

# PlateMesh© Extended Manual



Document Version 1.1 (4/7/2020)

This provides additional details about the inner workings of the program.

## CCX and CGX PROGRAMS

CalculiX CrunchiX (CCX) is a finite element solver with an input deck similar to ABAQUS.

CalculiX GrpahiX (CGX) is a pre/post processor that can read an FRD file created by CCX.

## RELEVANT FILES

The required input/outputs are located in the same directory that CalculiX is in. *The PlateMesh©* program files are preceded by an underscore so that they can be easily sorted.

**\_READ\_FIRST\_StructuralFEA\_Notes.txt** = notes about the CalculiX distribution from StructuralFEA.

**2.16\_Notes.txt** = notes about the 2.16 distribution of CalculiX (and some various end user options for CCX and CGX).

**\_PlateMesh.inp** = input file (also known as the deck). This input file is created for CalculiX, but is very similar (if not the same) as an input file ABAQUS.

**\_PlateMesh.dat** = output file 1 of 2. Contains displacements from the static step and the Eigenvalues and Eigenvectors for the first 3 modes.

**\_PlateMesh.frd** = output file 2 of 2. Contains node and element data needed to post-process the results in CGX. Also contains displacements, stresses, strains, Eigenvalues, and Eigenvectors, which can be accessed in CGX.

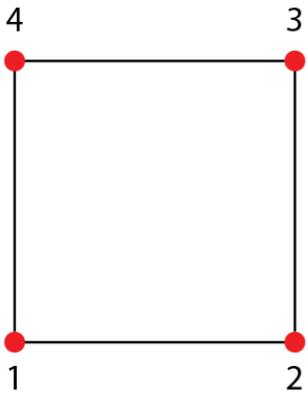
**\_PlateMesh.fbd** = if an image (deflection or fringe) is requested, this file created, which has a series of steps passed to CGX that create the image that is passed to Excel.

**\_PlateMesh.png** = if an image (deflection or fringe) is requested, this file created and passed to the Excel file.

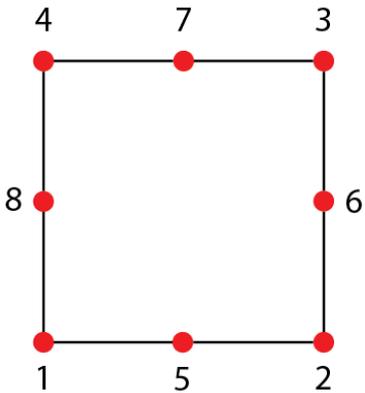
# QUAD4 AND QUAD8 NODE NUMBERING SEQUENCE

To define the element, the nodes must be listed in the correct sequence. For the QUAD4, the sequence is shown below (the node numbers run counter clockwise). See also the next section.

For the QUAD8, the sequence is shown below. The corner node numbers (4 corners) run counter clockwise in sequential order. The mid-side node numbers (4 mid-side nodes) start after the corner nodes and between nodes 1 and 2. They also run counter clockwise, as shown. See also the next section.



**QUAD4**

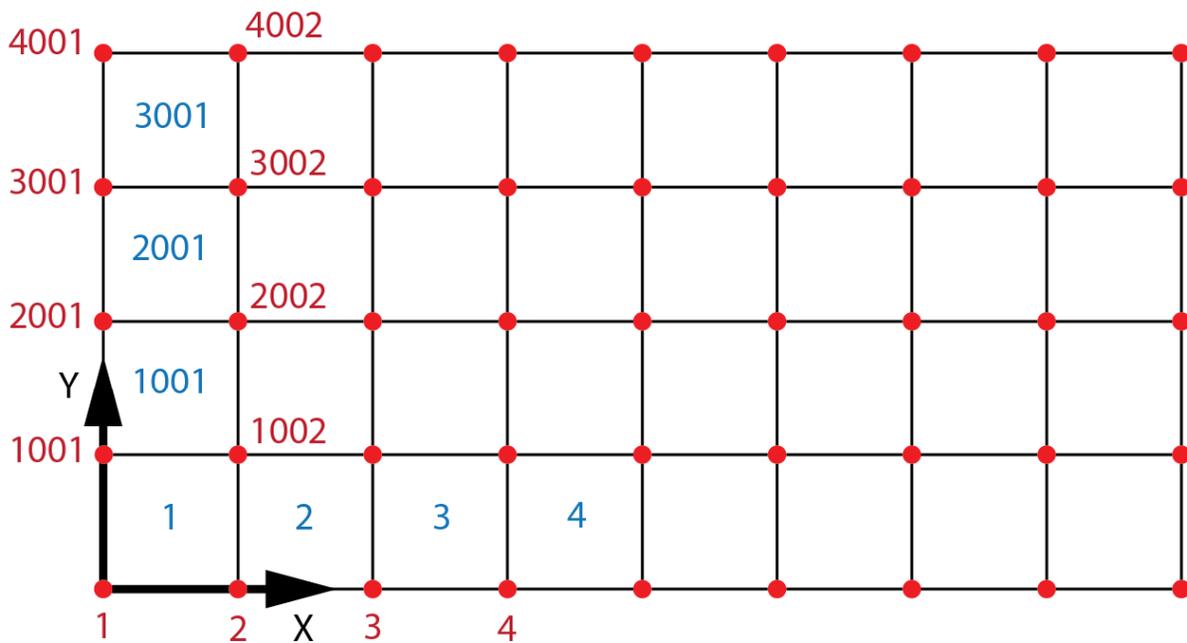


**QUAD8**

## NODE AND ELEMENT DEFINITIONS – QUAD4

The node numbering starts at the bottom left corner of the plate ( $X=0, Y=0$ ). Nodes increase by 1 as they move from left to right (see the figure below). Nodes increase by 1000 as they move from bottom to top (see the figure below).

To define the element, the first entry is the element number, followed by the 4 nodes. As shown in the figure below, the entry in the INP file for Element 1 is 1, 1, 2, 1002, 1001.



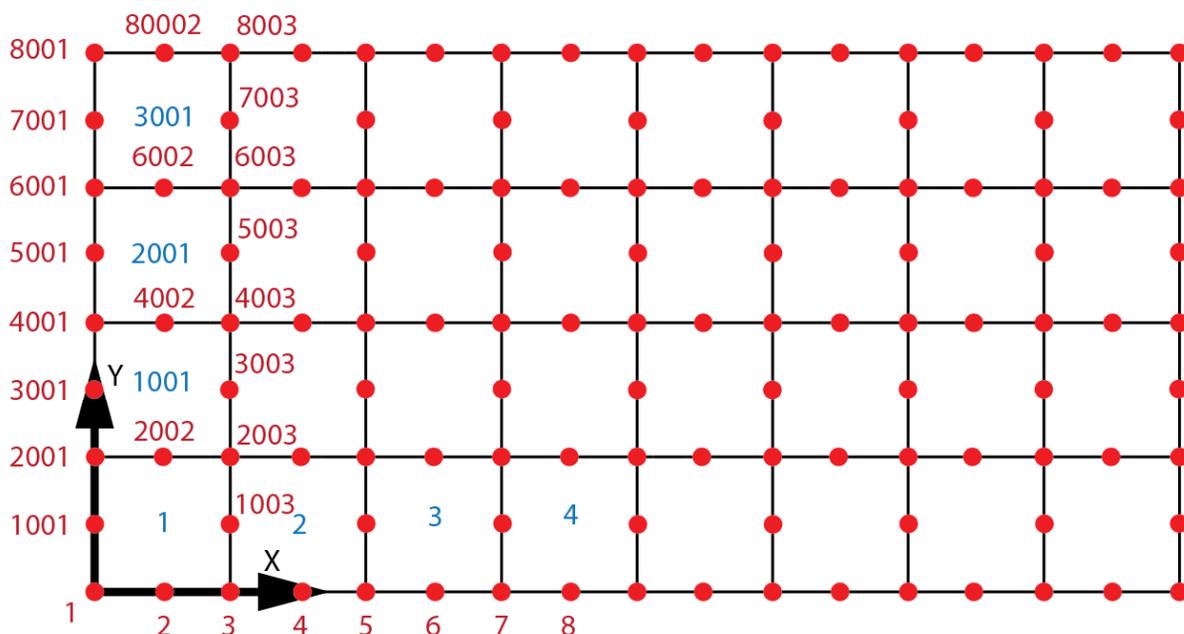
Element 1 : 1, 1, 2, 1002, 1001

Element 2001: 2001, 2001, 2002, 3002, 3001

## NODE AND ELEMENT DEFINITIONS – QUAD8

The node numbering starts at the bottom left corner of the plate ( $X=0, Y=0$ ). Nodes increase by 1 as they move from left to right (see the figure below). Nodes increase by 1000 as they move from bottom to top (see the figure below).

To define the element, the first entry is the element number, followed by the 8 nodes. As shown in the figure below, the entry in the INP file for Element 1 is 1, 1, 1, 3, 2003, 2001, 2, 1003, 2002, 1001.



Element 1 : 1, 1, 3, 2003, 2001, 2, 1003, 2002, 1001

Element 2001: 2001, 4001, 4003, 6003, 6001, 4002, 5003, 6002, 5001

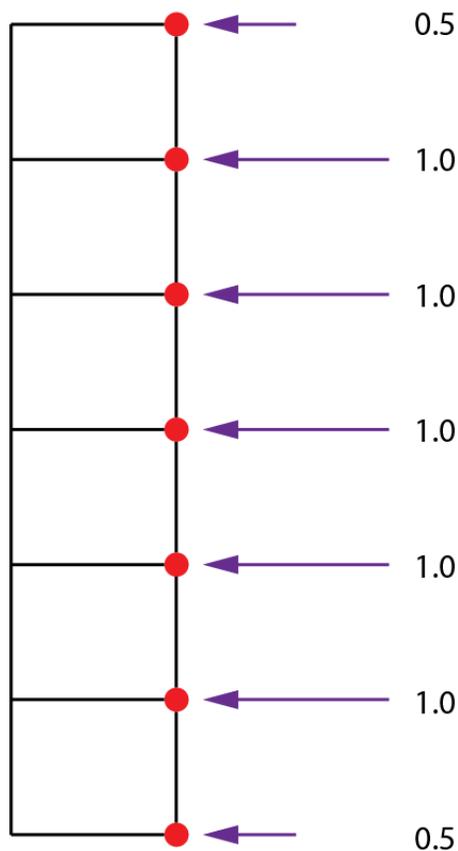
## MAXIMUM MESH SIZE

A maximum of 20,000 elements are allowed if QUAD4 is selected. A maximum of 10,000 elements are allowed if QUAD8 is selected. For both of these cases, these values are very large compared to the required mesh density for convergence.

## EDGE LOADING AND NODE WEIGHTING – QUAD 4

For the QUAD4, the nodes at the corners have a weighting factor that is half as much as the interior nodes (See the figure below). This applies to all loading scenarios ( $N_x$ ,  $N_y$ ,  $N_{xy}$ , and in-plane moments). This is an artifact of the stiffness of the shell elements.

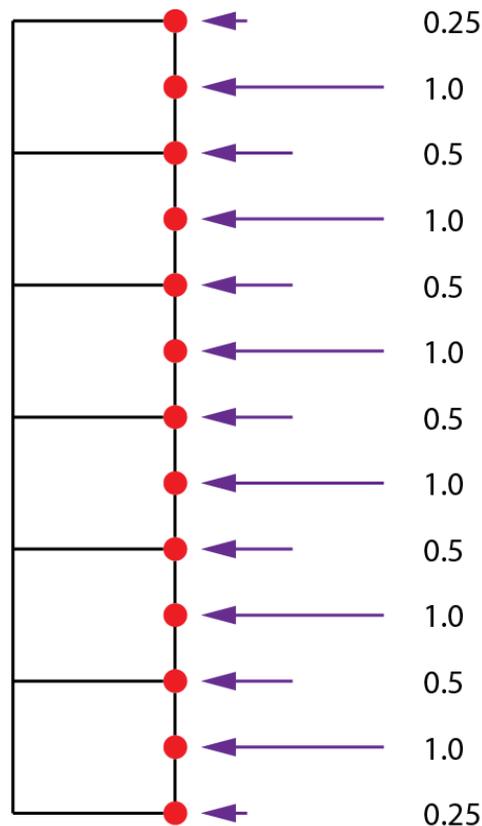
As an example, assume that  $N_x = -100.0$  and the Height is 2.0. The total applied load is  $-200.0$  on the right edge and for the picture, and there are 6 elements (not enough for convergence, but sufficient for demonstration). The corner nodes have an applied load of  $-16.666$  and the interior nodes have a value of  $-33.333$ . The total load is  $-16.666 \times 2 + (-33.333) \times 5 = -200.0$ . It can be verified in the CGX post-processor ("Datasets/ 1L 101"), which is the static step, that edge deforms in a straight line (as opposed to a jagged edge deformation). This indicates that the nodal weighting has been applied correctly.



## EDGE LOADING AND NODE WEIGHTING - QUAD 8

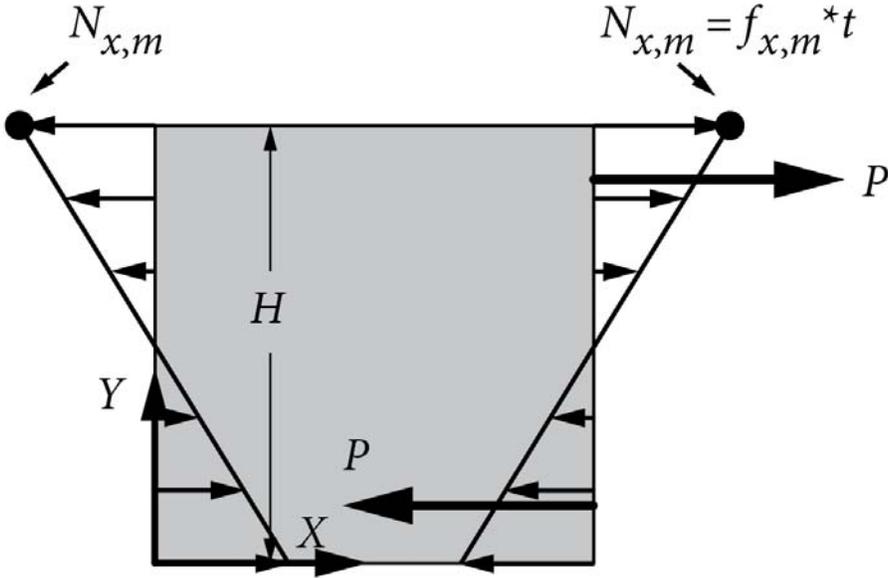
For the QUAD8, the nodes at the corners have a weighting factor that is 0.25 as much as the mid-side node (See the figure below). The nodes that share an adjacent element have a weighting factor that is 0.5 as much as the mid-side node (See the figure below). This applies to all loading scenarios ( $N_x$ ,  $N_y$ ,  $N_{xy}$ , and in-plane moments). This is an artifact of the stiffness of the shell elements.

As an example, assume that  $N_x = -100.0$  and the Height is 2.0. The total applied load is -200.0 on the right edge and for the picture, and there are 6 elements (not enough for convergence, but sufficient for demonstration). The corner nodes have an applied load of -5.555. The mid-side nodes have a value of -22.222. The shared nodes have a value of -1.111. The total load is  $-5.555 \times 2 + (-22.222) \times 6 + (-1.111) \times 5 = -200.0$ . It can be verified in the CGX post-processor ("Datasets/ 1L 101"), which is the static step, that edge deforms in a straight line (as opposed to a jagged edge deformation). This indicates that the nodal weighting has been applied correctly.

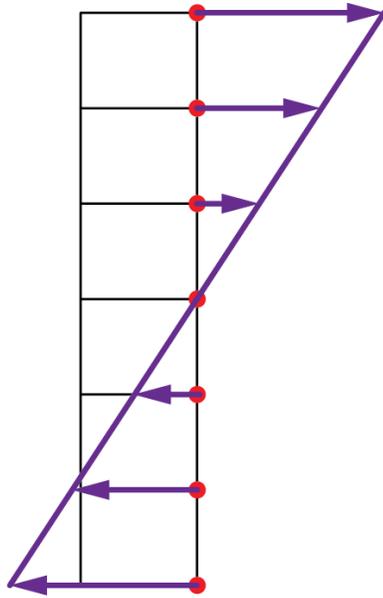


# IN-PLANE MOMENT LOADING

A figure of the in-plane moment loading is shown below (loading applied to the actual plate – not the loading applied to the finite element model – see below). In this case, the loading has X-direction values and the maximum running load (top and bottom locations) is designated by  $N_{x,m}$ . In other words, the stress at the top edge (and bottom edge) of the plate is  $f_{x,m}$  ( $N_{x,m} = f_{x,m} * t$ , where  $t$  is the total thickness of the plate).



Internal to the program, an in-plane moment is applied by first creating a linear distribution of loading, as shown in the figure below. However, as mentioned in the previously two sections, nodal weight factors must be considered. These weight factors are applied the linear distribution. Finally, an internal check is performed to ensure the equation shown in STEP 3 is satisfied (shown in the figure below).



**STEP 1 = Linear Distribution as shown in the figure to the left**

**STEP 2 = Apply nodal weighting factors (QUAD4 and QUAD8 are different)**

**STEP 3 = Ensure this equation is satisfied**

$$M = f_{x,m} t \cdot \frac{H^2}{6} = N_{x,m} \cdot \frac{H^2}{6}$$

The equation in STEP 3 above is determined by the steps shown below. The total load on the top half,  $P$ , is determined by using the maximum stress ( $f_{x,m}$ ) and a triangular distribution (see the figure below as well):

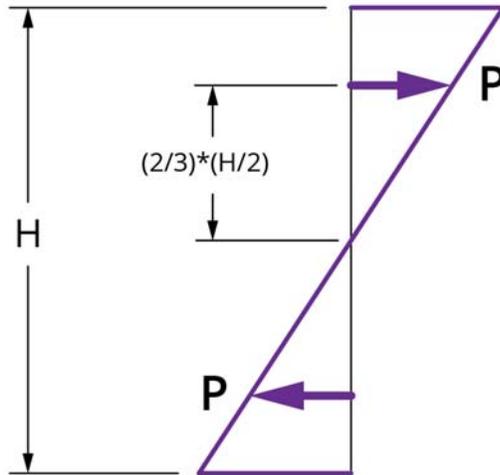
$$P = \frac{1}{2} \left( f_{x,m} t \frac{b}{2} \right)$$

The total moment is determined by  $P$  and the geometric parameters (see the figure below as well):

$$M = 2 \left( \frac{2}{3} \cdot \left( \frac{H}{2} P \right) \right)$$

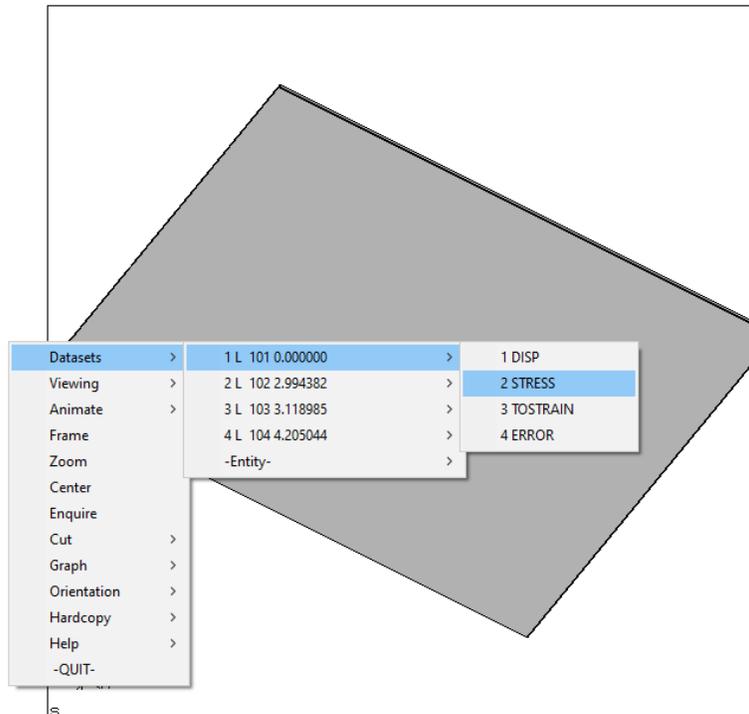
Finally, the total moment is related to  $N_{x,m}$  as follows:

$$M = f_{x,m} t \cdot \frac{H^2}{6} = N_{x,m} \cdot \frac{H^2}{6}$$

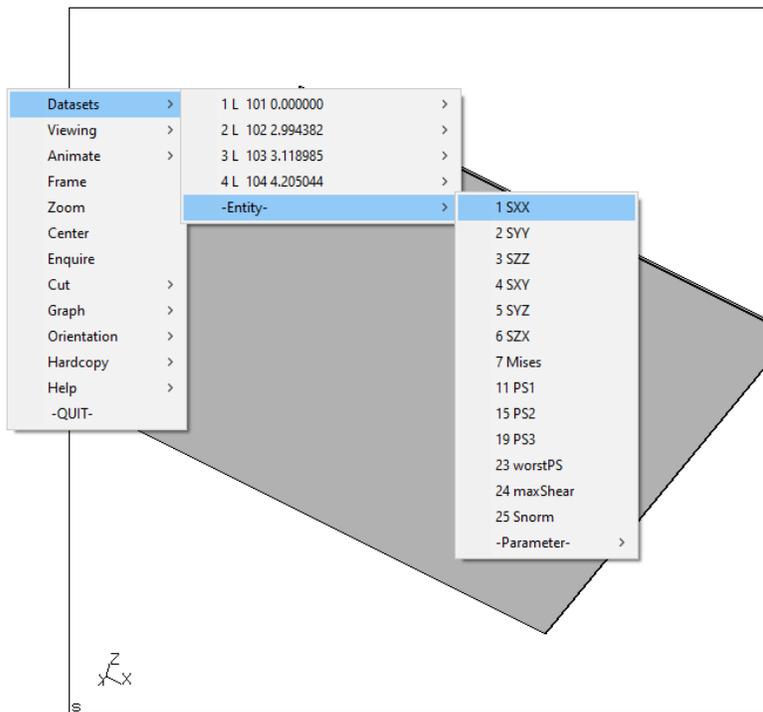


To verify the moment is correct, an example is shown below.  $N_x, m = 100.0$  and the thickness is  $0.05$ . Therefore, the X-direction stress should be  $+2000.0$  at the top and  $-2000.0$  at the bottom. The plate is  $10.0 \times 10.0$  and the parameters are QUAD8 with 10 elements along the edge is used.

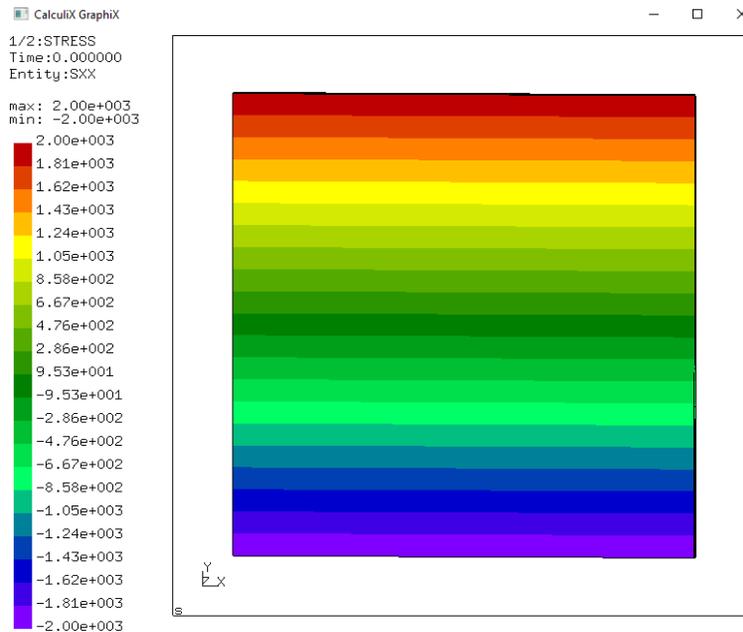
Using the CGX PostProcessor, the stress for static step can be selected via "Datasets / 1L X / 2 STRESS", as follows:



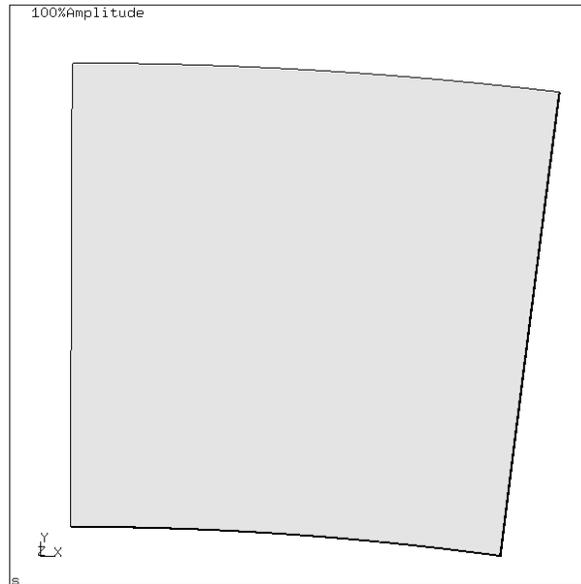
The X-direction stress can be displayed by selecting “Datasets / -Entity- / 1 SXX”, as follows:



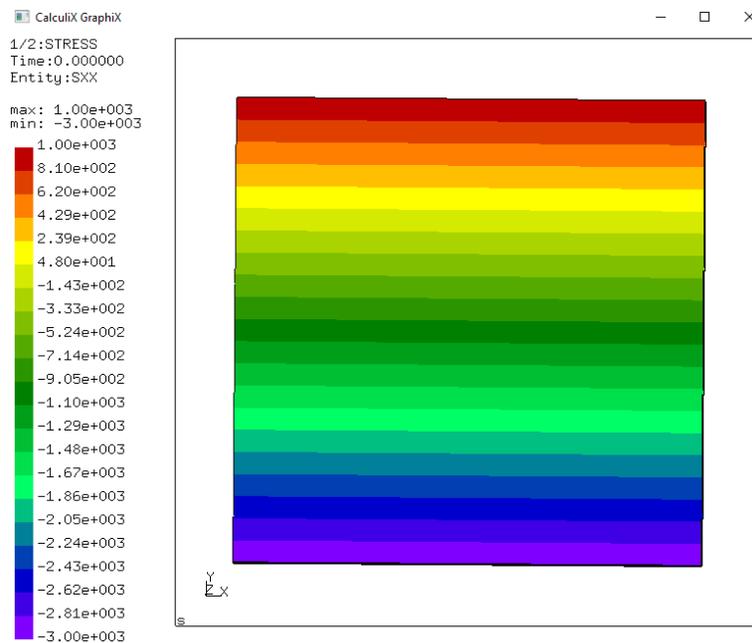
The stress at the top and bottom is as expected and the gradient is linear, as shown below. This indicates that the nodal loading is correct.



As another check, the static deformation (not the buckled shape) is as expected. Also, since the edge deforms in a straight line, both the distribution and the weighting has been applied correctly (weighting is different for QUAD4 and QUAD8).



Lastly, if  $N_x = -50.0$  (running load) is applied in addition to  $N_{x,m} = -100.0$  (in-plane moment), the stress at the top is expected to be  $+1000.0$  and the stress at the bottom is expected to be  $-3000.0$  ( $t = 0.05$ ). The expected result aligns with the following fringe plot, indicating that both loads have been applied (and superimposed) correctly.



## **ORTHOTROPIC MATERIAL (SPECIAL INPUT NOTES)**

For shell elements, CalculiX uses a “non standard” approach of expanding the shell elements into solid elements. One advantage of this is that transverse shear deformation effects are simple to implement.

However, for the CalculiX shell element, an orthotropic material must be fully defined in 3 dimensions (and  $E_z$  is part of the solution). For “standard” shell elements, there would be no need to define a property for  $E_z$ . The inputs are  $E_x$  and  $E_y$ ; and  $E_z$  is internally chosen (and passed to the input file) to be the minimum of  $E_x$  and  $E_y$  (user does not input  $E_z$ ). For an unknown reason, if  $E_x$  and  $E_y$  are very different and  $E_z$  is chosen to the average (or the maximum) of  $E_x$  and  $E_y$ , the solution will not converge. However, this issue has not been observed when  $E_z$  is defined to be the minimum of  $E_x$  and  $E_y$ .

Again, since the orthotropic material must be fully defined in 3-dimensions,  $\nu_{xz}$  and  $\nu_{yx}$  are internally defined to be  $\nu_{xy}$  (passed to the input file).

$G_{xz}$  and  $G_{yz}$  are user inputs and affect the transverse shear stiffness. For an isotropic material, these values are equal to  $G_{xy}$ . However, for a typical composite laminate,  $G_{xz}$  and  $G_{yz}$  are relatively low because they are dominated by the matrix. Because these values are relatively low, they can affect the Eigenvalue for some problems. This usually occurs for thicker laminates or for sandwich structures (See also *Practical Analysis of Aircraft Composites* by Esp, Chapter 17).

## **COMPOSITES (INPUTS)**

The following discussion is the similar to the one in the section “ORTHOTROPIC (SPECIAL INPUT NOTES)”

For shell elements, CalculiX uses a “non standard” approach of expanding the shell elements into solid elements. One advantage of this is that transverse shear deformation effects are simple to implement.

However, for the CalculiX shell element, an orthotropic material must be fully defined in 3 dimensions (and  $E_3$  is part of the solution). For “standard” shell elements, there would be no need to define a property for  $E_3$ . The inputs are  $E_1$  and  $E_2$ ; and  $E_3$  is internally chosen

(and passed to the input file) to be the minimum of E1 and E2 (user does not input E3). For an unknown reason, if E1 and E2 are very different and E3 is chosen to the average (or the maximum) of E1 and E2, the solution will not converge. However, this issue has not been observed when E3 is defined to be the minimum of E1 and E2.

Again, since the orthotropic material must be fully defined in 3-dimensions,  $\nu_{13}$  and  $\nu_{23}$  are internally defined to be  $\nu_{12}$  (passed to the input file).

G13 and G23 are user inputs and affect the transverse shear stiffness. For an isotropic material, these values are equal to G12. However, for a typical composite laminate, G13 and G23 are relatively low because they are dominated by the matrix. Because these values are relatively low, they can affect the Eigenvalue for some problems. This usually occurs for thicker laminates or for sandwich structures (See also *Practical Analysis of Aircraft Composites* by Esp, Chapter 17).

## **EIGEN SOLUTION - GENERAL**

To perform an Eigen solution, a static step is first need. This is shown in the output files (.frd and .dat files) and can be accessed in the CGX PostProcessor.

The second data set is the first Eigen solution. Three (3) Eigen solutions are requested in the INP file, but only first one is reported in the *PlateMesh* program. The other two solutions can be accessed in the output files (.frd and .dat files).

## **NEGATIVE EIGEN VALUES**

The Lanczos method is used for the Eigen solution. However, the Lanczos method may “miss” modes if the Eigenvalues are negative. Therefore, if the first Eigenvalue is negative, it is not reported. For example, if a value of  $N_x=+100.0$  is applied (without any other loads), no buckling will occur (tension loaded plate). However, a negative Eigenvalue indicates that the load is reversed. To resolve this issue, ensure that the loading is such that a positive Eigenvalue occurs.

# MODE VALIDITY CHECK

A model validity check is performed to help determine if the mode is valid. For example, if there is not a significant amount of Z-displacement (but a large amount of X and/or Y displacement), the mode is not a buckled mode. This is not common, but may occur if the plate is very thick. The validity check is as follows:

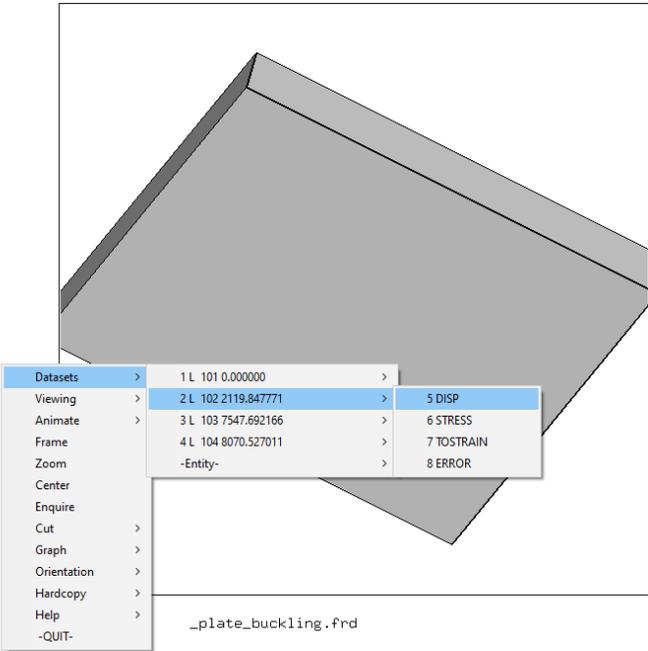
X = X-displacement for the first mode, Y = Y-displacement for the first mode,  
Z = Z-displacement for the first mode

$$M = Z / \text{SQRT}(X^2 + Y^2)$$

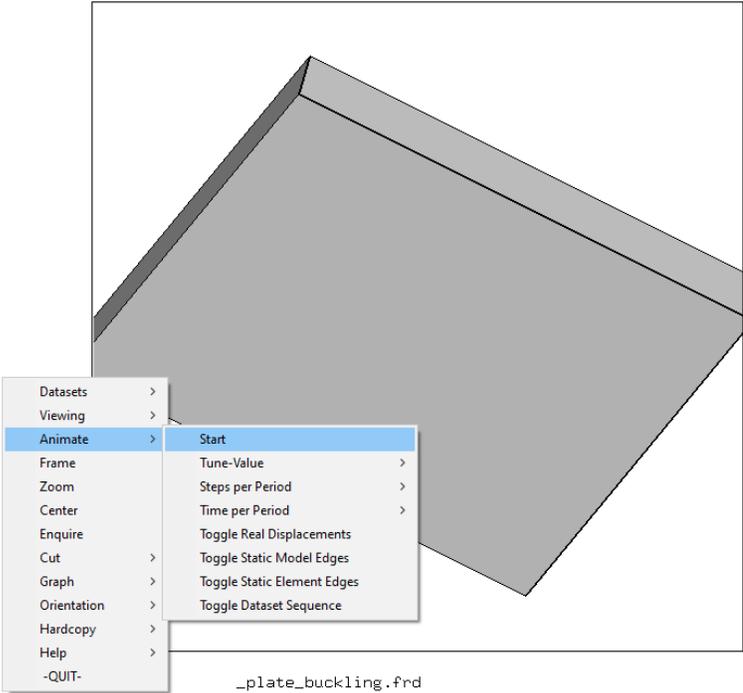
M1 = ABS (MAX of M for all nodes)  
M2 = ABS (MIN of M for all nodes)  
M3 = MAX(M1, M2)

If  $M3 < 0.1$  then this is not considered to be a valid mode. In other words, the Z-displacement is relatively small compared the X- and Y-displacements.

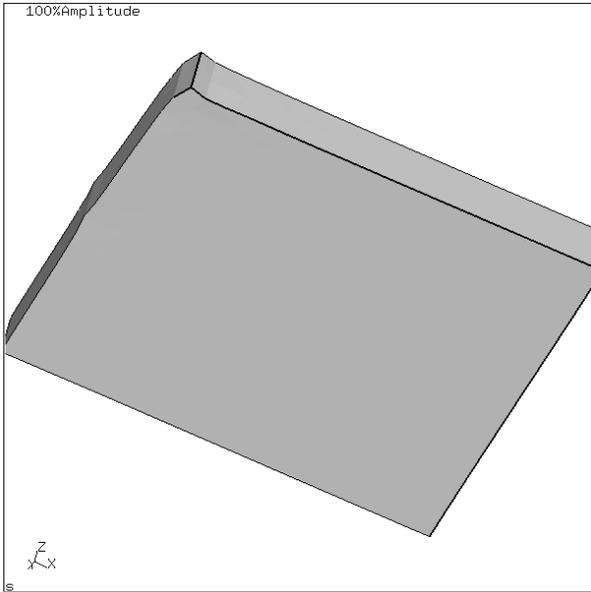
If a mode is reported as invalid, the Scale Factor may need to be very large to observe the deformed shape. Alternatively, the user can view the deformation in the CGX PostProcessor. As an example, the following figure shows a thick plate that has an invalid first mode. The first mode is "Datasets/ 2L 102 X" and start first by selecting the displacement as "Datasets/ 2L 102 X / 5 DISP".



After selecting the second dataset (the first mode) as shown above, select “Animate/Start”, as shown in the following figure.



The deformed shape is shown below, which is not a valid buckling mode. Rather, the plate is deforming primarily in the plane (with little out of plane deformation).



## VALIDATING THE APPLICATION OF LOADS

One way for the user to validate that the load is applied correctly is to look at the stresses for the static step. The stress is related to the running load by dividing the running load by the total plate thickness.

Step 1a. Ensure that an isotropic material is selected. If a composite is used, the results may not be as expected. This procedure is only used to validate the load application so it is acceptable to use an isotropic material even if the actual buckling solution is for a composite plate.

Step 1. Open the CGX PostProcessor "4 - View in CGX PostProcessers"

Step 2. Select "Datasets/ 1L 101 0.00000 / STRESS"

Step 3. Select "Datasets/ -Entity- / (select SXX, SYY, or SXY)"

Another approach is to view the deformation, as follows. Aside from the overall deformation, the edge deformation can be observed to ensure it is not "jagged"

Step 1. Open the CGX PostProcessor "4 - View in CGX PostProcessers"

Step 2. Select "Datasets/ 1L 101 0.00000 / DISP"

Step 3. Select "Animate/Start"

## SCALE FACTOR – INTERNAL ADJUSTMENT

The "Scale Factor" input has a default value of 10.0 and can be negative. There is an internal algorithm that causes a Scale Factor of 10.0 to show the max Z-displacement as 20% of the maximum of width or height of the plate.

The raw output from the FRD file has its own scale factor. Using this scaling factor, an internal scaling factor is determined to meet the conditions above, which is then passed to the FBD file to generate the image.

## **EDGE CONDITION CHECK**

One of the two conditions must exist, otherwise rigid body motion will occur.

(A) At least one edge is constrained

(B) At least two edges are simply supported.

There is a check for this scenario.

## **NODES SHEET**

The “Nodes” sheet shows the node number and X-, Y-, Z-positions of the undeformed plate.

This sheet may be used for reference.

## **CURVATURE**

Reserved section. Additional information to be added later.