD4 AND QUAD8

The program supports both a 4-noded quadrilateral element (referred to as QUAD4) and an 8-noded quadrilateral element (referred to as QUAD8). For most solvers, the QUAD4 is "tried and true", while the QUAD8 can sometimes exhibit issues (depending on the solver/formulation). However, the QUAD8 is much more forgiving for parameters such as aspect ratio and skew angle (skew angle is not a factor for *PlateMesh* because all of the elements are rectangular). Also, the QUAD8 requires fewer elements to reach convergence. For more information about convergence with the QUAD4 and QUAD8, refer to the Convergence Study document.

MESH DENSITY

For a given plate, the "short edge" is considered to be the minimum of W or H (relevant to the input "Elements on the short edge"). "Elements on the short edge" must be even. There are also minimum values to prevent an inaccurate result (a message box will appear if the minimum is not met).

The number of elements on the "long edge" are determined by creating the mesh such that the element aspect ratio is as close to 1:1 as possible (unless "Allow stretch?" is set to "Y" – See the next section).

ALLOW STRETCH OPTION

If "Allow stretch?" is set to "N", the element aspect ratio will be as close to 1:1 as possible, regardless of the plate's aspect ratio.

If "Allow stretch?" is set to "Y", the element aspect ratio will be as close to 2:1 as possible if W/H or H/W exceeds 2.0. This is done to minimize the number of elements for plates that have a larger aspect ratio.

Based on the Convergence Study document, setting Allow Stretch to "Y" does not seem to significantly affect accuracy for large aspect ratio plates.

FULL INTEGRATION OR REDUCED INTEGRATION

The element formulation may be either Full Integration or Reduced Integration, with the following exceptions.

For the QUAD4, Reduced Integration was shown to have very poor accuracy. Therefore, only Full Integration is allowed for the QUAD4.

If a composite is used, only the QUAD8 with Reduced Integration is allowed. This is a requirement of the CalculiX solver.

The Convergence Study document compares the Full Integration and Reduced Integration options.

AUTO MESH SETTINGS

Using an Auto Mesh Setting (Med or High) will populate the Element Type and Mesh Density fields with predefined values. The predefined values were determined from the Convergence Study document.

Med – QUAD8 with Reduced Integration is selected. There are 10 elements on the short edge and "Allow Stretch?" is set to "Y". For this scenario and for the cases evaluated, there was a max error of 4.5% and most cases were within 1%.

High – QUAD8 with Reduced Integration is selected. There are 16 elements on the short edge and "Allow Stretch?" is set to "N". For this scenario and for the cases evaluated, there was a max error of 1.8% and most cases were within 1%.

BASIC LOADING

The loading directions are shown in the Excel file (click the pictures to expand) and are also shown below. All loads are running loads or line loads. In other words, they are the load per unit length. Dividing a running load by the total plate thickness determines the applied stress. To be clear, this is the average through-thickness applied stress for composite materials.

For Nx and Ny, positive input values create tension loading and negative values create compression loading. Regarding the in-plane moments, there is more information in the Extended Manual.
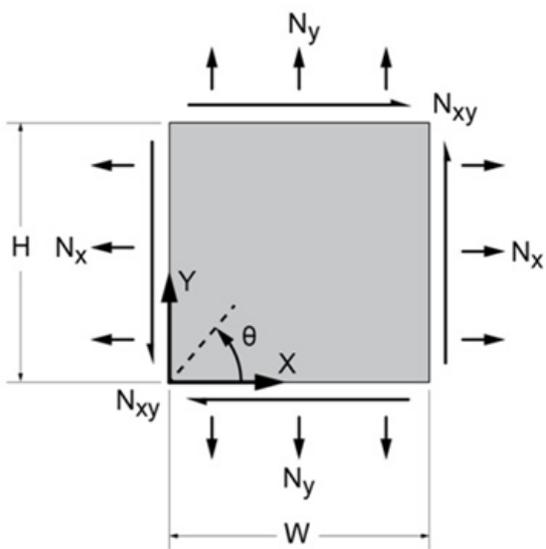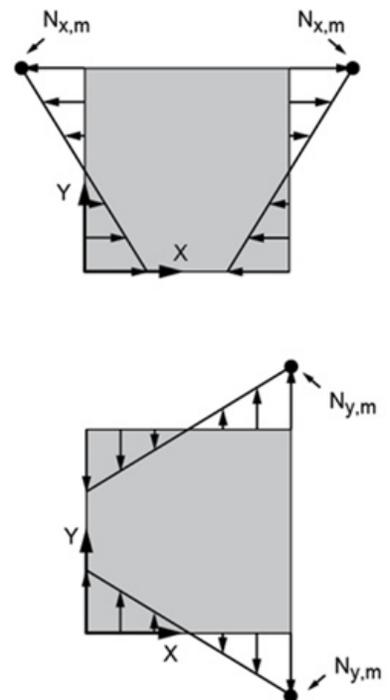
IN-PLANE MOMENT LOADING

The Extended Manual has further information about moment loading.

BOUNDARY CONDITIONS

The boundary conditions are shown in the figure below. At the bottom left corner (X=0, Y=0), the plate is constrained in the 1- and 2-directions (X- and Y-directions). On the top left corner (X=0, Y=Height), only the 1-direction (X-direction) is constrained. This allows the plate to be constrained from rigid body motion (at least in the X-Y plane) without overconstraining the model (for the most part – see the note below).
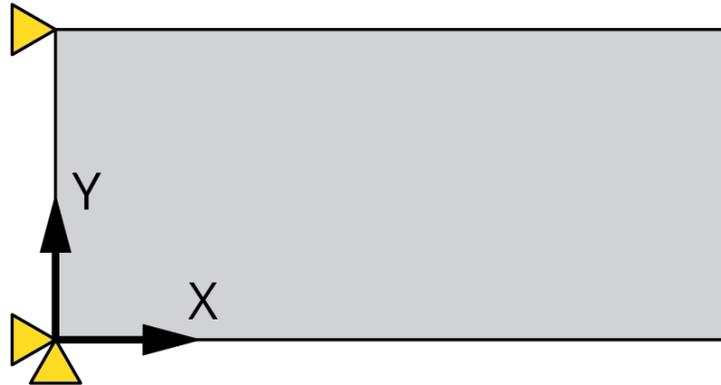
Note that depending on the applied loading, there may be loads applied to the same nodes (and same direction) as the constrained nodes. However, in such a case, the constrained DOF overrides the applied load (the load is essentially removed). In reality, these constraints are "minor" overconstraints. While they do not affect the deflected mode shape, the actual applied load is indirectly "created" by the constraint (as opposed to directly applying the load). However, provided the mesh density is "reasonable" and the loading is balanced otherwise, two things occur. (A) the reaction forces at the constraints should be close the ideal applied loads at these nodes and (B) the majority of the applied loading is still "pure". In addition, the weighting of the nodes at the corners is less than for the interior, reducing the necessity to make the loading "pure". Based on the various validation cases (see the relevant document), the program is highly accurate with this approach.

An alternate approach would be to apply "soft springs" to nodes that are interior to the edge. The advantage is that this ensures that the edge loading is "pure". However, there are two potential disadvantages. If the springs are too soft, there can be excessive rigid body motion. The initial Eigenvalues will also be related to rigid body motion. If the springs are too stiff, they can affect the stiffness of the model and the Eigenvalue will not predicted accurately. In general, there is a broad range of acceptable spring values, but this program uses the previous approach of constraining the corners.

Lastly, rigid body motion in the Z-direction and rotation of the X- and Y-axis must be prevented. This is accomplished by ensuring there is at least one clamped edge or at least two simply supported (SS) edges. The program will notify the user if this condition is not met.

CURVATURE

Curvature may be applied in one or both directions. The input is the radius of curvature and the directions are defined in the figure below.

The curvature should not be so large as to distort the mesh (i.e. close to that of a cylinder). The curvature may be verified by viewing the mesh via "3 – Show Deflection", via the CGX PostProcessor, or via the "Nodes" sheet.
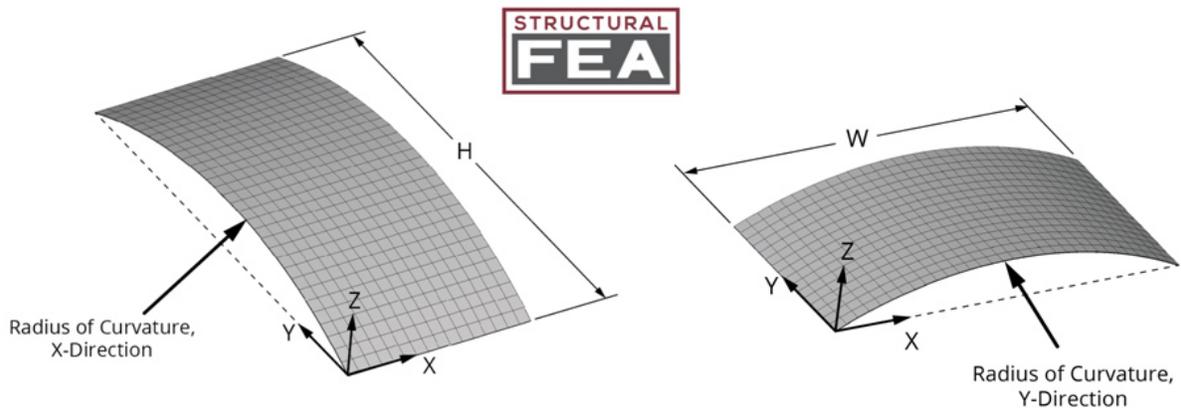
MATERIALS (GENERAL)

Only the chosen material is used to analyze the problem. The users may choose "A", "B", or "C", which correspond to Isotropic, Orthotropic, and Composite.

If "A" is chosen, all 3 values must be entered in the "Option A – Isotropic" section.
If "B" is chosen, all 7 values must be entered in the "Option B – Orthotropic" section.
If "C" is chosen, all 7 values must be entered for Mat 1. Mat 2, Mat 3, and Mat 4 are only used if they are called in the layup. In the layup section, at least one ply must exist (consisting of an entry for the Material Number and Theta (in degrees).

COMPOSITES (GENERAL)

Mat 1, Mat 2, Mat 3, and Mat 4 are considered to be "Materials" even though they have an associated thickness (which is not a material property). This is done for convenience because a ply usually has an associated thickness. In turn, the layup entry is more convenient because thickness is not required (and does not have to be matched up to the material, which can cause entry error).

If "C" is chosen (for Composite), QUAD8 must be chosen and Reduced Integration must be chosen. These are requirements from the CalculiX solver.

For an isotropic material, a positive shear load (+Nxy) will have the same Eigenvalue as an equal magnitude negative shear load (-Nxy). However, this is not the case for general composite laminates. Because of this, a note will appear as follows:

> NOTE: For composite laminates, the sign of the applied shear usually affects the result (unlike for an isotropic material). Highly recommended to run with the opposite shear sign.

TRANSVERSE SHEAR STIFFNESS

The transverse stiffness is affected by Gxz and Gyz. For isotropic materials, Gxy=Gxz=Gyz. In this case, the buckling load is not affected by the transverse shear stiffness because Gxz and Gyz are relatively large.

However, if Gxz and/or Gyz are relatively small, the buckling load can be affected. If an orthotropic material is selected, these values can be different than Gxy.

A common scenario where the transverse shear stiffness comes into play is for composites (especially sandwich structures). For composite laminates, the transverse shear stiffness is affected by G13 and G23 of plies. G13 and G23 are typically small compared to the in-plane shear stiffness of the laminate (G13 and G23 are dominated by the matrix). For solid laminates, the effect of transverse shear stiffness may (or may not) affect the buckling load (function of several factors). This effect is more significant for thicker laminates. *Practical Analysis of Aircraft Composites* by Esp (Chapters 17) discusses some of the parameters/conditions where transverse shear stiffness has an effect for solid laminates. For sandwich structures, the transverse shear stiffness often affects the buckling capability. This is because the core has relatively low values for G13 and G23 (and sandwich structures are usually much thicker than solid laminates). *Practical Analysis of Aircraft Composites* by Esp (Chapters 18) discusses this phenomenon and how is relates to the buckling of sandwich structures.

MESSAGE AND NOTES

Various notes may appear below the Boundary Condition/Loading region of the Excel file, as shown below. This section provides additional detail about these notes.

"Solution appears to be successful."

This note indicates that a result was found and that it passed the MODE VALIDITY CHECK (See the Extended Manual for further information). However, this does not guarantee the mode is correct. It is recommended to visually verify the mode is reasonable for thick plates (or other unknown scenarios where the correct mode may not be detected).

"Curvature has been applied."

This note appears to warn the user that the plate is not flat and that non-zero values are used for the curvature.

"NOTE: For composite laminates, the sign of the applied shear usually affects the result (unlike for an isotropic material). Highly recommended to run with the opposite shear sign."

This note is self-explanatory.

"NOTE: Eigenvalue is negative and not reported. See the manual for further information"

If the first mode is not positive, this indicates that buckling would not occur with the given loading. For example, a tension loaded plate. Check the load inputs. The Extended Manual discusses why negative Eigenvalues are not reported.

"Solution not successful. This mode does not appear to be valid. It has a relatively small Z-disp."

The MODE VALIDITY CHECK was not passed (See the Extended Manual for further information). This may occur if the plate is very thick (usually unreasonable thick).

"Solution not successful. Check the inputs. If this error persists, contact Structural FEA, LLC."

An unknown error occurred.

SCALE FACTOR FOR DISPLACEMENT

A "Scale Factor" value of 10 sets the maximum Z-displacement (for "3 – Show Deflection") to be 20% of the maximum of the Width or Height.

After changing the scale factor, the button "3 – Show Deflection" must be activated to update the deflected image.


CGX POSTPROCESSOR

After a model has been run (either via "2 – Run/Update" or "2 – Run/Update and Plot", the CGX PostProcessor may be opened by using the "4 - Veiw in CGX PostProcessor.

Zoom = middle mouse button
Rotate = left mouse button
Pan = right mouse button

Upon clicking to the left of the model, a menu will appear. To view the elements, select "Viewing/Toggle Element Edges"
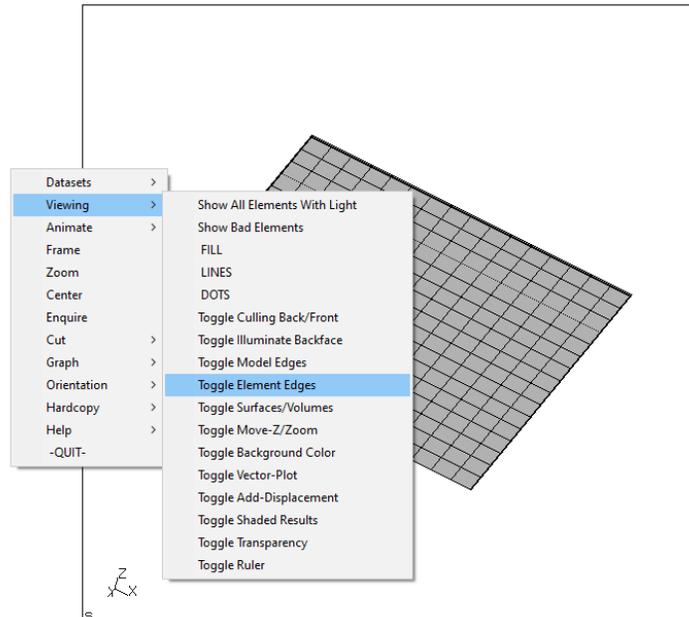
To view the mode shape, first select "Datasets / 2L X / 5 DISP", where X is the Eigenvalue. This makes the first mode shape active. The "1L" dataset is a static step, which is not a mode shape.
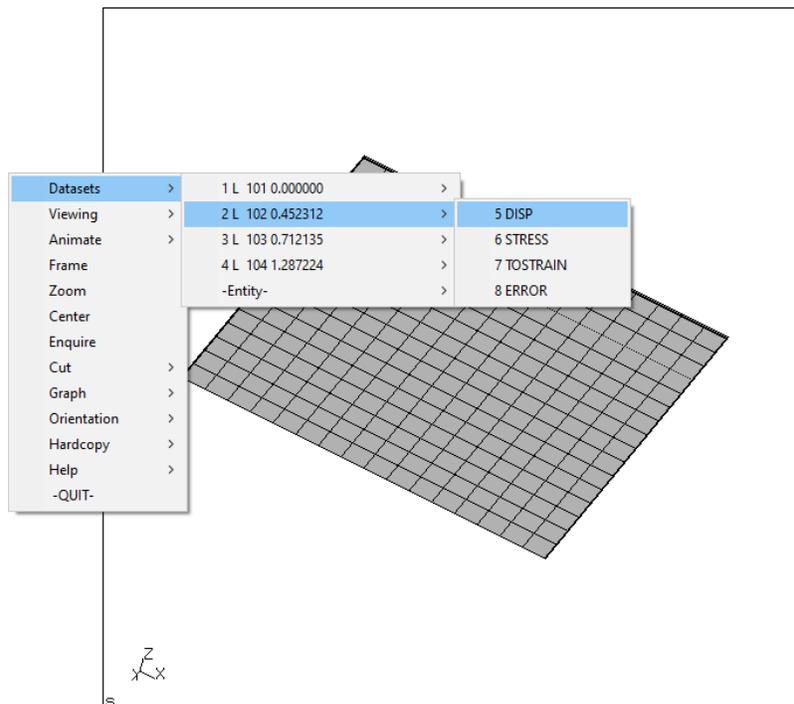
The mode shape can be animated by selecting "Animate/Start"
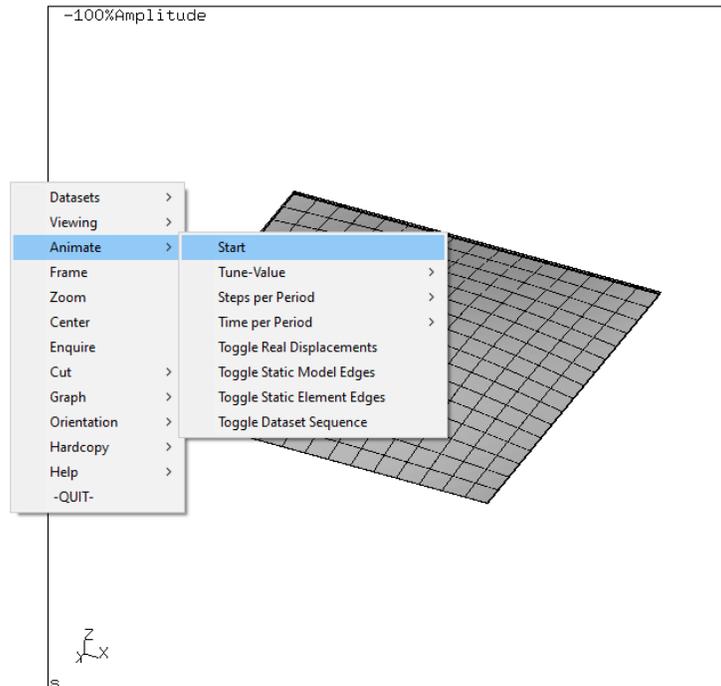
To view a fringe plot, first select the dataset (previously discussed). Then select "Datasets / -Entity- / 4 ALL".
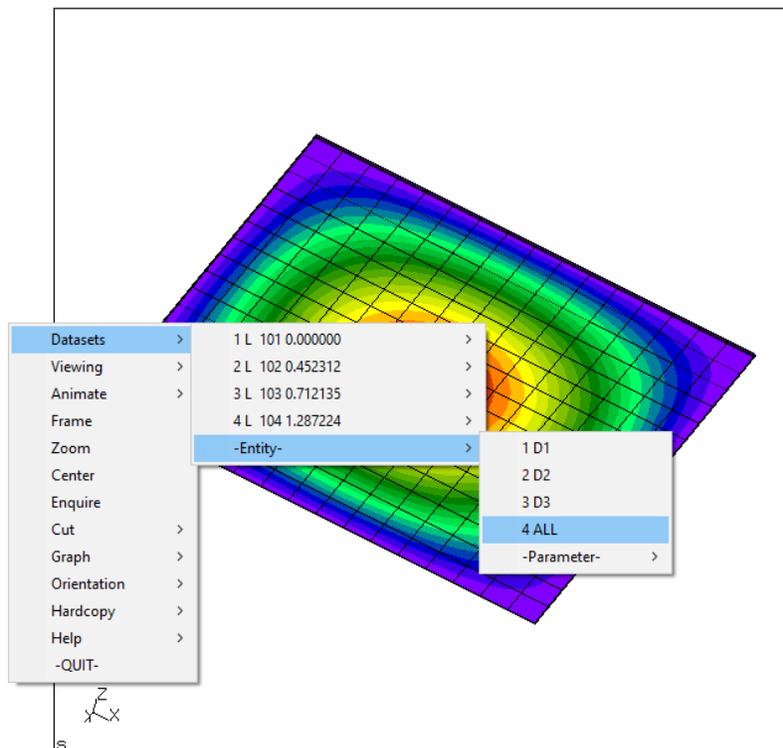
Three (3) mode shapes are requested from the solver (Datasets 2, 3, and 4). The other mode shapes may be viewed by selecting the appropriate data set. The Eigenvalues for each mode are also shown. For the above picture, the Eigenvalues for the first 3 modes are (0.452312, 0.712135, 1.287224) which correspond to Datasets (2, 3, and 4).

There is more information about the CGX PostProcess in the Extended Manual.

PLATE BUCKLING VERSUS COLUMN BUCKLING

For a column Eigen solution analysis, the Eigenvalue prediction is the maximum capability of the column (and Eigen buckling analysis is only valid for long columns that buckle elastically). Furthermore, column buckling can sudden an associated with a large amount of deflection. Therefore, relatively large safety factors are often used for column buckling analysis that is predicted via an Eigen solution (these also account for imperfections, load eccentricity, etc.). Also note that "short" or "intermediate" columns (Johnson columns) do not buckle elastically, and the Eigenvalue prediction can be much larger than the actual capability. However, that is a distinct phenomenon that can not be compensated by applying a safety factor to an Eigen buckling analysis.

Plates usually exhibit different behavior than columns. This is because they can often carry a load great than the initial buckling load. The maximum load carrying capability is also less sensitive to the initial imperfections and eccentricities of the load (even though those effects can change the initial bifurcation buckling point). Because of this, an Eigen buckling analysis for plates is often a useful analysis that may not require a relatively large safety factor (or no safety factor at all because of the post-buckling capability). Furthermore, most plates are thin and buckle elastically (unless they are relatively thick). This effect also makes an Eigen buckling analysis for plates relatively useful and practical. For additional discussion, see also *Practical Analysis of Aircraft Composites* by Esp (Chapter 17).