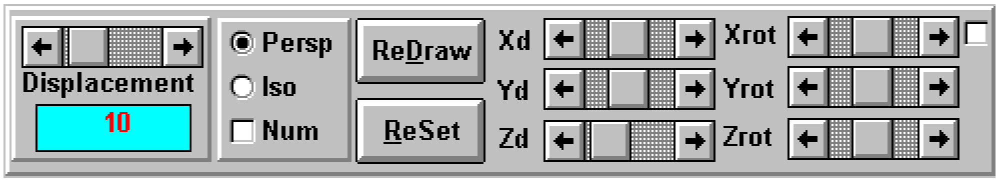
CADRE Help Contents

The Contents lists Help topics available for CADRE. Use the scroll bar to see entries not currently visible in the Help window.

To learn how to use Help, press ***F1*** or choose Using Help from the Help menu

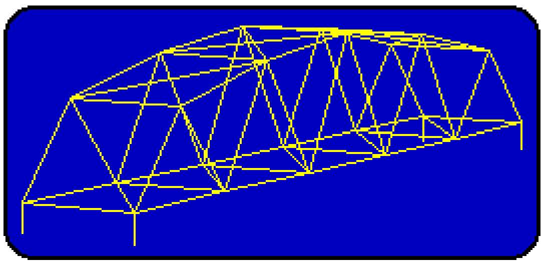
The help topics are divided into three sections:

Getting Started	Practice with sample problems
Help on Finite Element Analyses	Discussion of finite element analysis topics that are related to methods used by CADRE.
Help on using CADRE	Help topics for using CADRE to develop, solve and examine finite element models.


Displacement
10
Persp
Iso
Num
ReDraw
ReSet
Xd
Yd
Zd
Xrot
Yrot
Zrot

Exercise 1: Working with a finite element model

In this exercise you will learn how to examine, manipulate, and control a graphic image of a CADRE finite element model. The model is already constructed, solved, and stored in a file so you will not have to input any data. In order to provide instruction in all the available graphics controls, this exercise will employ a vibration model of a 3 dimensional bridge structure.



Additional exercises will provide instruction in building models, solving them, and examining the results. It is recommended that you print out this topic to make it easy to follow along as you perform the exercise.

Step 1 Loading and viewing the model

Pull down the ***File*** item on the menu bar and select ***Open File***.

In the Open Files dialog box select the directory [dynsamp] which contains the sample files for the *Dynamic* models. If you need help on this form press ***F1***.

Select the file named [bridge3d.fem] and open it.

The bridge should be displayed on the screen, ready to view. The Drawing Control Panel will also be displayed.

Step 2 Manipulating the model

Select the ***Iso*** box on the Drawing Control Panel and note the difference in appearance between the isometric and perspective,(***Persp***) views. Before continuing, select the ***Persp*** box again.

Using the right side of the ***Yrot*** scroll bar, rotate the bridge around the Y axis (the vertical axis of you screen) until you are looking directly across the bridge. The small check box in the upper right corner can be used to speed up the movement in response to the scroll bars.

Use the right button on the ***Zd*** scroll bar to approach and cross over the bridge, then retreat from the bridge with the left button. Note the way the perspective view behaves. Try the same with the isometric view but return to the perspective view before proceeding to the next step.

Press the ***Reset*** button to return the model to the original position. Now using the Yrot scroll bar click on the right side and rotate the model only half way to the end view. Adjust with a few clicks on the right side of the Xrot scroll bar to get a 3D view similar to the picture above.

Click on the check box beside the label, ***Num,*** to display the node numbers.

If you are using the factory default colors, the red numbers indicate the restrained nodes, the light blue numbers indicate the mass locations, and the yellow numbers indicate the unbound and unloaded (no loads or inertial forces) structural nodes.

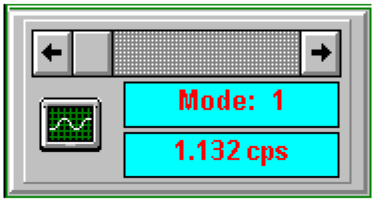
Now uncheck the ***Num*** box since the numbers will slow down other activities.

Step 3 Reloading previous data

Now load the previously calculated data. Pull down the ***File*** item on the menu bar and select ***Reload Results.***

In the Reload Results Dialog Box select the file "bridge3d.dta" (it should already be suggested so that you only have to select ***OK***).

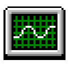
The bridge will be displayed already displaced in the shape of the first vibration mode. The Mode Panel will appear in the upper left corner, and the ***Displacement*** scroll bar will appear with a preset initial amplitude of 10 units.



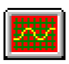
(Click on the figure above to identify the mode controls)

Step 4 Changing and animating modes

Press the ***Animate*** button on the Mode Panel to animate the displayed mode. The bridge should begin to vibrate in this mode. Notice that the ***Animate*** button is red when a mode is vibrating.



Animation stopped



Animation in effect

While animation is in effect, use the mode selector scroll bar to advance to higher modes. Notice that the vibrating frequency in cycles per second (cps) is given for each mode number.

You may need to change the amplitude with the ***Displacement*** scroll bar or rotate the model with the rotation scroll bars to get a clearer view of the modal character for some modes.

Note that many menu bar items are disabled while modes are being animated.

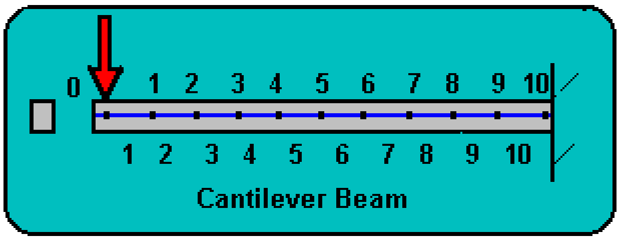
Press the ***Animate*** button to stop the animation.

You have now learned to examine, manipulate, and control a finite element graphic image. Experiment a little. If you get lost or confused during your adventure, click the ***Reset*** button. Also, try repeating the exercise with some of the other samples from the [dynsamp] directory.

When you are ready to build a model of your own. See Exercise 2.

Exercise 2: Building a static finite element model

In this exercise you will construct a cantilever beam finite element model which will represent the structure in this picture:



It is recommended that you print out this topic to make it easy to follow along as you perform the exercise.

Step 1 Starting a new model

Pull down the ***File*** item on the main menu and select ***New***.

When the title screen appears enter a title (for example: Cantilever Beam Problem).

In the file name text box enter:

[Cantbeam.fem]

This will be the DOS file name for the input data. You will select a directory for it later (the first time you save it).

Now press ***OK*** and advance to the nodal coordinate screen.

Step 2 Entering nodal coordinates

Nodal coordinates: There are eleven structural nodes and one reference node in this problem. The nodes will be identified as nodes 0 through 11.

The nodal coordinates are laid out by reference to a global coordinate system. For CADRE this system must be a right hand coordinate system.

In the picture of the beam above, positive X is considered to the right, positive Y is up, and positive Z is out of the screen toward the viewer.

All input data for this exercise will be in the English system of units (pound, inch, and second). The beam nodes will be represented by the following coordinates:

Ident	X(in)	Y(in)	Z(in)
0	0	0	0
1	12	0	0
2	24	0	0
3	36	0	0
4	48	0	0
5	60	0	0
6	72	0	0
7	84	0	0
8	96	0	0
9	108	0	0
10	120	0	0
11	0	1000	0

Now enter the nodal coordinates above as follows:

The Nodes Form, (similar to all the data entry forms) is set up to allow the entry of one node per page. The pages are turned by the scroll bar on the right edge of the form or the ***Next*** and ***Back*** buttons (or ***Page-Up*** and ***Page-Down*** Keys). In entry mode, a new page will automatically be added as you press ***Return*** or ***Enter*** key while in the last text box.

For example, for the first node, type 0 in the Ident box, press the ***Enter*** or ***Return keys***, type 0, press ***Enter*** or ***Return***, type 0 press ***Enter*** or ***Return***, type 0, press ***Enter*** or ***Return*** and you should be set up on page 2 and ready to enter the second line.

Continue to enter the data for all the nodes in the table in the same manner. For help while using the Nodes Form press ***F1***.

When all the nodes are entered, press ***OK*** to advance to the Elements Form.

After pressing ***OK***, a query box will ask you to confirm that you really want to leave the Nodes Form. Press ***Yes***.

Step 3 Entering element properties and connections

The beam properties are determined as follows:

The beam is 120 inches long. It is made of aluminum and has a constant square cross section of (.5 x .5 inches). It is fully restrained at one end and loaded with 1 pound on the free end. For the purposes of this exercise, the beam is considered weightless.

The elastic modulus, E, for this material is 10.5 million lb/in ^{2}.

The shear modulus, G, for this material is 4 million lb/in ^{2}.

The cross section area is 0.25 in ^{2}.

The formula for the area moment of inertia is:

$$I = \frac{bh^3}{12}$$

So I = 0.005208 in ^{4} for both Iy and Iz.

The polar moment of inertia is:

J = Iy + Iz is 0.0104167 in ^{4}.

The stiffness properties calculated from the above information are:

AE = 2625000.0 lb

EIy = 54687.5 lb·in ^{2}

EIz = 54687.5 lb·in ^{2}

JG = 41666.8 lb·in ^{2}

There are 10 straight beam elements, each 12 inches in length, and all with the same properties.

Enter the element properties as follows:

The element descriptions are entered one element per page just like the nodal coordinates were entered. For help while using the element form press ***F1***.

Element are identified automatically by a number formed from the nodes to which they connect (for example a beam connecting node 3 to node 4 will be numbered by CADRE as 0003.0004). These numbers are used as names to help identify individual elements when editing the data or viewing results.

Notice that the focus starts with the text box entitled "Origin node". Also note that the element type ***Straight beam*** is already selected. If you had wanted another element type you would use the drop down list to select an alternative.

In the next 3 connection boxes, type (or select from the list boxes) 0 for the origin node, 1 for the axis node, and 11 for the reference node. These nodes define the <u>beam element local coordinate system</u> for element number 0000.0001. Notice that the element identification number keeps pace with your selection of the end nodes.

Click on the ***Element type*** box to see the types of elements available in CADRE. Select ***Straight beam*** for this exercise (it is the default setting and so was already selected).

Click on the text box entitled "AE" to begin entering the element properties. Type in the element properties above, pressing the ***Enter*** or ***Return*** keys after each one. After the last property is entered the page for the next element will be set up and you could continue on in this manner. *But wait, there's a better way*....

First, press the ***Back*** button to go back to the first element page, then press ***F8*** to copy the displayed element properties to the special element clipboard (not the usual Windows clipboard). Now you won't have to type them again.

Press ***Next*** to return to the second element page. Select the connections and reference node (1,2,11) and then press ***F9*** to paste the element stiffness properties to this element.

Press ***Next*** to go to the next element page and repeat the procedure until all 10 elements are entered.

Press ***OK*** to advance to the Boundary Condition Form

Step 4 Defining the boundary conditions

The beam is to be restrained at node number 10. Since we have only one bound node, the complete boundary condition is described on one page. On the Bounds Form, type 10 in the node Ident text box or select the node Ident (10) from the list box.

Now for each degree of freedom (6 in all) check the box under the letter **R** to restrain that degree of freedom. All six are to be restrained.

Node 10 is the only bound node for this problem, so press ***OK*** to advance to the Loads Form.

Step 5 Selecting the problem type

Before the Loads Form appears, CADRE asks the user to decide whether the model is to be a *Static* or *Dynamic* Model. Selecting ***Static*** will set the Loads Form for entering loads. Selecting ***Dynamic*** will set up the Mass Properties Form for entering mass and Inertial data. In Exercise 4 you will construct a Dynamic model using the Mass Properties Form, but for now:

Select ***Static***.

Step 6 Entering External Nodal Forces

The beam is to be loaded in the down direction at node 0. Since we have only one loaded node, the Loads Form will have only one page. In the node Ident box type 0 or select 0 from the list box (actually zero is the default and is already displayed). Next type -1 adjacent to the Y in the load boxes, leaving all the other boxes with zero load. This provides a 1 pound down load at node 0.

Now we are finished constructing the model. Press ***OK*** to advance to the Main Menu where the model will be displayed.

Step 7   Viewing the model and input data

If everything was entered correctly, the model should be displayed on the screen as a straight line in the middle of the drawing screen and the Drawing Control Panel should be displayed above.

Check the ***Num*** box to display the nodal numbers.   The number 0 should be light blue and the number 10 should be red to indicate the load and restraint respectively.

If you received an error message or the beam appears to be plotted incorrectly, examine the data with the ***View*** option on the menu bar.   It is a good idea to check the data anyway.

Select ***View*** from the menu bar and then select ***View Configuration***.   This gives the basic model parameters; number of nodes, number of elements, etc.

Select ***View Coordinates*** to display a list of coordinates and compare them with the above table.   If they are not the same, you will have to return to the Nodes Form by way of the ***Edit*** item on the menu bar and revise them.

Select ***View Connections*** and check that each element is connected according to the picture shown above.   If they are not the same, you will have to return to the Elements Form by way of the ***Edit*** item on the menu and revise them.

Select ***View Elements*** and check the properties data.

Select ***View Bounds*** and check the boundary condition information.

Select ***View Loads*** and check that the load condition is correct.

Return to the graphic display by selecting ***View Model*** or by pressing ***Control-F***.

Congratulations, you have constructed a finite element model.

Under the ***File*** item on the menu bar save this finite element model by selecting ***Save***

File.

This model is ready to solve so go to exercise 3 for the solution exercise.

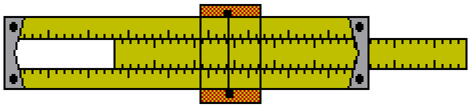
Origin node forces - Local coordinate system

Axis node forces - Local coordinate system

Origin node forces - Global coordinate system

Axis node forces - Global coordinate system

Exercise 3:   Solving a static load model



In this exercise you will solve a finite element model and examine the results.

The problem set up in exercise 2 will be solved.   If you have it displayed on the screen, continue, otherwise select ***Open File*** from the ***File*** item on the menu bar and open the file.   It is recommended that you print out this topic to make it easy to follow along as you perform the exercise.

Step 1   Derive some estimates in order to check the solution.

For this example its rather easy to get some good estimates.   The exact solution to the problem can be calculated from theoretical formulas.   These are the displacement at the load (node 0), the slope at node 0, and the moment and vertical force at the reaction (node 10).

$$y = \frac{PL^3}{3EI}$$ displacement at node 0

$$\theta = \frac{PL^2}{2EI}$$ slope at node 0

M = PL   moment at node 10

V = -P   vertical shear at node 10

y = -10.53 in

θ = +0.1317 rad

M = -120.0 in·lb

V = +1.0 lb

Also we can check displacements at the restraints to be sure they are as we specified. In this case all displacements at node 10 should be zero.

Step 2   Solving the problem.

Select the ***Solve*** item on the menu bar.

Choose ***Displacements***.

When prompted, confirm that you do want to solve and proceed.

During calculation the progress can be monitored by the messages in the status bar and by the progress gauge in the right panel of the status bar.

When the results are calculated, the ***Displacement*** scroll bar is shown. Move the scroll bar to amplify the displacements and in order to get a better visual representation of the displacement state of the model.

Step 3 Examining and interpreting the nodal displacements

First let's examine the displacements for the model.

From the menu choose ***View Displacements***.

A display like the one below should be presented where we have only shown the first and last lines (node 0 and node 10) of the displacement data.

Nodal Displacements - Page: 1

Node	X	Y	Z	Rx	Ry	Rz
000.000	0.000E+0	-1.053E+01	0.000E+0	0.000E+0	0.000E+0	1.317E-01

010.000	0.000E+00	0.000E+00	0.000E+0	0.000E+0	0.000E+0	0.000E+0

These displacements are in inches and radians and correspond to displacements in the global coordinate system. Positive displacements and rotations are according to the right hand rule.

The values for node 10 (last line of data) are all zero which confirms our expectations for the reaction displacements which were specified as fixed for this problem.

The first line of data shows the displacement at node 0.

X = 0 in

Y = -10.53 in (down displacement at node 0)

Z = 0 in

Rx =	0	rad	
Ry =	0	rad	
Rz =	0.1317	rad	(slope at node 0)

Compare these to the expected values calculated at the beginning of this exercise.

Of course, one of the most convenient ways to examine the displacements is graphically. It is clear from the graphic display that the model looks and behaves like a cantilevered beam restrained at the right end.

Step 4 Examining and interpreting the element forces

Now let's examine the internal loads.

From the menu select ***View Internal Loads***.

A table similar to the one below should be displayed, however only the first 4 lines are shown here. This section of the table corresponds to element ident number 0000.0001 as indicated in the Ident column. This is the element with node 0 at one end (origin node) and node 1 at the other (axis node). Each block of data corresponds to the loads in one element represented in two different coordinate systems (i.e. 24 loads in all)

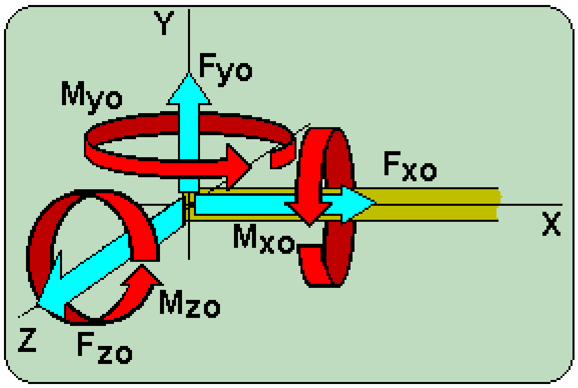
Element Nodal Forces - Global/Local Coordinates - Page: 1

Ident	Coord	Node	X	Y	Z	Mx	My	Mz
0000.0001	Global	Origin	0.000E+00	-1.000E+00	0.000E+00	0.000E+00	0.000E+00	-9.6
		Axis	0.000E+00	1.000E+00	0.000E+00	0.000E+00	0.000E+00	-1.2
	Local	Origin	0.000E+00	-1.000E+00	0.000E+00	0.000E+00	0.000E+00	-9.6
		Axis	0.000E+00	1.000E+00	0.000E+00	0.000E+00	0.000E+00	-1.2

Point to sections of this table with your mouse and click to see a description of the section.

The first two lines give the loads relative to the global coordinate system while the last two lines provide the loads relative to the beam element local coordinate system.

For structural analysis the element loads given in the local coordinate system are often the most useful. The last two lines of data give these. The bottom line lists the load acting on the element at its axis node. The line above that lists the load acting on the element at its origin node. Looking just at the local element loads listed for the node 0 (third line of data) we can compare them to the following figure. The figure shows the orientation of the positive directions for the forces and moments acting on element number 0000.0001 at its origin node.



Fxo	Axial force	0
Fyo	Vertical shear	-1 lb
Fzo	Lateral shear	0
Mxo	Torsion	0
Myo	Moment about Y	0
Mzo	Moment about Z	~ 0 in·lb

These are the loads acting on element number 0000.0001 at its origin node (left end in our example). The loads acting at the axis node (right end in our example) are listed in line 4 of the table above. The positive sense of these loads are exactly the same as shown in the figure for the origin node (left end) of the element.

Fxa	Axial force	0
Fya	Vertical shear	+1 lb
Fza	Lateral shear	0
Mxa	Torsion	0
Mya	Moment about Y	0
Mza	Moment about Z	-12.0 in·lb

Equilibrium can be verified by treating the element as a free body and summing the forces and moments acting on it from both ends.

The forces relative to the global coordinate system are the same in our example because we chose a reference node for this element that just happens to cause the local system to line up with the global system. This is not the usual case, since the beam element reference is normally selected to prescribe the best local beam element coordinate system for performing detail structural analysis on the individual element.

From the menu choose ***View Reactions***.

The results at the reaction (Node 10) are:

Y reaction = +1

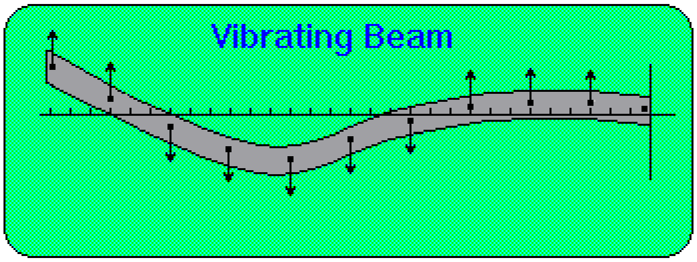
Rz reaction = -120 in lb

The reactions are always given for the boundary node in the global coordinate system.

That completes the exercises for static models. Go to exercise 4 for practice in constructing a dynamic vibration model.

Exercise 4: Building a dynamic vibration model

In this exercise you will modify the model constructed in exercise 2 in order to determine the natural vibration modes and frequencies of the cantilever beam.



It is recommended that you print out this topic to make it easy to follow along as you perform the exercise.

If you already have the model constructed in exercise 2 loaded and displayed, continue, otherwise, open it from the ***File*** item on the menu.

You will want to save the modified version in the directory with the dynamic samples, so select ***Save File As*** from the ***File*** item on the menu.

Select the directory [dynsamp] and save the model to that directory. You could also change the name of the file to avoid confusion with the *Static* problem in the other directory.

Step 1 Changing the type of operation.

If the model was constructed as a *Dynamic* model to begin with, we could skip this step. However, for this exercise the model already created in Exercise 2 will be modified to become a *Dynamic* vibration problem.

Although CADRE allows a significant amount of editing to be accomplished directly on the graphical model with the Screen Editor this exercise will introduce the basic form editing procedures.

From the menu choose ***Edit Operation***, and check the item ***Dynamic***.

The status box at the left bottom of the screen should change to display the new operation type. The ***Edit Loads*** item should now be replaced by the ***Edit Mass*** item.

To enter the degrees of freedom and the mass properties, click on ***Edit Mass***. The Mass Properties Form will appear. The main items on this form are:

A node identification entry box,

A list box of node identification numbers to select from,

A section for defining the degrees of freedom for the node,

A section for defining the mass and inertial properties, and

A section for entering the centroids of the masses.

Step 2 Derive the mass properties and distribute them among the nodes

Mass properties:

The beam in exercise 2 is 120 inches long and restrained at one end and free at the other. It is an aluminum beam (.5 x .5 x 120 inches). There is a 1 pound force acting down on one end (node 10), but we won't use that load this time. The nodes and elements have already been identified and numbered. The model is restrained at node 10.

The beam mass will be distributed among the nodes 0 through 10. This includes node 10, which is bound. There is no problem in defining mass and inertial forces for a bound node. These bound inertial forces will not have a direct influence on the problem as long as the node is restrained, but they reduce by a small amount the movable mass in the structure and they will figure significantly into the analysis of free-free modes in exercise 6.

Aluminum weighs 0.1 pounds per cubic inch. So the beam weighs 3 pounds total. There are 12 inches between nodes comprising 0.3 pounds of the beam weight. Consider that 6 inches on each side of the nodes 1 through 9 are lumped on those nodes. Then assume the left-over 6 inches (0.15 pounds) on each end are attributed to nodes 0 and 10.

To get proper units of mass [lb·in·sec^2] we divide these weights by the constant, 386 in/sec, resulting in...

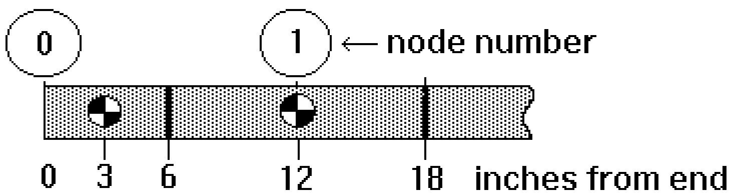
.000388601 for node 0

.000777202 for nodes 1 through 9

.000388601   for node 10

Location of mass centroids:

The centroids of the 12 inch beam sections lie directly on their respective nodes. However, the centroids of the 6 inch sections at the ends are offset from the end nodes by 3 inches (X = 3 and X = 117).



To account for this offset, the locations of the concentrated masses are entered into the center of gravity boxes on the Mass Properties Form.   The end nodes could have been redefined at X=3 and X = 117, but CADRE allows offset masses so we don't have to modify the current model data.

The degrees of freedom (DOF) must be sufficient to define an inertial force that can account for the offset masses.   When the DOF are not sufficient to take the static moment and inertia of the offset into account, the c.g. location for the mass in that dimension will be assumed to be the same as the nodal coordinate in that dimension regardless of any numbers in the boxes. (while performing this exercise you can press ***F1*** to see the help section on the Mass Properties Form and a table showing minimum degrees of freedom).

Instead of entering the differential offset from the node, the actual center of gravity location in global coordinates is entered in the Xcg text box.   CADRE calculates the offset internally.

Degrees of Freedom:

For this problem, considering only vertical vibration in the Y direction (consequently vertical inertial forces) so a 3 inch offset in the X direction would create a Z rotation moment (static moment and inertia) about the node. Therefore, we must at least specify both a vertical displacement (Y) and a rotation about Z (Zrot) for the end nodes (nodes 0 and 10) or CADRE will ignore the specified offsets.   The remaining nodes need only the Y degree of freedom.

Mass moment of Inertia:

Assume that the masses are concentrated (so small they have no inertia about their own centers of gravity). For beams with small displacements these assumptions are not bad as long as there are enough nodes. In any case, the inertia values (Iz) could actually be determined for each of the beam sections and entered into the table. In this case all the nodes would have to have at least the Y and Zrot Degrees of freedom specified. This would provide a slight improvement in accuracy but the problem would be twice as large.

Restraints:

There is still a restraint specified for node 10 (specified in exercise 2). Leave the beam restrained at this node (the beam must always be restrained at some node and stable under the particular load condition). In general, there should be as few restraints as possible without making the model unstable, and no section of the structure should be isolated by the restraints.

There are more precise ways of distributing lumped masses however, this specific distribution was chosen to demonstrate the use of offset masses in CADRE and to keep the mass symmetrically distributed when we demonstrate a free-free vibration problem in Exercise 6.

The following table shows the mass properties data to be entered:

Node	Degree of freedom X	Y	Z	Xrot	Yrot	Zrot	Mass	Inertia Iz	Location (Xcg)
0	0	x	0	0	0	x	.000388601	0	3
1	0	x	0	0	0	0	.000777202	0	at node
2	0	x	0	0	0	0	.000777202	0	at node
3	0	x	0	0	0	0	.000777202	0	at node
4	0	x	0	0	0	0	.000777202	0	at node
5	0	x	0	0	0	0	.000777202	0	at node
6	0	x	0	0	0	0	.000777202	0	at node

7	0	x	0	0	0	0	.000777202	0	at
node									
8	0	x	0	0	0	0	.000777202	0	at
node									
9	0	x	0	0	0	0	.000777202	0	at
node									
10	0	x	0	0	0	x	.000388601	0	117

Step 4 Entering mass properties

Now the data derived in step 1 will be entered.

When you first enter the Mass Properties Form from the ***Edit Mass*** item on the menu, the data from the *Static* problem are still influencing the Mass Properties Form. The displayed node is node 0 and the Y degree of freedom is already checked. This is due to the 1 pound load on node 0 in the Y direction back in exercise 2. We don't need this item so we will delete it as follows:.

> We can't delete page 1 because it's the only mass node specified, and CADRE requires at least one mass node to be specified at all times, even if it has zero mass. First use ***New Page*** button to add a new mass node page. Now use the ***Back*** button to go back to page 1. Then delete that page with the ***Shift_Delete*** Key or the ***Delete*** button. Now you should be set up with a clean slate for the vibration problem.

For the first node, 0, enter the node Ident, 0, in the Ident box. Check the degrees of freedom according to the above table (check the Y and the Zrot directions).

Xcg is 3.0 (leave Ycg as 0 and Iz as 0).

For the first displayed node (node 0), type the mass…

0.000388601

in the mass box.

Enter the rest of the mass properties as needed. Xcg is 3.0 (leave Ycg as 0 and Iz as 0 since concentrated masses are assumed for this exercise).

The button labeled ***Locate at Node***, is provided so that you can change the c.g. values to the nodes global coordinate position if you should desire to do so. The coordinates of the node are displayed in the frame to the right to allow for checking the coordinates of the node against the coordinates of the mass without replacing them.

If we were in the *entry mode*, having declared this a *Dynamic* model during the entry phase, the ***Next***/***Back*** buttons and the ***Page-Up***/***Page-Down*** keys would automatically advance to a new blank page. However, since we are modifying an existing file in *edit mode*, the scroll bars, ***Next***/***Back*** buttons, or ***Page-Up***/***Page-Down*** keys will not advance past the number of pages already set up. There is only one page right now so you will have to use the ***New Page*** button to add pages for describing each mass node.

Click on the ***New Page*** button to move to the next mass node page. The new page (page number 2) will be used to describe the next mass node (Ident number 1). Enter the mass node Ident no. (number 1) and then check just the Y degree of freedom for node 1.

Now type the mass…

.000777202

in the mass box.

Before moving to the next node with the ***New Page*** button, make it easy on yourself. Highlight the mass entry that you just made by dragging the cursor over it with the mouse. Then press ***Control-C*** to copy the number to the Windows clipboard. After this you can paste it with ***Control-V*** when you need it again. This particular feature of Windows applications can be used for convenience throughout CADRE and in most Windows applications.

Click the ***New Page*** button to go to page 3 for the next mass node (node Ident number 2 in this case).

Enter the node Ident, 2, and check the Y degree of freedom.

Click on the Mass box to set the focus there and...

Press ***Control-V*** to put the new mass number in the mass box

And press ***New Page***, and so on.

Remember when you get to mass node number 10 (Page 11) to enter both the Y and Zrot degrees of freedom, then type the smaller mass…

.000388601

in the mass box.

When you get to node 10, remember to add the center of gravity offset. The Xcg text box should currently show the global position of node 10 (120 inches). Enter the global position of the c.g. for the mass at node 10. This is the Xcg value is 117 which is 3 inches to the left of the global location of node 10.

When all the masses are entered, go back over your work with the scroll bar (on the far right of the form) or the ***Next*** and ***Back*** buttons (or Page Up and Page Down Keys) to check you work. The masses should all be the same except for 0 and 10 and the mass locations for these nodes should show Xcg to be 3 and 117 respectively.

Select ***OK*** to leave the Mass Properties Form.

You should now be back at the Main Menu with the Drawing Control Panel displayed.

Check the ***Num*** check box and examine the colors of the node numbers. If you are using the factory default settings, the restraint (number 10) should be red, the nodes 1 through 10, which have inertial forces, should be light blue. Although there is an inertial force specified for the restraint at node ten, the restraint takes precedent in coloring the node numbers.

At the Main Menu select ***Save File As*** and save the revised model to the [dynsamp] directory.

The model should now be a completed and ready for solution in exercise 5

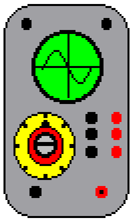
Animate button

Mode selector

Mode number or harmonic

Frequency in cycles per second

Exercise 5: Solving a vibration Model



In this exercise you will solve the cantilever beam problem set up in exercise 4 for the vibration frequencies and corresponding mode shapes.

It is recommended that you print out this topic to make it easy to follow along as you perform the exercise.

Step 1 Derive some estimates in order to check the solution.

For this example it's possible to find standard formulas that provide a check on the results. For a cantilever beam the theoretical formula (Mark's Standard Handbook for Mechanical Engineers) is:

$$f = c\sqrt{\frac{gEI}{WL^3}}$$

where:

f = frequency	Hz (cps)
g = acceleration of gravity	386 in/sec ^{2}
EI = Stiffness value (Ex. 2)	54687.5 lb/in ^{2}
W = Weight of the beam	3 lb
L = Length of the beam	120 in

c = Modal factor depending the harmonic

Mode	1	2	3	4	5
c	.56	3.57	9.82	19.2	31.8
f	1.130	7.204	19.81	38.74	64.17

Step 2 Solving the problem

Before solving of a vibration problem in CADRE it is necessary to solve and save the structural influence coefficients. This must be done for a new model and every time a structural change is made to the model. This is accomplished by:

1) Manually selecting ***Solve*** on the menu bar and then selecting ***SIC***, or
2) Checking the default in the Defaults Form to ***Do SIC before solution*** and saving the new defaults so that this will always be done automatically. (this is the factory default)

   However, if you try to solve a vibration problem and no influence coefficient file exists at all, CADRE will automatically calculate it and save it regardless of the Default Form setting.

Since this is a new file with no related structural influence coefficient file in the current directory you could just solve directly for the vibration modes and CADRE would handle everything for you. However, for practice, solve for the structural influence coefficients.

Select ***Solve*** from the menu bar and then select ***SIC***.

Respond ***OK*** to the confirmation request and the structural influence coefficient file will be calculated and saved under the same name as your model file but with the ***.sic*** extension.

Again, select ***Solve*** from the menu bar and then select ***Vibrations***.

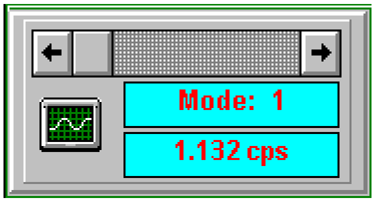
CADRE will request an input for the number of modes that you would like to determine. Since this problem has 11 degrees of freedom (13 minus the two that are bound at node 10), there are a maximum of 11 modes. However, accuracy will be not be very good for much more than the first 5 or 6 modes and the existence of the highest one is questionable. However, for this exercise, select all of them.

Choose ***OK***, and the solution will continue.

As the problem is solved, observe the messages in the status bar at the bottom of the screen and the progress with the gauge in the right panel. You could abort the solution by using the ***Cancel*** button which is only visible on the status bar during the solution.

Step 3 Examining the results

When the results are found the Mode Panel becomes visible showing the first mode and the model is displayed already displaced in the shape of the first mode.



Use the scroll bar to sweep through the modes and compare the frequencies with those provided by direct calculation above.

The results in hertz for the first 5 modes are:

Mode	Formula	CADRE
1	1.130	1.132
2	7.204	7.123
3	19.810	19.994
4	38.740	39.199
5	64.170	64.589

A more judicious selection of mass distribution could have resulted in improved results. In addition, using additional nodes and rotary inertia could provide additional improvement.

Now select vibration mode number 1 and press the ***Animate*** button on the Mode Panel in the upper left hand corner of the screen. You can adjust the amplitude of vibration with the ***Displacement*** scroll bar.

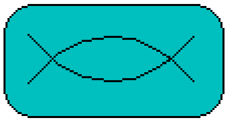
Now slowly sweep through each mode while it is vibrating. Note that CADRE could only resolve 10 modes. The additional degrees added to account for the offset masses are not really independent so only 10 actually exist for this model.

Press the ***Animate*** button again to stop the animation. Select ***Save Results*** from the ***File*** item on the menu bar and save the results. A name and directory should already be presented.

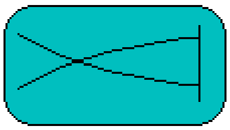
This completes exercise 5. You are now ready to move on to Exercise 6 for more advanced solutions with this same model.

Exercise 6: Advanced vibration problems

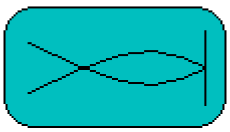
In this exercise you will solve the same beam model developed in exercise 4 but this time it will be solved for different conditions of restraint:



1. Completely free at both ends,



2. Fixed in rotation (but free to translate vertically) at one end and free at the other.



3. Hinged at one end and free at the other end,

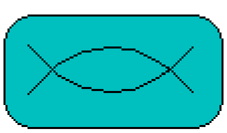
It is recommended that you print out this topic to make it easy to follow along as you perform the exercise.

All of these exercises use the special solution feature of CADRE for *Free Vibrations*.

The restraints used in exercise 5 will not be changed for these solutions. Models in CADRE must always be restrained and stable for the solution to take place. CADRE then superimposes a set of selected free body modes along with the calculated structural modes to develop a set of free vibration modes that are displayed. The finite element model file is not changed.

In this exercise, the same structural modes derived in exercise 5 will be calculated again in pass 1. Other restrained and stable solutions could also be used. The user selects the rigid body modes which CADRE will use in pass 2 to arrive at the desired free (or partially free) solutions.

Step 1 Completely free vibrations (free-free)



In the first advanced problem, the free-free vibration modes of the beam will be calculated. In this mode the free beam vibrates in a inertially balance fashion.

If the model is already opened and displayed, continue, otherwise open it from the ***File*** item on the menu bar.

You could make sure the structural influence coefficient file exists and is up to date by choosing ***Solve*** and then choosing ***SIC***. This is probably not necessary if you are following the exercises in order.

Choose ***Solve*** and then select ***Free Vibrations***.

The Rigid Body Modes Form will appear. You can press ***F1*** for help on using this form.

CADRE already suggests some selections for you. All the rigid body modes that are pertinent to your problem are already checked and all the remaining ones are disabled. This was determined from the degrees of freedom that you provided on the Mass Properties Form.

Checking a box will include that particular rigid body motion in the solution and effectively release that direction of restraint in your model. Unchecking it will exclude that rigid body motion, effectively leaving that degree of freedom restrained.

It is important to understand this clearly since it may seem opposite from the way restraints are specified on the Bounds Form.

Checking	**Releases that degree of freedom**
Unchecking	**Leaves that degree of freedom restrained**

For the first exercise, the completely free beam (free-free) will be solved. CADRE has already determined the Y (vertical translation) and the Zrot (Rotation about the Z axis) as the two rigid body modes that will completely release the system. Since they are already checked, just select ***OK***.

CADRE now asks for the modes that you wish to retain. Note that we had 11 as the maximum number of modes in exercise 5, we still have 11. The number of modes is reduced by the number of rigid body modes used and increased by the number of inertial forces released. In this case 11-2+2 = 11. This number of free modes is not always the same. If there were no mass associated with the restraint then only 9 would have been available.

Again some of the upper modes may not exist, or are so far out of range that the results are degenerate. Only about half the modes (the lower frequency ones) will be reliable.

Select all of them anyway and press ***OK***.

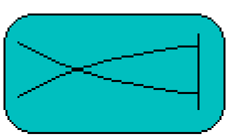
When the solution is finished, use the Mode Panel to examine the modes and their mode shapes. Animate them, and then turn on the ***Num*** check box and note that the red number 10 (restraint) now appears free and the beam is vibrating in a free-free fashion.

The results in hertz for the first 5 modes are:

Mode	Formula	CADRE
1	7.22	7.279
2	19.82	20.177
3	38.74	39.644
4	64.17	65.367
5	95.85	96.625

Note that the finite element model has not changed in this process. The boundary conditions are still the same and the model is the same as it was in exercise 5. Only the displayed results are different.

Step 2 Partially free vibration (free in translation, fixed in rotation)



Now you will solve for the beam in which the right end (node 10) is free to translate up and down but is restrained in rotation. This combination is useful if you wish to analyze only half of a symmetric structure (one side of a plane of symmetry) and solve for the symmetric vibration modes.

Select ***Solve*** from the menu bar and then select ***Free Vibrations***.

CADRE will probably ask if you want to save the results from the last problem. Say No, for now, since these results are easy to calculate any time you want.

The Rigid Body Modes Form is shown again. This time, free only the vertical translation but leave the Z rotation fixed. CADRE already has checked both Y and Zrot assuming you probably want to analyze a free-free model. Uncheck the Zrot check box so that that degree of freedom (rotation about the global Z axis) will remain fixed.

Select ***OK***.

CADRE now asks for the number of modes to be retained and informs that 11 are available (11 - 1 free body + 1 released inertial force).

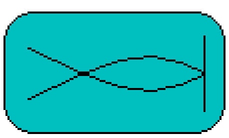
Choose ***OK*** to select all the available modes.

When the solution is finished examine them as before.

The results in hertz for the first 3 modes are:

Mode	Formula	CADRE
1	1.806	1.803
2	9.686	9.788
3	23.960	24.198

Step 3 Partially free vibration (hinged-free)



Now you will solve for the beam in which the right end (node 10) is free to rotate but is restrained in translation. This scheme is useful if you wish to analyze only half a symmetric structure (on one side of a plane of symmetry) and solve for the antisymmetric modes.

Select ***Solve*** from the menu bar and then select ***Free Vibrations***.

CADRE will ask if you want to save the results from the last problem. Select ***No***, for now, since these results are easy to calculate any time you want.

The Rigid Body Modes Form is shown again. This time, free only the Zrot rotation restraint but leave the Y translation fixed. CADRE already has checked both Y and Zrot, assuming you probably want to solve a free-free model. Uncheck the Y check box so that that degree of freedom will remain fixed.

Select ***OK***.

CADRE now asks for the number of modes to be retained and informs you that only 11 are available (11 - 1 free body + 1 inertial force). Choose ***OK*** to select all the available modes.

When the solution is finished, examine the data as before.

The results in hertz for the first 5 modes are:

Mode	Formula	CADRE
1	4.94	4.997
2	16.06	16.159
3	33.497	33.697
4	57.309	57.394
5	87.309	86.750

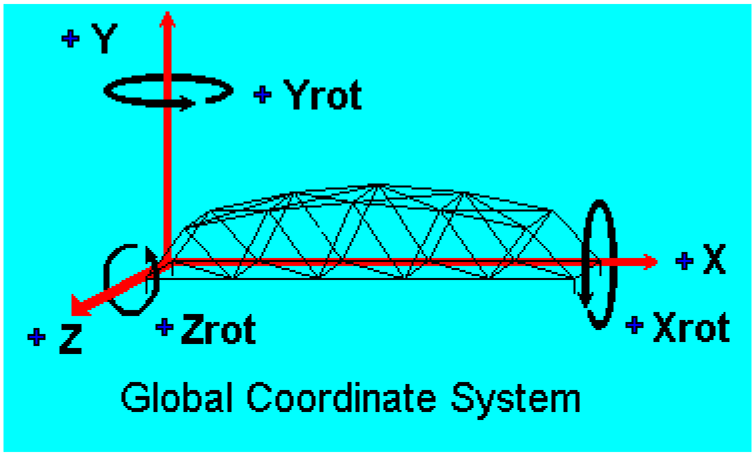
This completes exercise 6. As an additional exercise you could try the same solution but uncheck both Xrot and Y this time. Since this will leave the beam restrained, the results should be the same as calculated in exercise 5.

Getting Started with CADRE

The following exercises will help you get started using CADRE to solve *Static* and *Dynamic* problems. The exercises generally progress with the same model so that it is best to start at the beginning and work through them.

These exercises will use some of the sample files that are provided with cadre.

Exercise 1	Working with a finite element model
Exercise 2	Building a *Static* load model
Exercise 3	Solving a *Static* load model
Exercise 4	Building a *Dynamic* vibration model
Exercise 5	Solving a vibration model
Exercise 6	Advanced vibrations problems


+ Y
+ Yrot
+ X
+ Z
+ Zrot
+ Xrot
Global Coordinate System

External Loads

External loads are the forces and moments that act on the structure.

In CADRE external loads act only on the structural nodes of a model. The units of specified loads must be consistent with the other dimensions of the problem. In the English system forces will usually be expressed in pounds and moments in inch-pounds, however other units are also possible. The user specifies non-zero known external loads at each loaded node in the global coordinate system. Nodal loads of zero do not need to be specified, they are assumed to act on each non-restrained node unless another value is specified.

Restraints

Restraints are the known displacements that restrain a structure.

Although not necessarily so, these known displacements will often be zero, indicating that that point of the structure does not move. For example in analyzing the Eiffel Tower the points where the tower is attached to the ground would be restrained (displacement set to zero). Sufficient restraint must be applied to a model to prevent any rigid body movement of the structure, either in translation or rotation. Failure to sufficiently restrain a model will give unsatisfactory results and may result in an error when a solution is attempted.

In CADRE restraints can only be applied at structural nodes. Each node has six degrees of freedom, three translation (X, Y, Z) and three rotation (Xrot, Yrot, and Zrot). Restraint may be specified for any degree of freedom of any structural node.

Displacement

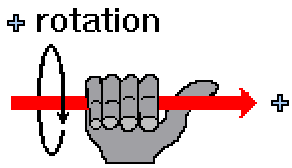
A displacement is the change in the position or orientation of a point on a structure subsequent to the application of a loading condition.

In general, both linear and angular displacements are possible. Thus the end of a horizontal cantilevered beam will be displaced downward in position and angle by a certain amount if a weight is applied to its end. The finite element method assumes that displacements under load are small and proportional to the applied load (therefore the constitutive equations are linear). Large displacements may be analyzed by performing piecewise linear analyses of the structure.

The solution of a finite element problem consists of computing the unknown displacements of the nodes of the model under load from the constitutive equations.

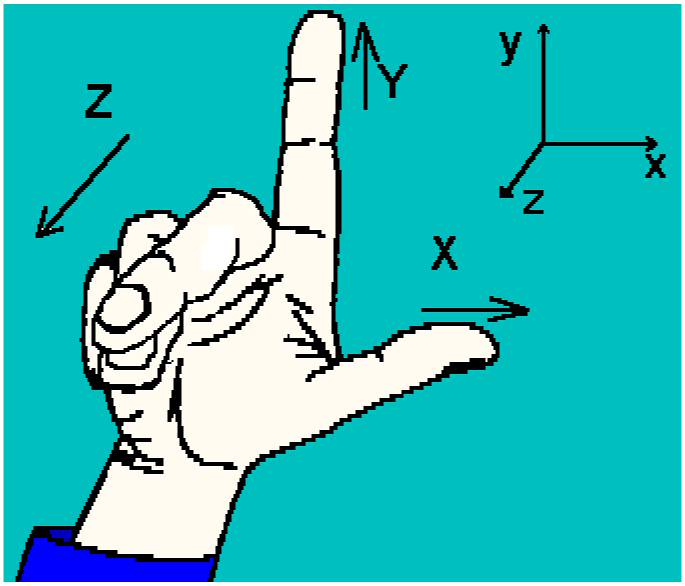
Known displacements are called restraints.

CADRE incorporates six degrees of freedom per node, three linear (X,Y, Z) and three angular (Xrot, Yrot, and Zrot). Therefore, in general six displacements of a node are possible, one for each degree of freedom. The units of computed displacements are consistent with the units used to specify the coordinates of the nodes of the model. So if the coordinates of the nodes are specified in feet, the computed displacements should be interpreted in feet. All angular displacements are in radians. The computed displacements are given in the model global coordinate system.



Right hand rule

Right Handed Coordinate System



Global Coordinate System

The Cartesian coordinate system (X, Y, Z) used to locate all the nodes of a model.   It is also the global coordinate system used for specifying external loads:

The coordinate system is a right handed coordinates system.

Fx, Fy, and Fz (the external forces) are defined positive in the positive X, Y, and Z directions and

Mx, My, and Mz, (the external moments) are defined positive according to the right hand rule for rotations in which the thumb of the right hand points in the positive X, Y, and Z directions respectively.

Internal Loads

The internal loads are the forces and moments that act within the structure.

In CADRE internal loads act on a finite element of a model at its corresponding structural nodes. The units of computed internal loads are consistent with the other dimensions used to specify the problem. In the English system forces will usually be expressed in pounds and moments in in-pounds, however other units are also possible. The computed internal loads are given in both the beam local coordinate system, and in the global coordinate system. Unknown external loads (i.e. reactions) at a structural node with specified restraints may be computed to be the the sum of the internal element loads acting on the individual elements attached to that point.

Straight Beam

The straight beam is an idealized one dimensional element with constant cross section.

It is capable of carrying axial force, and transverse force in two directions, transverse bending in two directions and torsion.   The Areal Properties (AE, EIy, EIz, & JG) are specified by the finite element analyst with reference to the beam local coordinate system.

In CADRE the finite element analyst specifies the following data to define a straight beam:

- Beam type   (=S)
- Origin Node
- Axis Node
- Reference Node
- AE
- EIy
- EIz
- JG

Tapered Beam

The tapered beam is an idealized one dimensional element with cross sectional area properties that vary linearly from one end to the other.

It is capable of carrying axial force, and transverse force in two directions, transverse bending in two directions and torsion. The Areal Properties (AE, EIy, EIz, & JG) are specified by the finite element analyst at each end of the beam with reference to the beam local coordinate system.

In CADRE the finite element analyst specifies the following data to define a straight beam:

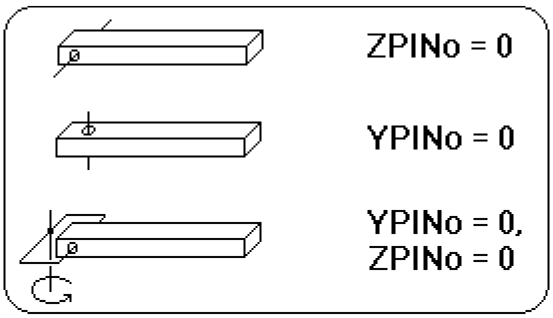
- Beam type (=T)
- Origin Node
- Axis Node
- Reference node
- (AE)o
- (EIy)o
- (EIz)o
- (JG)o
- (AE)a
- (EIy)a
- (EIz)a
- (JG)a

Note: A taper ratio of zero is not allowed in CADRE, so there can be no zero valued properties.

Pinned Beam

The pinned beam is an idealized one dimensional element with constant cross section.

It is capable of carrying axial force, and transverse force in two directions, transverse bending in two directions and torsion.   The Areal Properties (AE, EIy, EIz, & JG)   and pin configuration at the ends of the beam are specified by the finite element analyst with reference to the beam local coordinate system.   If a free pin is specified at a beam node then the beam cannot react moment at that point about the axis of the specified pin.



In CADRE the finite element analyst specifies the following data to define a pinned beam:

- Beam type   (=P)
- Origin Node
- Axis Node
- Reference Node
- AE
- EIy
- EIz
- JG
- YPINo
- ZPINo
- YPINa
- ZPINa

CADRE accepts 0 to define a free pin and 1 to define a rigid pin (i.e. no pin).   Any fraction (f) between 0 and 1 can be used to define a degree of end fixity.

See Elements Form

Finite element analyst - A finite element analyst is any user of a finite element program.

Element Properties - All the properties that describe an element including its type, nodes, and its stiffness properties.

Stiffness Properties - AE, EIy, EIz, JG, for the beam cross section.

Beam Elements

A beam element is one modeled after the typical engineering representation of a slender beam subject to bending, shear, and torsion (See for example Roark's Formulas for Stress and Strain, chapter 7).

The beam element is one dimensional (length only) and subject to load or restraint only at its end points (nodes).   The internal loads computed for the beam element consist of the forces and moments applied to its end points.

In CADRE a beam element is characterized by:

the nodes that define its end points and orientation,

the stiffness properties of the beam cross section, and

the pin conditions (only if the beam is a pinned beam)

Structural node

In CADRE, by convention, all points used to define the beam elements are called *nodes,* including a third point for each beam element that is used to define its local coordinate system. Only points that define the ends of the beams are included in the displacement solution of a model. These are called the *structural nodes*.

By convention the structural node that defines one end of the beam is called the origin node and the structural node that defines the other end is known as the axis node. These are distinguished from nodes that are used only for defining the beam local coordinate system, which are known as reference nodes. Structural nodes are those that are actually part of the model and for which displacements are computed.

A node that is a structural node for one element may act as a reference node for a second element. (See beam local coordinate system figure.)

Finite element

A finite element is a mathematical representation of a piece or part of a structure.

It is characterized by a set of nodes that define its boundaries and geometry and a set of element properties that define behavior within the element.

CADRE implements three types of Beam elements that can be used to model frame type structures:

Straight Beam

Tapered Beam

Pinned Beam

Reference node

In CADRE, by convention, all points used to define the beam elements are called *nodes,* including a third point for each beam element that is used to help define the beam local coordinate system. This third point is called the *reference node* for the beam element.

This reference node, in combination with the two structural nodes for the beam, defines the beam local coordinate system. By convention these three nodes define the X-Y plane of the beam local coordinate system with the Y-axis positive on the side to the X-axis that the reference node lies. A node that is a reference node for one element may act as a structural node for a second element. Displacements are not computed for reference nodes.

Nodes

Nodes comprise the boundary points and intersection points of elements.

They are characterized by their nodal identification number and by their Cartesian space coordinates(X, Y, Z). In general, each node has six degrees of freedom, three in translation and three in rotation. The finite element analyst specifies the three space coordinates of each node when defining a model. After solution of a finite element problem the analyst obtains the set of displacements for each node. Internal loads in the beam elements are computed from the displacement of their end nodes.

In CADRE each node is characterized by its nodal identification number and its three Cartesian space coordinates. The set of nodes necessary for a structural model is determined by the beam elements used. Each beam element in CADRE requires three nodes to completely define its orientation, two structural nodes, to define its end points and one reference node to define the ***reference plane*** for its bending properties.

Factoring and Converting (Registered version only)

The Factoring and Converting utility is available with the ***Conversion*** item under the ***Convert*** item on the main menu bar which is provided with the registered version of CADRE. This general conversion utility allows selected portions of the input data file to be multiplied by factors supplied by the user.

Some uses are:

- Flexibility with input data. For example, when a new data file is created, just the inertia properties (excluding moduli) can be entered as element properties, then the element properties can be multiplied by the moduli (E and G) with the factoring utility provided the moduli are the same for the whole model.
- Increasing or decreasing load levels by a specified factor.
- Converting coordinates from another file to the correct units of the current file.
- Converting to other systems of measurement.

Converting between standard systems of measurement:

The Factoring and Converting Utility provides preset groups of multiplying factors that can be used to convert between the following measurement systems:

- IPS Inch Pound Second
- FPS Foot Pound Second
- CGS Centimeter Gram Second
- MKS Meter Kilogram Second

CADRE is written as a unit independent tool. Therefore, any system can be used as long as the units are consistent.

CADRE can not identify the measurement system established for the current data file. However, CADRE will perform a correct conversion from one consistent system to another consistent system provided the user identifies both the correct "***From***" and the correct "***To***" systems.

Only the loaded file in memory is converted and the newly converted file is not saved until the users requests that it be saved. Calculated results are not converted and any previously calculated results will be inconsistent with the newly converted file.

Units

Units are the names of the dimensions in the system of measurement used to specify quantities. Consistent use of units is essential in all engineering computations, including finite element analyses. The finite element analyst must be consistent in the use of units in specifying a finite element problem so that no conversion factor is necessary to interpret the results. This means that if one length is specified in inches, all other length dimensions must also be in inches. Many consistent sets of units are possible, but the following are used most often:

Extra conversion utilities provided with the registered version can be used to effectively transform the model to a variety of common systems.

Quantity	English System	Metric System
Length	inch	meter
Coordinate	inch	meter
Displacement	inch	meter
Angle	radian	radian
Force	pound	newton
Moment	inch·pound	newton·meter
Stress	pound/inch ^{2}	pascal (newton/meter)
Area (A)	inch ^{2}	meter
Moment of Inertia (I)	inch ^{4}	meter
Young's modulus (E)	pound/inch ^{2}	newton/meter
Shear modulus (G)	pound/inch ^{2}	newton/meter
Polar moment of inertia (J)	inch ^{4}	meter
Mass	pound·sec ^{2}/in	kilograms
Mass Moment of Inertia (I)	pound·sec·in	kilogram·meter ^{2}

Finite Element Analysis Overview

Finite element analysis is used widely in structural engineering to analyze complex structures by computer to obtain deformation, stress, strain and vibration information.

The finite element analyst provides numerical data to define:

> the finite elements that describe the geometry and characteristics of a structure, and

> the known external loads and restraints that describe a loading condition which acts on the structure.

The computer solves for:

> the displacement field of the structure under load and

> the internal loads created in the finite elements.

CADRE implements the finite element method in a manner adapted for analyzing the statics and dynamics of frame type structures. CADRE offers the use of:

> straight beam,
> tapered beam, and
> pinned beam

elements formulated using standard engineering theory for slender beams (see for example Roark's Formulas for Stress and Strain, Chapter 7). This allows the user an efficient, straight forward way of representing a wide variety of frame structures of interest to structural analysts and dynamicists.

Finite Element Analysis: Contents

This section contains a brief overview of the finite element analysis methods.

Overview of finite element analysis

Building the Model

Structural influence coefficients

Vibration analyses

Areal Properties - The cross section properties of a beam including area, A, moment of inertia of area in bending, Iy or Iz, polar moment of inertia of area, J.   Also refers to the products of these areal terms with the elastic moduli (i.e. AE, EIy, EIz, JG)

Idealize the Structure

Idealize the structure as a set of interconnected beam elements

Many engineering structures are comprised of a set of beams into what is known as a frame structure. A well known example is the Eiffel Tower in Paris. The behavior of these types of structures may be approximated using a network of beam type finite elements. Finite elements implemented in CADRE are:

- straight beam,
- tapered beam, and
- pinned beam.

These elements are formulated using engineering theory for slender beams (see for example Roark's Formulas for Stress and Strain, Chapter 7). A beam, or more precisely, its neutral axis, is represented by a line between two nodes of the model. The finite element analyst provides the identification number and the Cartesian space coordinates(X, Y, Z) for each node. In addition, for each beam element the analyst provides:

- The beam type (selected from a list),
- The identification numbers of its defining nodes,
- The areal properties of the beam cross section,
- The configuration of any pins (for pinned beams only).

The skill, knowledge, and judgment of the finite element analyst will determine the accuracy of a finite element model. This develops through extensive experience and study. This help file can only provide the user with a rudimentary understanding of the subject. Nevertheless the authors believe that even the novice finite element user may gain many valid insights into structural behavior rather easily using CADRE.

The For Further Reading section will be of use to the user interested in mastering the finite element method. Some general guidelines to follow for successful model building using beam elements:

1. Be sure there are sufficient restraints to prevent rigid body motions of the model.

2. Use care in connecting very flexible members to very rigid members. This can result in ill-conditioning the stiffness matrix.

3. Number the nodes to maximize the benefit of a small bandwidth.

Assemble the Node Point Data

ASSEMBLE THE NODAL POINT DATA FOR THE MODEL

The identification number and the X, Y, and Z Cartesian coordinates of each node are required input. The set of nodes necessary for a structural model is determined by the beam elements used. Each beam element in CADRE requires three nodes to completely define its orientation, two structural nodes, to define its end points and one reference node to define the ***reference plane*** for its bending properties.

Nodal identification numbers may be any decimal number however, integer numbers from 0 to 9999 are recommended to avoid confusion since the element identification is formed from the identification number of the element's end nodes. However, decimal numbers can be used temporarily to insert nodes and maintain the proper ordering of the nodes. Nodes can always be re-identify the nodes later to restore integer values.

The value of a nodal identification number determines the ordering of its corresponding degrees of freedom in the structural stiffness matrix. Judicious choice of nodal identification numbering can reduce storage and computation resources of the computer by reducing the bandwidth of the structural stiffness matrix. CADRE has a special re_ordering routine that can be used to minimize the bandwidth. For a more thorough discussion of bandwidth refer to the list of references.

Once the node point data have been assembled it may be entered into CADRE using the INPUT forms.

Bandwidth

CADRE saves the structural stiffness matrix in "banded" form.

The semibandwidth of the stiffness matrix is the greatest distance (columnwise) of any non-zero term of the matrix from the diagonal [max(J-I)+1 for all x(I,J) where J>I]. The bandwidth of a given finite element model is determined by the ordering of the nodes determined by the nodal identification convention of CADRE. The nodal number, or rank, is determined by the value of its nodal identification number, the larger the number, the greater its rank. The semibandwidth of a finite element model stiffness matrix is determined as follows:

B=MAX[(ABS((origin node rank) -(axis node rank))+1)*6] for all elements

Therefore, the general rule of thumb is to keep the nodal identification numbers of nodes common to any one element as close as possible (i.e.: close, so as to allow the fewest number of intervening nodes in the sequence.).

CADRE has a special re-ordering routine that can be used to minimize the bandwidth.

Assemble The Properties Data

Assemble the Properties data for the individual beam elements

CADRE offers three beam types:

straight beam,

tapered beam, and

pinned beam

These elements are formulated using engineering theory for slender beams (see for example Roark's Formulas for Stress and Strain, Chapter 7). The data required for a beam element depend on its type, but generally fall in the following categories:

Beam type, (selected from a list)

The identification numbers of its defining nodes (three decimal numbers),

The area and inertial properties of the beam cross section (4 or 8 floating point numbers),

The configuration of any pins (for pinned beams only - 4 integers).

 **Specify Nodal Restraint Conditions**

Specify a nodal restraint condition for the model

Restraints are the known displacements that restrain a structure from rigid body motion. Restraints may only be applied at structural node points. These restraints should be in agreement with the kinematics of the structural problem. In CADRE the restraints are specified by:

The node identification number of the restrained node, and

the restrained degrees of freedom (6 values),

Free indicates no restraint,

A number indicates specified displacement,

Fixed or *Restrained* indicates exactly zero displacement.

In CADRE the restraint data is entered and edited by means of a restraint data entry form which automatically sets up the restraint values based on menu choices.

Specify A Loading Condition

Specify a loading condition for the model

The loading condition for the model consists of the set of external loads acting on the model. By finite element convention the external loads act only on the structural nodes of the model. Loads that are applied to elements should be distributed to the structural nodes in a rational manner. The external load set is the set of non-zero external forces and moments acting on a structure. Only nodes with non-zero external forces or moments need to be identified in the external load set. In CADRE the loads are specified by:

> The node identification number of the point of application, and

> The values of the external load for each degree of freedom (6 decimal values)

The units for the values of loads must be consistent with the system of units chosen for the model.

Once the loading condition for the model is determined, it may be entered into CADRE using the Loads Form.

For *Dynamic* problems, instead of entering external loads, the mass properties and degrees of freedom for the selected nodes are entered. This is analogous to the *Static* case in that these mass nodes provide the inertial loads that act on the dynamic system in vibration.

Node Identification Number

The node identification number is a number that the user assigns to a node. It is used as the name for the node in subsequent reference. Although any decimal value can be used, integers from 0 to 9999 are recommended to avoid confusion because the element identification number is automatically formed from the element's end node numbers. Decimal values can be used judiciously to insert a node between two other nodes to maintain a desired ranking as described below. The nodes can always be re-identified to integer values after the data are entered are sorted.

The node numbers are used to determine the node's rank in the stiffness matrix. For example, the following list of three numbers could be the identification numbers for three nodes:

5

5.5

6

For these numbers their rank is as follows:

5 is rank 1

5.5 is rank 2

6 is rank 3.

Node stiffness data are incorporated into the stiffness matrix in locations corresponding to the node's rank. By judiciously choosing the node identification numbers the user may minimize the semibandwidth of the stiffness matrix, thereby reducing data storage requirements and computation time. The general rule of thumb is to keep the nodal identification numbers of nodes common to any one element as close as possible (i.e.: close, so as to allow the fewest number of intervening nodes in the sequence.)

Solving The Model

Once the necessary data have been entered into CADRE, a solution may be carried out. The basic unknowns in the finite element formulation are the displacements of the nodes from their original position. The solution technique employed in CADRE is Gaussian elimination operating on a symmetric, banded matrix. The solution time is a function of the number of structural nodes in the model, and the bandedness of the stiffness matrix.

Once the displacements have been calculated, the loads in individual members are calculated from the known displacements at their end nodes.

The results of the solution may be viewed numerically from the ***View*** item on the menu, or graphically, in the case of displacements.

Interpreting the results

Once a solution has been executed in CADRE the analyst may view the results using the ***View*** menu.

To see the resulting displacements select ***View*** then ***Displacements***. The table of displacements represent the motion of the indicated node subsequent to loading. The units of the displacements will be the same as the units used to specify the coordinates of the node points. All rotational displacements are in radians. The user may also view the displacement under load graphically.

To see the resulting internal loads in the elements of the model select ***View loads***. The table presented for the internal loads lists four lines of data for each element. These lines of data represent, respectively:

1. The load acting on the element at the origin node in the global coordinate system.
2. The load acting on the element at the axis node in the global coordinate system.
3. The load acting on the element at the origin node in the beam local coordinate system.
4. The load acting on the element at the axis node in the beam local coordinate system.

The units of the internal loads will be consistent with the system of units chosen for the model.

The reaction loads at restrained nodes may be viewed with the ***Reactions*** item under the ***View*** item on the Main Menu bar. Reactions loads are provided for the bound nodes and are always shown in the global coordinate system.

FURTHER READING

1. O. C. Zienkiewicz, The Finite Element Method in Structural Continuum Mechanics, 1967, McGraw-Hill.

2. H. C. Martin, Introduction to Matrix Methods of Structural Analysis, 1966, McGraw-Hill.

3. R. D. Cook, Concepts and Applications of Finite Element Analysis, 1974, John Wiley & Sons.

4. W. C. Young, Roark's Formulas for Stress and Strain, Sixth Edition, 1989, McGraw-Hill.

Analysis: Building the Model

On a fundamental basis, building a finite element model consists of assembling the equations that represent the behavior of a structure under load.

Conceptually, however, building a model proceeds in an intuitive manner, much like building the structure itself. The analyst provides the characteristics of individual beams, specifies how they are connected, and relies on the computer to assemble the equations according to the conventions of the finite element program.

The accuracy of a model will depend on the skill and knowledge of the analyst, but the procedure involved in building a model is the same for all. The basic steps in building a finite element frame model are:

- Idealize the structure.
- Assemble the node point data.
- Assemble the element properties.
- Specify a nodal restraint condition.
- Specify a loading condition.
- Solve the model.
- Interpret the results.

Analysis: Structural Influence Coefficients

CADRE uses a structural influence coefficient matrix for dynamic analyses.

A structural influence coefficient (SIC) is defined as the displacement at a selected node due to a unit load at another (or the same) node. The influence coefficient is usually represented by a matrix element. For example:

a(i,j), where the units are in terms of displacement per unit load (e.g. lb/in)

In the simplest terms, this value represents the displacement in a particular direction at point i due to a unit load applied in a particular direction at point j.

The displacement directions can be defined in terms of angular rotation or translation and the unit load can be defined in terms of a unit force or a unit moment.

The directions of movement prescribed for a node are its degrees of freedom. An influence coefficient can be determined for each degree of freedom in the model.

A matrix of structural influence coefficients can be determined by solving for the displacements at a set of nodes with a unit load applied at one degree of freedom for one of them. This can be repeated until the load has been located at each degree of freedom for each node.

A full matrix of influence coefficients (for all degrees of freedom in the model) is actually the inverse of the finite element stiffness matrix which is developed for a finite element solution. However, for dynamic analyses, it is convenient to determine a reduced set of influence coefficients.

Once the matrix of influence coefficients is determined, inertial properties can be prescribed for the selected degrees of freedom and the vibration modes and frequencies can be determined.

Changing the structural model will usually change the structural influence coefficients, so after editing any structural parameters such as nodal coordinates, elements, boundary conditions, or the selection of the unit load points, the structural influence coefficient matrix must be recalculated.

Changing the mass (or inertia) data will not change the structural influence coefficient matrix.

Eigenvalues and Eigenvectors

Many dynamics problems can be resolved to a simple matrix equation which is in the form of the well known eigenvalue problem.

The standard form for the eigenvalue equation is:

$$[D-\lambda I]\{y\} = \{0\}$$

It can be shown that the determinant, $|D-\lambda I|$, must be zero.

The values of λ that make this true are called the eigenvalues. Once determined, each of these values can be substituted back into the eigenvalue equation to determine the corresponding value of {y}. The column matrix {y} associated with each eigenvalue l is called an eigenvector.

For a dynamics problem, there are as many eigenvalues and eigenvectors as there are <u>degrees of freedom</u> represented in the model. Each eigenvalue is related to a natural frequency and each eigenvector is related to the mode shape of that frequency.

Algebraic solution of this equation is impractical for a large model, but many matrix iteration methods have been developed to extract the eigenvalues and eigenvectors from a large matrix. For a complete discussion of extraction methods refer to <u>Wilkinson</u>.

The methods used in CADRE are:

Power iteration method

This method of iteration results in convergence to the dominant (largest) eigenvalue and eigenvector simultaneously. These correspond to the lowest natural frequency and vibration mode shape. Then a process of deflation is used in which the dominant mode is suppressed in the dynamic matrix without changing any of the other eigenvalues and eigenvectors.

Then the iteration process is performed again for the next eigenvalue/eigenvector pair and so on until all are found. Often, convergence will fail at some point and higher modes cannot be found. Fortunately the modes of interest are usually the lower frequency modes.

For reference, the power method provided by CADRE uses the Hotelling method of bi-orthogonality between the left and right handed eigenvectors as a means of deflation. This method is discussed in detail in Wilkinson.

QR algorithm

The CADRE default algorithm. It is considered the most dependable for most problems.

This is an iteration scheme in which all eigenvalues are determined at the same time but not the eigenvectors. Once all the eigenvalues have been found, then each eigenvalue can be used in a variety of ways to find the corresponding eigenvector.

The method provided in CADRE for determining the corresponding eigenvectors is the Inverse Iteration method.

Analysis: Vibration Analyses

Vibration analysis of a finite element model is conducted in CADRE by determining a set of structural influence coefficients for the structural nodes (or a subset of these nodes) and applying lumped masses at (or relative to) these nodes. If the influence coefficients are calculated for rotary as well as translational degrees of freedom, then rotary inertial properties can also be included.

Inertial forces: The inertial forces for a structure can be described according to Newton's second law of motion;

Force = (Mass) x (Acceleration)

or in matrix form;

{F} = [M]{a}

where {F} is the column of forces;
[M] is the square matrix of inertial properties at the nodes;
{a} is the column matrix of accelerations of the nodes.

For simple harmonic motion the acceleration is

$$\{a\} = -\omega^2 \{y\}$$

where ω^2 is the square of the circular frequency,
and {y} is the column matrix of the displacements of the nodes.

Displacements: The displacement of the structure under any applied loading {P} can be defined by:

{y} = [A]{P}

where [A] is the matrix of structural influence coefficients (SIC)

{P} is the column matrix of applied loads

Since the structural influence coefficient matrix is the inverse of the stiffness matrix, the above equation is just an inverted matrix representation of Hook's law which describes the proportionality between load and displacement.

In the case of a freely vibrating structure the external applied loads {P} can be considered to be the inertial forces {F} so that,

$$\{y\} = -\omega^2 [A][M]\{y\}$$

The mass matrix [M] must match the degrees of freedom selected for the influence coefficients [A]. The product of [A][M] is called the dynamic matrix [D].

$$\{y\} = -\omega^2 [D]\{y\}$$

or,

$$[D-(1/\omega^2)I]\{y\} = \{0\}$$

The equation above is solved for the vibration modes and frequencies. The solution of this equation has a well known eigenvalue form:

$$| D - (1/ \omega^2) I | = 0$$

so that after solving this equation, for the Eigenvalues and Eigenvectors, the frequencies can be related to the eigenvalues and the mode shapes can be related to the eigenvectors.

Free Vibration Modes: All vibration analyses conducted by CADRE are free in the sense that they are unforced and undamped vibration problems. To distinguish between a restrained (but unforced) vibrating structure and an unrestrained free vibrating structure CADRE sometimes uses the term free-free. However, any reference to free vibration in CADRE should be considered to be unrestrained vibration of an unforced and undamped system.

CADRE calculates free-free vibration modes by first calculating a set of modes with the restrained model (the model **must** always be restrained and stable), and then adding a set of user selected rigid body modes.

CADRE then finds the proportions of each of these modes (both restrained and rigid body) which can be superimposed to provide a set of freely vibrating inertially balanced modes.

Once the proportions are found then the actual modes are calculated and displayed.

The actual model boundary conditions are not changed by this process.

 **Overview of CADRE:**

CADRE is a finite element analysis program designed to solve static and dynamic structural frame problems. The structural model is constructed using a series of data entry forms which involve:

- defining the nodal coordinates
- defining the element properties and connections
- defining the boundary conditions
- defining the loading conditions, or
- defining mass and inertial data for dynamic models

There are two types of operations available in CADRE:

Static	Static analysis for deflections and internal loads.
Dynamic	Vibration modes and frequencies.

All new models are created the same until after the boundary conditions are entered. CADRE then asks the user to specify ***Static*** or ***Dynamic***. The next form in the entry procedure will be the Loads Form or the Mass Properties Form, depending on the users response. After entering the data CADRE presents the Main Menu and a display of the model is presented. The model can be rotated, moved, magnified, and if results are calculated, the structural deformation and internal loads can be displayed.

For the *Static* model CADRE provides a menu option to *Solve* for internal loads and displacements. These results can be reviewed, printed, and saved for future examination.

A *Dynamic* model is developed in the same way as a *Static* model except, instead of entering loading conditions on the Loads Form, mass points are identified on the Mass Properties Form. The degrees of freedom are identified by checking the nodal degrees to be included in the dynamic analysis. These are the degrees of freedom for which lumped mass and inertia properties will be entered on this form.

A *Static* model can be changed to a *Dynamic* model and vise versa, by use of the ***Edit Operation*** item on the menu bar.

The vibration analysis can be conducted for restrained models and for free-free models. Once the results are obtained the modes of vibration are displayed and can be animated on the graphics screen.

CADRE Keyboard Hotkeys

Cadre provides the following keyboard hot keys

On the Main Menu

Key	Operation
F1	Help contents
F2	Sort Nodes
F3	Sort elements
F5	Save a finite element model file under a new name
F6	Toggle the option: Reposition on click
F7	Exit CADRE
Control_N	Create a new finite element model file
Control_O	Open a finite element model file
Control_S	Save a finite element model file
Control_V	Save calculated results
Control_R	Reload calculated results
Control_I	Print out the finite element model file (input file)
Control_P	Print out the calculated results (output file)
Control_M	Print out the model graphic
Control_D	Print out the displayed information on the view screen
Control_F	View the model on the view screen
Control_T	Toggle between normal and full screen modes
Control_A	Animate the model with a vibration mode
Control_G	Merge a file
Control_U	Undo a merging operation

On the data entry forms:

Key	Operation

F1	Help on the specific input form
Escape	Edit Mode: Cancel & return to main menu
	Entry Mode: Return to previous entry form
Page Down	Go to the next node or element
Page Up	Go to the prior node or element
Shift-Insert	Insert a node or element before the displayed one
Shift-Delete	Delete the displayed node or element
Home	Go to the first node or element
End	Go to the last node or element
Enter	Next data box or if at the last box, go to the next node or element

USING CADRE: Contents

Help contains the following topics on using CADRE.

- Overview
- Keyboard
- Setting defaults
- Main menu
- Entering and editing the data
- Setting the operation
- Sorting
- Solving
- Managing files
- Viewing the data
- Viewing the model
- Printing
- Error Messages
- Merging models

CADRE: Defaults

The options and defaults for CADRE can be selected from the ***Options*** item on the menu bar. This submenu has two items, ***Sound*** and ***Defaults***. The ***Sound*** item is used to activate or deactivate the CADRE sounds.

The ***Defaults*** item provides access to the Defaults Form. This form is accessed from the Main Menu with the item labeled ***Options***.

Controls: There are two main control buttons for this form:

OK	Activates the current selected defaults and exits the form
Cancel	Exits the form without setting the defaults

Menu bar: There are four main items on the menu bar

Save	Saves the current choices as the user defaults for future sessions
Restore	
--**Factory**	Restores factory settings (immediately activated)
--**User**	Restores the last-saved user defaults (immediately activated)
Fonts	
--**Form**	Changes the label fonts on all CADRE screens.
--**Printer**	Changes the fonts used for printer output.
Help	Displays this help topic

Saving user defaults: The default items can be saved with the ***Save*** item on the menu bar. Then, the new settings will be stored in the CADRE.INI file and will be set on startup in any future sessions.

Temporary changes: If the ***OK*** button is selected without first choosing the ***Save*** item on the menu bar, the new settings will be in effect only for the current session.

Factory defaults: If the Restore Factory item is selected, the original factory settings (which cannot be modified) will be restored. These also will be in effect only during the current session, unless the ***Save*** item is used to save them to the CADRE.INI file.

The default screen has four sections:

General Colors
Drawing Screen Options
Vibration Options
Miscellaneous Options

Defaults: General Colors

The General Colors section of the Defaults Form contains ten buttons.

List box color:	Sets color of all listboxes in CADRE.
Text box color:	Sets color of all textboxes in CADRE.
Scrollbar color:	Sets the scrollbar color.
Form caption bar color:	Sets caption bar color on all forms.
Form caption text color:	Sets caption text color on all forms.
Menu bar color:	Sets color of all menu bars.
Menu text color:	Sets text color of all menu bars.
Disabled text color:	Sets the color of text of disabled items.
Startup screen color:	Sets the color of the startup screen.
Main menu trim color:	Sets the color of the background panel of the controls panel on the main menu.

Defaults: Drawing Screen

There are five items that can be changed in the default options.

Background Color The color of the drawing screen can be selected.

Foreground Color The color of the object drawn on the screen can be selected.

Bounds Color The color of the numbers of the bound nodes on the Model.

Loads Color The color of the numbers of the loaded nodes or the lumped mass nodes on the model.

Reposition of Click With this option checked, any part of the model (or screen) that is clicked with the right mouse button will be moved to the center of the screen. A small square will be displayed surrounding the screen center when this option is active. Use ***F6*** to toggle this option from the Main Menu.

Defaults: Vibration Analyses

This section of the Defaults Form allows some defaults to be set which effect the way the vibration calculation are performed:

Do SIC before Solution: Before solving any vibration problem the influence coefficients for the structural model must have been calculated and saved in a file. This can be accomplished manually from the Main Menu under the ***Solve*** item. However, you can set the default so that this occurs automatically before conducting any vibration analysis.

Calculating the structural influence coefficient solutions can be time consuming and repeating it is necessary only if structural changes have been made to the model (restraints, stiffness, degrees of freedom, or geometry). Changes such as inertia and mass properties do not require a new structural influence coefficient solution.

Therefore, it may be a disadvantage to automatically recalculate the structural influence coefficient.

On the other hand, changing the structural model and forgetting to solve for a new structural influence coefficient file can result in erroneous vibration results (sometimes without any indication of this fact).

Animate frequency: Once the vibration modes are calculated, they can be animated. CADRE allows you to set the speed at which items will be animated on the screen. The number entered is actually the number of redraws per cycle so *the smaller the number, the faster the animation*.

Maximum iterations: Some vibration problems may be very slow to converge. This item sets the maximum iterations after which CADRE will stop the iteration and accept the current accuracy. This item only effects the performance of the Power Algorithm

Frequency Format: This provides a selection for the display of the frequency in the frequency text box. The formats provided are:

0.0	One decimal place, Fixed
0.00	Two decimal places, Fixed
0.000	Three decimal places, Fixed
0.00E+00	Three significant figures, Scientific

0.000E+00 Four significant figures, Scientific

Eigenvalue Algorithms: CADRE allows you to select from two types of algorithms for extracting eigenvalues and eigenvectors for the vibration solution. Each method can be employed with varying degrees of success depending on the particular problem.

Power Algorithm: Iterates on a each mode progressively for both frequency and mode shape. Then the dynamic matrix is deflated eliminating the derived mode from the dynamic matrix, this is repeated until as many modes and frequencies as possible are found. The ***Tolerance*** for the convergence criterion is shown in the text box at the right of the selection button. The maximum iterations is also shown. Both these values can be modified by the user.

QR algorithm: Calculates all frequencies simultaneously by iteration, then determines each mode shape individually by the inverse iteration method. The ***Tolerance*** for the convergence criterion is shown in the text box to the right of the selection button. It can be modified by the user.

Defaults: **Entry Options**

There are two options that can be changed:

Node Checking: Automatic node checking can be turned on or off on each entry form. This option sets the form startup default value for node checking in the entry mode for the Elements Form. Node checking identifies any attempt to connect an element to an unidentified node and gives the user a chance to add the unidentified node to the data file.

Sound on startup: Setting this option will result in the ***Sound*** item under the ***Options*** item on the menu bar to be checked on startup (sound enabled).

On Screen Element Editor

The Element Editor is reached by choosing the ***Screen Edit*** option under ***Edit*** on the Main Menu. First the Element Editor will be displayed. If desired, click on the ***Nodes...*** button on the Element Editor control panel or press ***Cont_Spacebar*** to enter the Nodal Editor.

In the on-screen element editing mode you can:

Split an element into two elements.

Delete an element.

Add a new element.

To activate the Element Editor, select ***Screen Edit*** from the drop down menu under the Edit item on the Main Menu bar. The Element Editor panel will be displayed. It can be positioned at a convenient spot by dragging it with the mouse.

Once the Element Editor is activated, each time you click and drag with the mouse on the drawing screen, a frame will appear. Any nodes enclosed by the frame will be marked with an square white bullet that is context sensitive. Both structural nodes and reference points will be tagged in this way.

There is a limit of approximately 240 nodes that can be marked at one time, so just enclose the region you want to edit.

Use the left mouse button to select the first node and the right mouse button to select the next node. The selected nodes will be placed in the first and second boxes on the editor panel.

If there is an associated element between the selected nodes:

> The two listed nodes will rearrange themselves if necessary so that the first one is the element origin node and the second is the element axis node.
>
> A distinctive line will mark the element on the model.

The element reference node will be displayed.

The ***Split*** and ***Delete*** buttons will be enabled and the ***Add*** button will be disabled.

If there is no associated element between the selected nodes:

The reference node will be automatically filled in with the highest numbered node in the model.

The reference node can be changed by clicking on another node in the model with a mouse button while the Control key is held down.

A distinctive line will mark the model where an element can be added.

The ***Split*** and ***Delete*** buttons will be disabled and the ***Add*** button will be enabled.

Splitting Beam Elements:

First select the end nodes of the element to be split using the left and right buttons. A distinctive line will mark the element to be split.

Select the percent split by using the vertical scroll bar.

The percent split is measured from the origin node.

Press the ***Split*** button to split the beam into two segments.

Element properties for both segments are the same as the original beam.

The reference node for both segments is the same as for the original beam.

CADRE automatically renumbers the segments with identification numbers that reflect the new nodes.

New nodes, added by the splitting operation, are numbered one higher that the highest numbered node.

An ***Undo*** button is available immediately after the split operation.

Adding Beam Elements:

First select the origin and axis nodes of the element to be created using the left and right mouse buttons. A distinctive line will mark the location of the new element.

An element can't be added if an element already exists between the selected nodes.

Use the default reference node (highest node number in the model) or select one by holding down the Control Key while clicking on the chosen reference node with the mouse.

Select the ***Add*** button to insert an element between two identified nodes.

Element properties are initially a straight beam type with zero stiffness. The element identification number will be automatically set by CADRE based on the end node numbers.

Adding stiffness properties to the new element is done through the normal Elements Form.

The new element can be deleted with the ***Delete*** button.

Deleting Beam Elements:

First select the end nodes of the element to be deleted using the left and right buttons. A distinctive line will mark the element to be deleted.

Select the ***Delete*** button to delete the element.

An ***Undo*** button is available immediately after the deletion.

Access to hidden nodes: When node markers lie on top of each other, the ones in back can be accessed by clicking with the right mouse button. Each time a node marker is clicked with the right mouse button the marker is placed in back of any others in the same planar location.

After clicking on a node or highlighting an element, precise coordinates (or other attributes such as element properties, loads, masses, and restraints) can be set by entering the ***Edit*** item from the Main Menu bar. The editing forms will be displayed with the clicked node (or element) already displayed and ready for modification.

These editing operations can increase the bandwidth dramatically. Therefore, it may be beneficial to use the re-ordering routine under the ***Tools*** Item on the Main Menu bar.

Re-identify the nodes, if desired, using the appropriate tools under the ***Tools*** item on the Main Menu bar.

On-Screen Node Editor

The Nodal Editor is reached by choosing the ***Screen Edit*** option under ***Edit*** on the Main Menu. First the Element Editor will be displayed, then you can click on the ***Nodes...*** button on the Element Editor control panel or press ***Cont_Spacebar*** to enter the Nodal Editor.

From the Nodal Editor you can go back to the Element Editor by clicking on the button labeled ***Elements...*** or by using the ***Cont_Spacebar*** keys.

In the Nodal Editor you can:

Add new nodes

Move nodes

Delete unattached nodes

Entering the Nodal Editor mode changes the view to Isometric and allows the display of only 3 different planes, one at a time. These are the XY plane, the XZ plane or the YZ plane. Rotating the model is not possible while in the Nodal Editor.

Select the desired plane from the option buttons, then set the third dimension with the scroll bar or type it directly into the text box.

Hold down a mouse button and drag to open up a rectangle that can be dragged over the model. Nodes contained in the enclosed region will be marked with a bullet. Releasing the mouse button leaves the rectangle and the marked nodes.

Adding a node:

After clicking and dragging the highlighting rectangle over the model to show the desired nodes, move the mouse to where you want to add a node. Notice that the coordinates are shown in the status bar at the bottom of the screen.

While holding down the Control Key and the Alt Key simultaneously, click with the mouse and a node will be added at that point. If inside the highlighting rectangle, it will immediately be marked, otherwise it will not be visible until you drag the rectangle over this location

Moving a node:

After clicking and dragging the rectangle over the model to highlight the desired nodes with the markers, place the mouse on the node, hold down the left mouse button and drag the node to a new location.

Only the planar coordinates are changed by the move, the coordinate in the other dimension will remain unchanged.

Access to hidden nodes: When node markers lie on top of each other, the ones in back can be accessed by clicking with the right mouse button. Each time a node marker is clicked with the right mouse button the marker is placed in back of any others in the same planar location.

Undoing a move: A move operation can be undone by using the ***Undo move*** button which is visible immediately after a nodal move operation.

Deleting a node:

After clicking and dragging the rectangle over the model to highlight the desired nodes with the markers, place the mouse on the node, hold down the Control Key and the Alt Key and click with the mouse. The node will be deleted.

Only free nodes can be immediately deleted. If the node is connected to an element or used as a reference node by some element, you will be asked to provide another node identification to substitute. Otherwise you can not delete the node.

You can switch back to the Element Editor by clicking on the ***Elements...*** button on the control panel or by pressing ***Cont_Spacebar***.

These editing operations can increase the bandwidth dramatically. Therefore, it may be beneficial to use the re-ordering routine under the ***Tools*** Item on the Main Menu bar.

Re-identify the nodes, if desired, using the appropriate tools under the ***Tools*** item on the Main Menu bar.

Screen Editor

The Screen Editor is used to edit the graphic model while it is displayed on the screen. The following modes of editing operations can be selected.

- Element Editor
- Nodal Editor

The Element Editor mode is the default on entry into the screen editor. Toggling between the two modes of editing operation is done with ***Cont_Spacebar*** or by clicking on the buttons for ***Elements...*** or ***Nodes...*** on the editor control panels.

CADRE: Editing the data

CADRE allows you to edit the data file with the ***Edit*** item on the main menu bar.

Edit Operation	Allows changing the operation
Edit Title	Change document title as well as the input data file name using the Title Form
Edit Nodes	Change nodal data and nodal coordinates using the Nodes form
Edit Elements	Change elements properties and connections using the Elements Form
Edit Bounds	Change the boundary conditions using the Bounds Form
Edit Loads	Change loading conditions or structural influence coefficient points using the Loads Form
Edit Mass	Change Mass Node Degrees of freedom points and Mass Properties using the Mass Properties Form
Screen Editor	Perform certain edits directly on the graphic model while it is displayed on the screen using the Screen Editor. This includes adding elements, splitting elements, and deleting elements.

Selecting any of these items sets the *edit mode* for each of the data entry forms.

New items can be entered by using the ***Insert*** or ***New Page*** buttons. Items can be deleted with the ***Delete*** button. Selecting ***OK*** will keep all the changes made while in this form. Selecting ***Cancel*** will ignore all changes made while in the form.

Printing: Print Selection Form

The Print Selection Form is enabled when a request to print the results is received by CADRE. This form allows the user to select, by identification number, the nodes for which displacements will be printed and the elements for which internal loads will be printed.

The form first appears as a node selection screen then as an element selection screen.

CADRE allows the selection of ***All*** or ***None*** by means of check boxes.

CADRE allows the user to pick & choose items in the list box using the usual Windows controls (Shift and Control Keys with the mouse) for data selection:

Holding the Control key down and selecting item by item with a mouse click.

Holding the Shift key down and selecting a range of items with a mouse click.

Selecting a range of items by dragging the mouse.

MENU: Managing Printed Output

The printer fonts and styles can be set under the ***Options*** item on the main menu. These are set with the Defaults Form and can be saved as part of the user defaults.

Printing is accomplished under the ***File*** item on the main menu. In each of the print options listed in the sub menu, the user is first presented with a message box that provides the following controls:

Print button	- Go ahead and print.
Setup button	- Choose printer and page setup.
Cancel button	- Cancel and return to the main menu.
Margins	- Enables the setting of margins. All other buttons will be disabled until the **Accept** button is pressed.
Print-to-file check box	- The output will go to a user specified ASCII text files in the current directory instead of printing to the printer.
Feild separator list	- Only enabled when the *Print-to-file box* is checked. The selected field separator will be output after each data item making the text file compatible for importing into data base or spreadsheet programs where it can be accessed by plotting and graphing software.

CADRE provides for printing:

- A report of the input data file
- A report of the calculated results for:
  - -Displacements and element loads, or
  - -Mode shapes and frequencies
- The graphic display of the model
- The currently viewed information on the view screen

Print Input: The input data report is printed out. This consists of;

- Configuration data
- Nodal Coordinates
- Element Properties

- Boundary Conditions
- Load Conditions
- Inertial properties (for *Dynamic* models)

For a large model this can be quite a lot of printing so make sure its what you want to do. CADRE provides a *Cancel* button on the status bar which is active while the data is being printed to the printer buffer (not very long). After this you will have to enter the Windows Print Manager to cancel the print order.

Print Results: The output data report is printed according to the user selection.

For *Static* problems, a selection form is provided which allows the user to select the nodes for which displacements are to be printed and the elements for which internal loads will be printed.

For *Dynamic* problems, the set of frequencies and mode shapes are printed out in a report displaying the relative displacements of the mass nodes for each vibration mode.

Print Model: The model graphic is printed just as it is displayed on the screen.

Print Display: Selecting the currently displayed information will result in printing that portion of the data that is displayed on the view screen. This includes the graphic of the model if that is what is showing at the time.

Print to file: The input and output data reports can also be sent as ASCII text to a file so that they can be imported into a word processor. To print to a file check the ***print to file*** option before printing. The data are saved in a text file that is chosen by the user.

Solving: Vibrations

Solving for vibration modes and frequencies:

Some vibration calculation defaults can be set from the ***Options*** item on the menu bar. These relate to eigenvalue extraction routines, numbers of iterations, and the rate at which animation can be displayed on the screen.

To solve for vibration data, the model must have already been constructed along with the a definition for the mass node points and corresponding inertial data. The structural influence coefficient solution must have already been run so that the structural influence coefficient file is stored on disk in the same directory as the finite element model data file. If CADRE does not find such a file it will automatically solve for it.

Under ***Solve*** on the main menu bar, select ***Vibrations***.

If no structural influence coefficient file is found, CADRE will calculate it first.

If the data file has been modified and is unsorted, CADRE will sort it first.

If the structural influence coefficient file is not compatible, CADRE will report this fact, so the user can recalculate it from the ***Solve*** item on the menu bar.

If the structural influence coefficient file is not up to date, yet compatible with you model, bad results will occur without any indication. Ensure that the structural influence coefficient file has been updated after any structural change to your model.

Retention: CADRE asks for the number of modes to be retained and shows the maximum available with the current problem. Enter the number of modes desired. CADRE will calculate from the lowest frequency mode and up.

CADRE will stop at the highest mode achievable by iterative convergence, or at the maximum number of modes the user selects, whichever is less.

Eigenvalue extraction: While calculating, the status bar identifies the iteration algorithm and shows the progress of the eigenvalue extraction process. While an iteration is in progress the following iteration keys are available:

Escape	Stop iteration, accepts values and moves to next step
Cancel	Cancels the solution exercise

Solution Sequence:

> ***For the QR Algorithm*** all eigenvalues (frequencies) are calculated first. Then each eigenvector (mode shape) is calculated in turn.
>
> ***For the Power Algorithm***, each eigenvalue and eigenvector are calculated in pairs. The number of iterations is shown on the screen. The iteration will stop when sufficiently converged, or after exceeding the maximum iteration value set in the Defaults Form. The Default Form values can be changed even while calculations are taking place.

Interpolation: Only the nodes associated with lumped masses are included in the vibration analysis, therefore, interpolation is necessary in order to display the full model mode shapes.

So, after finding the modal displacements, CADRE will insert these displacements (treating them as specified displacements) into the overall finite element model and automatically recompute it using the ***Displacements*** solution routine. This results in a display of the model that is fully deformed according to the vibration modes.

Canceling calculation: The status bar displays a cancel button (only during calculation) that can be used to abort the calculation.

Solving: Structural Influence Coefficients (SIC)

Purpose of SIC: The structural influence coefficient matrix is developed and saved in a file for use in vibration analysis. This routine is enabled in the *Dynamic* operations mode.

In addition, the defaults options can be set so that the structural influence coefficient matrix is automatically calculated before any vibration solution. Regardless of this setting, if no structural influence coefficient file is found, CADRE will calculate it automatically before doing a vibration analysis.

The structural influence coefficient data cannot be printed out or viewed. They are saved in ASCII format on disk in file with the extension, (.sic) in the current directory and are used only for vibration analyses.

Solving: Select ***SIC*** from the ***Solve*** item on the main menu.

> If it is not enabled, the operation is not set to ***Dynamic***. Select edit operation and change to ***Dynamic***.

The solution will proceed and when finished, the structural influence coefficient file will be automatically saved under the input file filename using the (.sic) extension.

Selecting Rigid Body Modes

Rigid Body Modes Form is used to free the restraints.

The actual boundary conditions as prescribed on the Bounds Form are never actually modified. In fact, this data is used by CADRE to establish proper rigid body modes which are used to develop the display of the model as a freely vibrating body.

On entry to the form the rigid body modes that are not relevant to the defined model are disabled. The enabled modes are the degrees of freedom for which there is a defined load direction and opposing restraint system.

On entering the form all the enabled modes necessary to completely free the model are pre-selected. All the user need do is select ***OK*** to calculate the free-free modes.

Checking a box will add (superimpose) that rigid body motion and thereby eliminate that degree of restraint in the model.

> **It is important to understand this clearly since it may seem opposite from the way restraints are specified on the Bounds Form.**

Checking	**Releases that degree of freedom**
Unchecking	**Leaves that degree of freedom restrained.**

Rotation point: (Partially free models)

If all the available rigid modes are selected, the user need not select any rotation point for the rotary rigid body modes. In this case the rotary rigid body modes are referenced about the model centroid. However, if less than all the available rigid modes are selected a rotation point may be needed for consistency.

For example:

> If the available rigid modes were a Y displacement mode and Z rotation mode, and only the Z rotation mode was to be selected with the Y displacement left restrained, a point of rotation is needed. The rotation point should be selected at

the node of the boundary condition with the Y restraint, since locating it elsewhere would be inconsistent with having a Z rotation and zero displacement in the Y direction at that point.

When the user selects a partial combination of rigid modes in which a rotation point is needed, CADRE ***may*** display a rotation point selection list to allow the user to select a node as the rotation point. The list only shows those nodes that have boundary conditions. When only one boundary is available in the model, then it is obvious which to use, and CADRE selects this node automatically without asking.

In many cases, selecting less than all of the available rigid body nodes, when more than one rotation point possible, is not a good idea. There may be no consistent solution with that selection. In fact, this is the usual case; therefore:

> **Anytime CADRE displays the rotation point selection list the user should be very suspicious that there may be no consistent solution to his selection of rigid body modes. CADRE will calculate anyway but the results may appear strange. The request by CADRE for a rotation point should be taken as a warning.**

Exercise 6 in the Getting Started section of this help file provides an example that is useful in understanding the Rigid Body Modes Form and its uses. This example only has one boundary node so CADRE uses this as the rotation point. The user is not prompted.

Solving: Free-Free Vibrations

Solving for free vibrations:

Structural Influence Coefficients: The model must be developed along with the mass node points and corresponding inertial data. The structural influence coefficient solution must be run so that the structural influence coefficient file is stored on disk in the same directory as the input data file. CADRE performs free vibration modes by first calculating a set of modes with the restrained model (the model must ***always*** be restrained and stable for any calculation process in CADRE) in pass one, then superimposes rigid body modes to obtain the final modes in pass two.

Under ***Solve*** on the main menu bar, select ***Free vibrations***.

Rigid Body Modes: CADRE displays the Rigid Body Modes Form which allows choices in freeing the model restraints.

Retention: CADRE asks for the number of modes to be retained and shows the maximum available with the current problem. Enter the number of modes desired. CADRE will calculate from the lowest frequency mode to the highest achievable by iterative convergence, or at the maximum number of modes the user selected, whichever is less.

Double Pass Calculation: For free-free modes, CADRE conducts the complete vibration analysis in two passes. The pass number is shown in the program status box after the algorithm identification on the status bar. The steps in the solution are as follows:

1. **Restrained Modes**. The restrained modes are found by going through the process of extracting eigenvalues and eigenvectors for the restrained model. The number of modes retained for this pass is the number selected by the user PLUS the number of rigid body modes that will be included in the second pass.

   Any impurities in any of the calculated restrained modes will result in a very poor representation of the all the free-free modes. Its a good idea to run the restrained solution first, then view the modes to decide how many to retain for the free-free solution. When deciding this, remember that the number of restrained modes that will be used in the free solution will be the number of free-free ones you select PLUS a number equal to the number of rigid body modes.

2. **Superposition**. The set of restrained modes and the user-selected rigid body modes are superimposed to modify the dynamic matrix to form a free-free dynamic matrix.

3. **Free vibration modes**. The eigenvalue and eigenvector extraction process is repeated to determine free-free vibration frequencies and mode shapes. The number of modes retained in this pass is the number originally selected by the user.

4. **Interpolation**. Once the free-free modes are found, they are applied to the model as specified displacements. The finite element ***Displacements*** solution is automatically conducted as an interpolation tool so that the deformed shape of the complete model can be displayed.

Canceling calculation: The status bar displays a cancel button (only during calculation) that can be used to abort the solution process.

Solving: Loads and Displacements

The operation is normally set between the entry of boundary conditions and the entry of load conditions when creating a new model. The operation can also be set to *Static* under the ***Edit Operation***. This enables the ***Displacement*** item under the ***Solve*** item on the main menu.

This solution is used to calculate all the internal loads in the structure in both global and local coordinates.

The global displacements of all structural nodes are also calculated.

Select ***Displacements*** under the ***Solve*** item on the main menu.

Status bar: During the calculation the status bar displays the progress with messages such as:

- Forming Stiffness Matrix
- Inserting Boundary Conditions
- Forming load vector
- Forward Elimination
- Back Substitution

The status bar displays a cancel button (only during the calculation) that can be used to abort the calculation process. The status bar also displays a progress gauge.

The solution will continue in the background while the user performs other menu operations. However, those operations that would interfere with the calculation are disabled.

When a solution is reached (without errors), the model can be viewed with exaggerated displacements. Use the ***Displacement*** scroll bar to exaggerate the displacements.

MENU: Solving the finite element model

CADRE provides four solution choices:

Loads and Displacements

Structural Influence Coefficients

Vibrations

Free-Free Vibrations

Viewing Input, Model & Results

Data Controls For viewing the Input and Results data, Control buttons are provided. These are Page Up, Page Down, End, Home. The corresponding keys on the keyboard can also be used to quickly scan the data. Cadre provides a separate print command Print display which prints out the data section being viewed.

Input data file: The input data file can be listed and viewed on the screen.

The data sections that can be viewed are:

- Configuration
- Nodal Coordinates
- Element Connections
- Element Properties
- Boundary Conditions
- Loading Conditions
- Mass & Inertial Properties

Output data file: The output nodal displacements and element loads can also be viewed on the screen by selecting ***Internal loads, Displacements*** or ***Reactions*** items under the ***View*** item on the main menu bar.

Displacements: Each node is listed by identification number with the displacements, in global coordinates, for all six degrees of freedom.

Internal Loads: Each element is listed by element identification number followed by 4 rows of 6 numbers. The first two rows are the origin end and axis end forces respectively in global coordinates for all six degrees of freedom. The next two rows are the origin end and axis end forces respectively in local coordinates for all six degrees of freedom.

This item is also available with vibration modes but the results are not very meaningful, since mode shapes normalized to unity at the maximum displacement.

Reaction Loads: The external loads acting on the model at each bound node, in the global coordinate system, can be viewed.

Model viewing: When a completed input data file is loaded or entered, CADRE automatically presents the Main Menu with the graphical representation of the model displayed on the screen. The ***View model*** item on the menu provides a way to return to the model from other viewing procedures. This item can also be accessed from the keyboard by Control-F.

Graphic results: When valid results are calculated, a graphical view of the static displacements or vibration modes, as the case may be, can also be viewed.

Graphic controls There are a number of graphical controls for Viewing the model from different perspectives.

Menu: Printer and Screen Fonts

CADRE allows you to change the fonts that are used to display data on the screen or that are sent to the designated printer. The CADRE system fonts used for form labels, dialog boxes, and status messages can also be selected. Both are selected from the menu bar on the Setup & Default Form. This form is selected under ***Options*** on the menu bar of the main menu.

The default fonts sizes have been chosen to allow a good representation of all the data on a normal page and on the forms. Selecting other fonts and sizes (especially larger) may result in a confused printout or data display. Experiment to determine the best overall selection for the system.

Menu: Status Bar

CADRE provides a status bar at the bottom of the screen.

Operation box:	Shows the type of operation currently selected.
Status box	Shows the current CADRE state (entry, edit, view, animate, etc.)
Message box	Shows any CADRE information or system messages, including any errors encountered in drawing the model or solving it.
Run box	This counts iterations, counts down, counts up, etc. just to provide some intermediate information on progress between movements of the status gauge.
Cancel button	Displayed during calculations in order to allow the user to abort the calculation process.
Status gauge	Graphical representation of percent accomplished of tasks. The description of the task is shown in the status bar message box.

Make a bitmap of the model

Save a bitmap of the model (or portion of the display) to a file

When the model is displayed;

Press the left mouse button down and drag the rectangle to enclose the portion of the model to be saved.

Release the left mouse button.

The rectangular area changes to inverse and the Make Bitmap form appears.

Press the ***Save*** button to save to a file.

CADRE requests file name and automatically assigns the *.bmp extension.

On pressing OK a picture of the rectangular area will be saved on the designated path and file.

Note: Although shown inverse, the bitmap is saved in original colors.

Copy a bitmap of the model (or portion of the display) to the clipboard

When the model is displayed:

Press the left mouse button and drag the rectangle to enclose the portion of the model to be copied.

Release the left mouse button.

The rectangular area changes to inverse and the Make Bitmap form appears.

Press the ***Copy*** button to copy to the clipboard.

Note: Although shown inverse, the bitmap is saved in original colors.

CADRE: Main Menu

The Main Menu is the operations center for CADRE. This is where the model is displayed after it is constructed or loaded. From this point you can:

- Manage files
- Merge models
- Print Input and Output
- Edit the input data
- Change operations
- Solve for displacements, loads, and vibrations
- View input, output, and model graphics
- Sort elements and nodes
- Select user setup & defaults
- Select fonts for the screen and printer
- Make a bitmap of the displayed model

CADRE: Entering and Editing Data

Data entry - The data file is managed in two different modes. These are the entry and edit modes obtained when ***New*** is selected from the ***File*** item on the menu bar of the Main Menu or when ***Edit*** is selected from the menu bar of the Main Menu. The controls on each data entry form are slightly different and work in different ways depending on the mode (entry or edit).

Data forms - The data are entered in five forms:

- Title Form
- Nodes Form
- Elements Form
- Bounds Form
- Loads Form (*Static*)
- Mass Properties Form (*Dynamic*)

Screen Editor - A large amount a editing can be accomplished directly on the visual display of the graphic model using the Screen Editor.

CADRE: Setting The Operation

There are two modes of operation in CADRE.

Static	Loads and Displacements of a *Static* model
Dynamic	Vibration frequencies and mode shapes for a *Dynamic* model.

The data entry requirements differ for these two modes of operation. In the *Static* mode, loads data are entered; in the *Dynamic* mode, mass data are entered. For a new model in the entry mode, the operation is selected between the entry of boundary conditions and the entry of loads (*Static*) or mass properties (*Dynamic*). All other data for the models are the same. The operation can be changed from the ***Edit*** item on the menu. Selecting ***Edit Operation*** on the Main Menu will allow the operation to be changed.

Static: Selecting ***Static*** enables the solving routine for *Displacements*. When entering a new *Static* model, the user will pass through the Loads Form (for entry of the nodal forces for a load condition) and then directly to the Main Menu.

Dynamic: Selecting ***Dynamic*** will enable the ***SIC***, ***Vibration***, and ***Free Vibration*** solution options under the ***Solve*** item on the main menu bar. The ***Edit Loads*** and ***View Loads*** items will be changed to ***Edit Mass*** and ***View Mass***. When entering a new *Dynamic* model the user will pass through the Mass Properties Form (to enter mass and inertial data) before arriving at the Main Menu.

Changing from Dynamic to Static: The input data file is saved according to the operation mode that is selected at the time of the save. This operation mode is encoded in the data file itself. In addition, CADRE expects the data to consist of different types of information depending on the operation selected (i.e. SIC file name, mass data, etc.).

Therefore:

> **Take care in changing a Dynamic file back to the Static operation. If it is saved in this form the reference to the structural influence coefficient file and all the inertial data may be lost. Also any results saved under the Dynamic operation mode may no longer be valid under the Static operation.**

CADRE: Sorting Nodes and Elements

CADRE provides for sorting the nodes and elements under the item ***Tools*** on the menu bar.

Sorting Nodes: The internal reference to nodes is by their rank order . Therefore, it is essential that the nodes be properly arranged before a solution is attempted. Sorting the nodes arranges them according to the identification number that the user assigns to each node.

This sorting procedure provides an opportunity to insert a node into the list with a minimal change in the bandwidth of the stiffness matrix. For example, if the user has an element with node numbers 1 and 2, and he wishes to divide it into two elements, he can edit the nodes and elements which might involve inserting a new node number 1.5 between 1 and 2. Regardless of where the user entered this node on the Nodes Form the sorting operation will arrange it between node 1 and node 2 and adjust all other references accordingly.

The nodes can always be renumbered with a starting value and an incremental value. This is accomplished under the ***Tools*** item on the main menu.

Sorting separates out the non-structural reference nodes and re-identifies all internal references to nodes in the element connections data, the boundary conditions data, and the loads data.

Automatic sorting: The nodes will be sorted when the ***OK*** button is selected from a data entry form. This helps ensure that the data are in a form that can be plotted and solved.

Disabling sorting: For a large model where nodal sorting may take several minutes, you may want to disable the automatic node sort. This could be useful when you plan to make a change that will not effect the arrangement but would nevertheless trigger a sorting operation (e.g. a change in a stiffness property for an element). This can be accomplished under the ***Tools*** item on the menu bar by checking ***Disable sorting***. CADRE will not sort the nodes with this item checked unless you manually select ***Sort nodes*** under the ***Tools*** item.

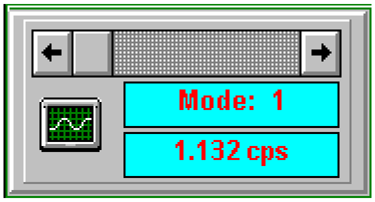
If you disable automatic sorting; take special care to ensure that sorting gets accomplished before solving if you make a change that would warrant sorting.

Manual sorting: You can manually sort nodes or elements at any time by selecting the ***Sort nodes*** or ***Sort elements*** under the ***Tools*** item on the menu bar. If errors occur during plotting or solving, try manually sorting.

Sorting elements, mass, loads, and bounds: It is not necessary to sort these items. However, it may be useful to sort them in order to find them more easily in the editor and in the output lists, especially for a large file. They can be sorted by selecting the ***Sort elements, mass, loads, or bounds*** items under the ***Tools*** item on the main menu.

Viewing the vibration modes

Viewing and animating vibration modes:



The Mode Panel appears in the upper left corner when modal data are available for viewing.   This panel is for selecting and animating the mode.

The Mode Panel provides:

Mode selector: The Mode Panel scroll bar is used to select the mode to display. The model is then drawn with a preset amount of displacement in the shape of the selected mode.   This displacement can be increased or decreased as desired by the ***Displacement*** scroll bar.

Mode No. box: Displays the number of the mode.

Frequency box: The frequency in cycles/second is displayed in the frequency text box.

Animation button: The ***Animate*** button on the Mode Panel is used to set the model to vibrating in the selected vibration mode.   While animation is in effect many menu items are disabled.   They will be re-enabled when animation is terminated with the ***Animate*** button.

The model can be rotated and manipulated with the other graphic controls while animation is in effect.

The ***Animate*** button is red and appears depressed while animation is in effect.

Other controls:

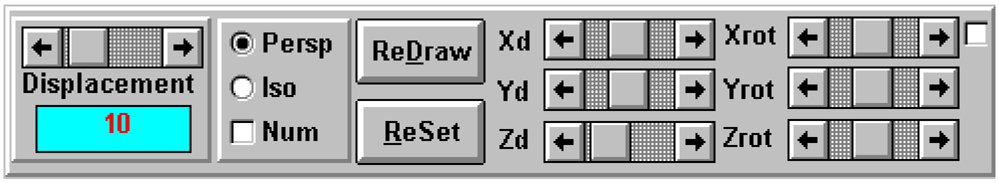
An alternative means of initiating and terminating animation is to select ***Animate*** from the ***Tools*** item on the menu bar, or by pressing [Control-A] from the keyboard.

A full screen view can be obtained by selecting ***Toggle display*** from under the ***View item*** on the menu bar. CADRE returns to the normal display screen when this menu item is depressed again. This can be done while animation is in effect. A keyboard hot key, Control-T, can also be used instead of the menu item.

CADRE: Viewing the Model

Viewing and manipulating the model:

The model is automatically displayed on the screen after it is successfully entered or loaded from disk. The graphics controls are displayed above and under the menu bar on the Drawing Control Panel and can be used to manipulate the graphic for different views. CADRE also provides for <u>viewing the vibration modes</u>. The colors of the graphic display can be changed with the <u>Defaults Form</u>.



Note: Some controls are only visible while results are available for viewing.

There are two control buttons on the <u>Drawing Control Panel</u>:

Reset	This button returns the scroll bar location and orientation controls back to the form entry positions. The model is redrawn in the original position.
Redraw	Causes a forced redraw of the model with the current data. This is necessary after returning from any data editing that would have changed the model display.

There are two view modes for the model that can be selected with the option buttons on the <u>Drawing Control Panel</u>:

Persp	Provides an perspective view of the model so that parts of the graphic that are farther away appear smaller.
Iso	Provides an isometric view as if the eye were far away.

Three scroll bars allow rotation of the model about the stationary screen coordinates. These coordinates are defined as X positive to the right, Y positive up and Z, positive out of the screen.

Xrot	controls rotation of the model about the horizontal axis.

Yrot controls rotation of the model about the vertical axis.

Zrot controls rotation of the model about the Z axis.

Two scroll bars move the center of the screen relative to the model.

Xd Moves the model left or right.

Yd Moves the model up or down.

One scroll bar, ***Zd***, on the Drawing Control Panel controls the size of the model. It responds differently for perspective and isometric views. For the perspective view, Changing Zd allows the user to approach or retreat from the object and it grows larger or smaller respectively. For the isometric view, the model is simply magnified.

Two check boxes are also provided for speed of movement and numbering.

[Speed] The unlabeled check box in the extreme right hand upper corner of the Drawing control Panel is used to accelerate movement with scroll bars that manipulate the mode. When checked motion is 5 times as fast.

Num This Check box provides for showing the node identification numbers at the nodes. The colors for the bound node numbers and loaded node numbers can be selected on the Defaults Form.

The ***Displacement*** scroll bar on the Drawing Control Panel is only enabled if valid results have been calculated or loaded. It can be used to exaggerate the displacements so that the deformed model can be viewed.

The Mode Panel becomes visible only when vibration modes and frequencies are calculated or when previously calculated vibration data are loaded. This panel allows for viewing the vibration modes, displaying the frequency, and animating the mode.

If the defaults option ***Reposition on click*** is checked in the Defaults Form, the model will automatically be repositioned when the right mouse button is clicked and the clicked point will be centered in the small square (only visible if this option is selected) at the center of the drawing screen. In the isometric view mode this point will remain in exactly in the center during magnification. This allows for microscopic examination of a detail that is small relative to the model (such as a joint). Use ***F6*** to toggle this option.

Full Screen

A full screen view can be obtained by selecting ***Toggle display*** from the under the ***View*** item on the menu bar. CADRE returns to the normal display screen when this menu item is depressed again. This can be done while animation is in effect. These screens can also be toggled from the keyboard by pressing Control-T.

Files: Saving the input data file

Saving the input data file:

Although CADRE will often prompt the user to save the file, CADRE will never save it to the disk file until the user actually requests the save to take place.

The input data file can be saved from the ***Save File*** item under ***File*** on the main menu bar.

When a new file is started in the Title Form, it is given the default name:

new.fem

This is a file name in memory only, it is not automatically saved to disk. The first time the user tries to save this file the Save As Form will be presented so that the name can be changed to one chosen by the user. This form will also allow the directory to be chosen.

The input file can also be saved from within any entry form by selecting the ***Save*** button.

Saving from the entry form is a good idea when entering a new file in the *entry mode*. In *edit mode* the file should always be saved from the main menu ***File*** item.

The file can always be saved under a different name and directory by choosing the ***Save File As*** item.

The file will always be saved with an (.fem) extension, whether the user includes it or not.

Selecting a merging file

Selecting a merging a file:

The merging file is opened from the ***Merge*** item (under the ***File*** item on the Main Menu).

The Merge File Selection Form is presented. This form allows the selecting of the directory and file name. Only files with the (.fem) extension will be displayed for selection.

When a file is selected and the ***OK*** button is depressed, the Merge Form is presented.

Files: Opening the input data file

Opening the input file:

The input data file is opened from the ***Open File*** item (under the ***File*** item on the Main Menu).

The Open Files Form is presented.

This form allows the selecting of the directory and file name.   Only files with the (.fem) extension will be displayed for selection.

When a file is selected and the ***OK*** button is depressed, the file will be opened, and the model will be displayed ready for viewing.

Previously calculated results are *not* automatically loaded with the input data.   This is done separately as described under Reloading the results.

After opening, the file can be run with the ***Solve*** item on the main menu, or previously solved data can be loaded with the ***Reload Results*** item (under ***File*** on the main menu).

For quick opening of recently used files, the names of the last four opened files are listed under the ***File*** item on the menu bar.

Files: Saving the Results

Saving Results:

The results data file can be saved from the ***Save Results*** under ***File*** on the Main Menu).

The ***Save Results*** form is presented.

This form offers a suggested name and allows the changing of the file name and directory.

The output file will always be saved with a (.dta) extension, regardless whether the user includes it or not.

Files: Reloading the Results

Reloading the output file:

The output data file can be opened from the ***Reload Results*** item (under ***File*** on the Main Menu).

The Open Files Form will be presented.

This form will allow the selecting of the directory and file name. Only files with the (.dta) extension will be displayed for selection and CADRE will suggest a file selection.

After selecting the file and pressing ***OK*** the data will be loaded and the viewing screen will be set up for viewing the results. CADRE detects the type of data automatically (*Static* or *Dynamic*).

Glossary

Term	Definition
AE	The cross sectional area of the beam (A) multiplied by Young's modulus for the beam material (E). This is a measure of axial stiffness per unit length.
AEo	The value of AE at the origin node of the beam.
AEa	The value of AE at the axis node of the beam.
Areal Properties	The cross section properties of a beam including area, (A), area moment of inertia (Iy or Iz), polar area moment of inertia, (J). Also refers to the products of these *areal* terms with the elastic moduli (i.e. AE, EIy, EIz, JG)
Axis Node	The node that defines one end of the beam and together with the *origin node* defines the beam's Neutral Axis. (See beam local coordinate system figure.)
Axial Rigidity	The stiffness in compression or tension of a beam element.
Bandwidth	(or semibandwidth) The **semibandwidth** of the stiffness matrix is the greatest distance (columnwise) of any non-zero term of the matrix from the diagonal term. The bandwidth of a given finite element model is determined by the ordering of the nodes determined by the nodal identification convention of CADRE. The nodal number, or rank, is determined by the value of its nodal identification number, the larger the number, the greater its rank.
Beam Type	A designator for the types of beams used in CADRE (S for straight beam, T for tapered beam, and P for pinned beam.
CADRE	1) A really good finite element analysis program for frame type structures. 2) A small group of highly skilled people. 3) A frame type structure.
Centroid	The **mass** centroid is the center of gravity (C.G.) of a mass object.
Center of Gravity	The point in an object the object's mass could be concentrated. At this point, a force could be applied and the object's inertial reactions to this applied force would not result in a tendency for the object to rotate.
CGS	Centimeter-gram-second system of units. Consistent units dictates that force is expressed in dynes and mass in grams.
Degree of Freedom	The number of independent parameters needed to define the displacement state of a node. In CADRE each node has 6 degrees of freedom (X,Y,Z, and rotation about each axis). In reference to the full finite element model, the term is used to describe the total number of degrees of all nodes that make up the model.

DOF See Degree of Freedom.

DXF file Drawing Exchange File format created by Autodesk for AutoCAD. It has become an industry standard for the interchange of drawing files.

Drawing screen The screen on the main menu on which the model is drawn for viewing and manipulating.

Eigenvalue The roots of an equation (represented in the form of a determinant). These are the roots that make the value of the equation equal to zero. In vibration problems the natural frequencies are related to the Eigenvalues of the ***Dynamic*** system.

Eigenvector A matrix vector related to an eigenvalue. In vibration problems the eigenvectors are related to the displacement mode shapes of the vibrating system.

EIY Young's modulus for the beam material (E) multiplied by the moment of inertia of area of the beam cross section about the beam local coordinate system Y-axis (Iy). This is a measure of bending stiffness.

EIYa The value, EIy, at the axis node.

EIYo The value, EIy, at the origin node.

EIZ Young's modulus for the beam material (E) multiplied by the moment of inertia of the beam cross section about the beam local coordinate system Z-axis (Iz). This is a measure of bending stiffness.

EIZa The value, EIz at the axis node.

EIZo The value, EIz at the origin node.

Elastic Modulus Youngs modulus, E, or the Shear modulus, G.

Element Identification Number This identification number, assigned automatically by CADRE is used for reference to the element throughout CADRE. It is a decimal number based on the element's end nodes (i.e. the identification number 0046.0125 is the element whose end nodes are 46 and 125.

Element Properties The properties that describe an element including its type, nodes, and its stiffness properties.

Finite element analyst A finite element analyst is any user of a finite element program.

Force In CADRE a force is considered to be a load that tends to cause a body to translate (not rotate). The term Load can refer to either forces or moments.

FPS Foot-pound-second system of units. Consistent units dictates that force is expressed in pounds and mass in slugs (lb-sec{two.mrb}/foot).

Frequency The number of repetitions of a cycle of displacement in a given time period. The frequency can be the **circular frequency** (ω) given in radians per second or **Hertz** (f), given in cycles per second. They are related by $\omega=2\pi f$. CADRE presents vibration results in Hertz.

IPS Inch-pound-second system of units. Consistent units dictates that force is expressed in pounds and mass in lbsec^2/inch. This mass unit has no official name but is sometimes called a "bug". There are 12 slugs in one bug.

Iteration A repetitive calculation in which the input to the calculation depends on the result of the same calculation. The calculation is initiated with an assumed result and repeated until the output is the same as the input (or approximately so) at which point the result has **converged**.

Global Coordinates The Global coordinate system is the coordinate system that is defined for the finite element model as a whole. In CADRE it is a right-handed system.

Handlebar The small dash [-] located in the upper left corner of a form in Windows 3.1 that can close the form or provide access to the window control menu. This is replaced by an icon in Windows 95. Also Windows 95 provides an "X" in the upper right corner that can be used to close the window.

JG The polar moment of inertia of the beam cross section about the neutral axis (J) multiplied by the shear modulus for the beam material (G). This is a measure of torsional stiffness per unit length about the beam axis.

JGa The polar moment of inertia of the beam cross section about the neutral axis of the beam (J) at the axis node multiplied by the shear modulus for the beam material (G).

JGo The polar moment of inertia of the beam cross section about the neutral axis of the beam (J) at the origin node multiplied by the shear modulus for the beam material (G).

Force In CADRE a force is considered to be a load that tends to cause a body to translate (not rotate). The term Load can refer to either forces or moments.

Free-Free In CADRE its the same as free vibration. The name comes from simple beam vibration problems in which the beam is free and both ends, and consequently unrestrained.

Free Vibration In CADRE this refers to undamped, unforced **and unrestrained** vibration. Vibration of a free body.

Load The term Load can refer to either forces or moments.

Local Coordinates The coordinate system defined for an individual element. In CADRE it is a right-handed system. Element properties and internal loads are defined relative to the local coordinates.

MKS Meter-gram-second system of units. Consistent units dictates that force is expressed in newtons and mass in kilograms.

Mode Panel A control panel that appears after a dynamic solution. The panel allows the selection of modes, display of frequency, and animation control.

Mode Shape The displacement state of a dynamic system (dynamic finite element model) vibrating in one of its natural modes. The displacement is usually a relative displacement state with the maximum displacement set to unity.

Moment In CADRE a moment is considered to be a load that tends to cause a body to rotate (not translate). The term Load can refer to either forces or moments.

Node Checking A CADRE option in which nodes specified for elements, boundary condition, and loads, are checked for validity. If they are unknown, CADRE allows the user to temporarily go to the Nodes Form to enter the unidentified node.

Node Identification Number Although the rank order of the nodes is shown on the form, the nodes are defined in terms of an identification number (numerical decimal number) which can be different from the rank order of the node. This identification number is used for reference to the node throughout CADRE.

Origin Node The node that defines one end of the beam and the origin of the beam coordinate system. (See beam local coordinate system figure.)

Pinned Beam A beam element in which is free to pivot, unrestrained about one or both ends in one or more directions.

Reference node The node that together with the *origin node* and the *axis node* defines the X-Y plane and Y-axis of the beam local coordinate system. (See beam local coordinate system figure.)

Right hand rule The right hand rule defines the positive sense of moments or torques about an axis.

Right hand system With a right hand coordinate system the positive senses of the x, y, and z axes of a right hand coordinate system are indicated respectively, by the directions to which the thumb, index finger, and middle finger of the right hand point when they are extended to form right angles.

Rotation restraint X, Y, or Z rotation restraints are boundary conditions applied to a node so that the node is restrained in rotation about an axis parallel to the X, Y or Z (global coordinate system) axes respectively.

Shear modulus The constant of proportionality between shear stress and shear strain. Usually identified with the symbol **G**. Shear stress = G x Shear strain. See Youngs modulus.

Simibandwidth See ***bandwidth***.

Specified displacement: A boundary condition which is defined by a specified displacement in one or more degrees of freedom.

Static balance See Static moment.

Static moment The product of a mass and its distance from a reference axis. Sometimes called static balance.

Stiffness properties AE, EIy, EIz, JG, for the beam cross section. Sometimes refers to these properties divided by the element length.

Tapered beam A beam element in which the properties diminish or increase linearly from one end to the other.

Unit load A force of one unit (i.e. 1 pound in the English system). Used to determine the displacement (influence) per unit load when defining influence coefficients.

Youngs modulus The constant of proportionality between stress and strain. Usually identified with the symbol **E**. Stress = E x Strain; where Strain is in displacement per unit length and stress is in force per unit area. The units of **E** are force per unit area.

YPINa Pinned beam property. The condition indicator for the pin parallel to the Y-Axis at the axis node (0 - pin activated, 1 - for no pin, intermediate values indicate a degree of fixity).

YPINo Pinned beam property. The condition indicator for the pin parallel to the Y-Axis at the origin node (0 - pin activated, 1 - for no pin, intermediate values indicate a degree of fixity).

ZPINa Pinned beam property. The condition indicator for the pin parallel to the Z-Axis at the axis node (0 - pin activated, 1 - for no pin, intermediate values indicate a degree of fixity).

ZPINo Pinned beam property. The condition indicator for the pin parallel to the Z-Axis at the origin node (0 - pin activated, 1 - for no pin, intermediate values indicate a degree of fixity).

Title Form

PURPOSE:

The Title Form is used to define a title for the project.   This title is only used for identification of the results.   It is included on the output when the file is printed.

The Title Form is also used to define the file name for the finite element model input file.

The default file name is:

new.fem

This can be changed on the Title Form, or the first time when the user chooses to save the input data file.

Node Identification Number Although the rank order of the nodes is shown on the form, the nodes are defined in terms of an identification number (numerical decimal number) which can be different from the rank order of the node. The nodes will later be sorted, either automatically or manually, according to the assigned identification number. This identification number is used for reference to the node throughout CADRE. The identification number (**I**dent) is useful for the later insertion of new nodes in the model.

Nodes Form

PURPOSE:

The Nodes Form is used to enter or edit all the nodal coordinates. Additional reference points, needed to establish the orientation of some elements, are also entered on the Nodes Form. CADRE will automatically sort out which are reference nodes and which are structural nodes.

ENTERING AND EDITING NODAL DATA:

Entering data: The nodal coordinate entry form, (similar to all the data entry forms) is set up to allow the entry of one node per page. The pages are turned by the scroll bar on the right edge of the form or the ***Next*** and ***Back*** buttons (or ***Page-Up*** and ***Page-Down*** keys). In entry mode, a new page will automatically be added as you press ***Return*** or ***Enter*** key while in the last text box.

In *edit mode* CADRE will not allow user to progress beyond the last node without using the ***New Page*** button. ***New Page*** or ***Insert*** buttons must be used to add new items in the *edit mode*.

Identification number: Although the page number for each node is shown on the form, the nodes are defined in terms of an identification number which can be different from the rank order of the node. The nodes will later be sorted, either automatically or manually, according to the assigned identification number.

This identification number is used for reference to the node throughout CADRE. The identification number (Ident) is useful for the later insertion of new nodes in the model.

Changing node identification: In *edit mode*, the node identification can be modified. The user will be advised of the change and asked if all the other references to the old identification number should be updated. The user can select ***Yes***, ***No*** or ***Cancel***. Selecting ***No*** will make the change without updating other references. *This can result inconsistent data*.

CONTROLS AND OPTIONS:

Menu bar: The menu bar contains an Undo and a Help item:

Undo - will restore the coordinates of the currently displayed node to the last values written to memory.

Help - will show this help page (also use ***F1***).

Scroll bar: The scroll bar is a fast way of moving through the list of nodes. When moving off of a node with the scroll bar the data for that node are written to memory.

Control Buttons: For entering and editing nodes there are the following buttons:

OK Stores nodal data on the form to its place in memory and exits the Nodes Form. In entry mode CADRE progresses to the Elements Form, while in edit mode CADRE returns to the Main Menu. If changes have been made, sorting of the nodes is automatic.

Next Stores nodal data on the form to its place in memory and advances to the next page for entry or edit. In *entry mode* the next page initially displays zeros. In *edit mode* it shows the next node identification number and associated coordinates. In edit mode the ***Next*** button will not progress past the last node.

Back Stores nodal data on the form to its place in memory and displays the previous node.

Insert Stores nodal data on the form to its place in memory and sets up to insert the next node before the current one. It is not essential to use the insert button since the nodes will be sorted later according to the identification number.

Delete Deletes the node currently shown on the page, moving the higher ranking nodes down by one page.

Prev Scrn Only available in entry mode. It returns to the previous screen (the Title Form). You are prompted to save the file since backing up to the title screen in entry mode will erase the entry data and set you up for a new program. Opening this saved file later will return you to the nodes screen in the entry mode so that you can continue.

Cancel	Only available in edit mode. The changes made after entry to the form will be ignored and the data will return to the state existing on entry to the form or to the last save if the user has pressed the ***Save*** button during the edit session.
Exit	Only available in entry mode. CADRE exits the entry session allowing the user to return later and pick up at the same place. After prompting, CADRE saves the file. Later, when opening this file from the Main Menu, CADRE will return to the same place to continue the development of the new model.
New Page	Only available in edit mode. Provides for adding a new node page for entering another node.
Save	Saves the input data file. If saved in the entry mode, opening the file later will return you to the same point for entry nodes.
Import	Only available in the entry mode. Used for importing a set of coordinates. They must be in ASCII text format and listed in the following manner: X,Y,Z,X,Y,Z,X,Y,Z etc. This button allows the user to import nodal coordinate data generated in another application, such as a BASIC program or a word processor.

Handlebar: In *entry mode*, pressing the handlebar to close the form is the same as using the ***Exit*** button. In *edit mode*, pressing the handlebar will return you to the Main Menu (same as the Cancel button).

Keyboard:

Key	Operation
F1	Help on the specific input form.
Escape	Edit Mode: Cancel & return to main menu.
	Entry Mode: Return to previous entry form.

Page Down	Go to the next node page.
Page Up	Go to the prior node page.
Shift-Insert	Insert a node page before the displayed one.
Shift-Delete	Delete the displayed node page.
Home	Go to the first node page.
End	Go to the last node page.
Enter	Next data box. If in the last box, go to the next page.

Element Library Form

The Element Library Form is used to create or maintain the list of element properties. These elements can be referenced by a user-defined descriptive name. They can be selected from a menu and assigned to any element in the model. The Element Library Form is accessed by way of the ***Edit Library*** item under the menu item ***Library*** on the Elements Form menu bar.

Adding element definitions. Elements can be added to the list in three ways:

- Elements can be copied from the Elements Form and placed on the library list with a descriptive name. This is accomplished by the ***Define*** button on the Elements Form.
- Elements can be added to the list directly by using the ***Add*** button on the Element Library Form and then typing the properties onto the form.
- Elements defined for another model can be imported and merged with the current models library elements by using the ***Import*** button on the Element Library Form.

Deleting library definitions. Elements are deleted from the Element Library Form with the ***Delete*** button.

Assign. When in the Elements Form, use the ***Assign*** button to bring up the list of library elements. Then one of them can be selected and its properties will be assigned to the current model element by clicking with the mouse. The associated element library name will appear **Definitions** text box.

Match. This button is used to compare the current displayed element to the list library element properties and if a match exists, the library name is assigned to the displayed element.

Match all. This menu item under ***Library*** is the same as ***Match*** above except that all elements are matched with the library and assigned names according to any that match.

Update all. With this menu item under ***Library***, the entire list of elements in the model is compared (by assigned name) to the elements in the library and the properties are updated to correspond to the current library properties. With this feature you can modify all the elements of a certain class by modifying only the library definition and using ***Update all*** revise the elements.

See Also: Elements Form

Elements Form

PURPOSE:

The Elements Form is used to enter or edit all the elements, their properties and to connect them to the nodes to build the finite element model.

ENTERING AND EDITING ELEMENTS:

Entering data: The elements are entered by Identification number, Type, Origin Node, Axis Node, Reference Node, and then by their beam properties. The Elements Form, (similar to all the data entry forms) is set up to allow the entry of one element description per page. The pages are turned by the scroll bar on the right edge of the form or the ***Next*** and ***Back*** buttons (or ***Page-Up*** and ***Page-Down*** keys). In entry mode, a new page will automatically be added as you press ***Return*** or ***Enter*** key while in the last text box.

In *entry mode* the next element is initially shown as all zeros. This new element with zero entries will be automatically removed if you exit (Press ***OK***) without changing any data or pressing any other controls.

In *edit mode* the ***Enter*** key or ***Next*** and ***Back*** buttons will not allow going beyond the last element page (the ***New Page*** button is provided for this).

Element Identification number: The element identification number is for user reference and for use by the Element Editor. The elements can be sorted by identification number for easier reference. Elements are numbered automatically by CADRE based on their connecting nodes (e.g. 0102.0095 for the element with node number 102 on the origin end and node 95 on the other end).

Element types: The type of elements supported by CADRE are straight beams, tapered beams, and pinned beams. They are selected from the drop down box under element type. The Elements Form adjusts the data entry areas according to the type selected.

Connections: The nodes to which the beam is connected are selected from the list boxes for Origin Node, Axis Node, and Reference Node. Refer to the diagram below for a description of these terms. They can also be typed in directly. If a node is entered which is has not already been entered in the nodes form, and if node checking is selected, a prompt will be displayed that will allow you to return to the Nodes Form temporarily to enter the coordinates and identify the new node. You can ignore the prompt but eventually all nodes that an element uses must be identified in the list of nodes.

Properties: The element properties are entered according to the element type and are described with reference to a;

local coordinate system.

This coordinate system is defined with the abscissa along the beam axis and the origin at one end. A reference node is used to define the X-Y plane and establish the orientation of the coordinate system. It is a right handed coordinate system.

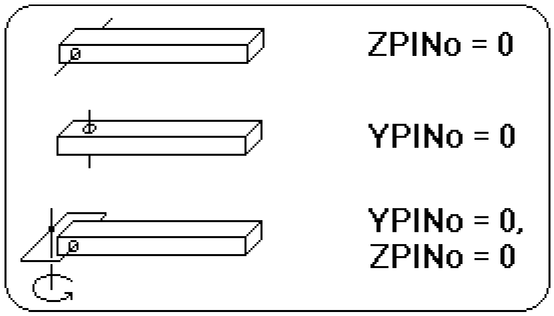
The element properties can be copied to the clipboard and pasted to other elements by using the menu bar items, or the ***F8*** and ***F9*** keys. Refer to the ***Glossary*** for a description of the properties nomenclature.

Type	Properties
Straight Beam	AE, Ely, Elz, JG
Tapered Beam	AEo Elyo, Elzo, JGo, AEa, Elya, Elza, JGa
Pinned Beam	AE, Ely, Elz, JG, YPINo, LPINo, YPINa, ZPINa

Pinned properties: For the pin properties of the pinned beam, a number 0 indicates a node free to rotate in the indicated direction. For example:

YPINo = 0 implies the beam is allowed to freely rotate at the origin node about an axis parallel to the Y axis (local coordinates).

ZPINa = 1 implies that the beam is restrained at the axis node about an axis parallel to the Z axis (local coordinates).



CADRE accepts 0 to define a free pin and 1 to define a rigid pin (i.e. no pin). Any fraction between 0 and 1 can be used to define a degree of end fixity according to the following formula:

$$f = \frac{1}{1 + \frac{2EI}{KL}}$$

Where: EI is the beam bending stiffness of the rigid restrained beam in the direction of pin rotation.

K is the torsional spring constant (torque/radian) defining the pin rotational rigidity.

L is the length of the beam.

CONTROLS AND OPTIONS:

Menu bar: The menu bar contains ***Undo***, ***Edit***, ***Library***, and ***Help*** items:

Undo This is used to restore the current element data when it has been changed but not yet saved to memory.

Edit **Copy and Paste.** These can be used to ***Copy*** the current element properties to the clipboard (***F8***), or to ***Paste*** properties data from the special clipboard to the current element (***F9***).

Replace all This is used to copy the element properties on the special clipboard (put there by ***F8***) to all elements in the model whose properties match the current displayed element.

Library This item can be used to enter the Library Elements Form where a menu of library elements can be prepared and maintained (***Edit Library***). It also contains ***Replace all***, ***Update all***, ***Define***, ***Assign***, ***Match*** and ***Match all*** menu items. See Element Library Form.

Help This will show this help page (Also ***F1*** can be used).

Scroll bar: The scroll bar is a fast way of moving through the elements. When moving off of an element with the scroll bar the data for that element are written to memory.

Control Buttons: For entering and editing elements there are the following buttons:

OK Stores element data on the form to its place in memory and exits the Elements Form. In entry mode CADRE progresses to the Bounds Form while in the edit mode CADRE returns to the Main Menu. If changes have been made sorting of the *nodes* is automatic.

Next Stores element data on the form to its place in memory and progresses to the next element page for entry. In the entry mode the next element is initially displayed with zeros. In edit mode CADRE shows the next element in the list of elements. In edit mode the ***Next*** button will not progress past the last element page (See **New Page** button).

Back Stores element data on the form to its place in memory and displays the previous element.

Insert Stores element data on the form to its place in memory and sets up to insert a new element page before the current one. It is not really essential to use the ***Insert*** button since the elements don't need to be in order. Also, the elements can be sorted (main menu tools) later according to the element identification number.

Delete Deletes the element page currently shown and renumbers the remaining pages.

Prev Scrn Only available in entry mode. It returns to the previous screen (the Nodes Form).

Cancel Only available in edit mode. The changes made after entry to the form will be ignored and the data will return to the state existing on entry to the form, or to the state of the last save if the user has pressed the ***Save*** button during the edit session.

Exit Only available in entry mode. CADRE exits the entry session allowing the user to return later and pick up at the same place. After prompting, CADRE saves the file. Later, when opening this file from the Main Menu, CADRE will return to the same place to continue the development of the new model.

New Page Only available in edit mode. Provides for adding a new element page to the file at the bottom the elements list.

Define Presents an input box for a descriptive name, then places this element with the displayed properties on a menu of defined elements. These element properties can be selected with the ***Assign*** button. After executing ***Define***, the current displayed element is assigned the new library item by name.

One way to break this link between this element and the library item is to execute ***Define*** again with the element displayed and enter "None" as the descriptive name. Another way is with the ***Assign*** button as described below.

Assign Presents the Library Elements List. Clicking on one of the elements on the list will assign those properties to the current element and link that element to that library item. This button is only available if there are elements on the list.

Unassigning: The way to break the link between an element and its associated library item is to execute the ***Assign*** button then select the ***None*** button above the selection list.

Match Clicking on this button will assign the name of a library item to the displayed element based on matching the element properties. The match is made with the first matching element on the Library Elements List. The name appears in the definition box.

Handlebar: In *entry mode*, pressing the handlebar to close the form is the same as using the ***Exit*** button. In *edit mode*, pressing the handlebar will return you to the Main

Menu (same as the Cancel button).

Keyboard:

Key	Operation
F1	Help on the specific input form.
F8	Copy the current element properties information to the special element clipboard.
F9	Paste the properties of the element on the clipboard to the current element.
Escape	Edit Mode: Cancel & return to main menu.
	Entry Mode: Return to previous entry form.
Page Down	Go to the next element page.
Page Up	Go to the prior element page.
Shift-Insert	Insert an element page before the displayed one.
Shift-Delete	Delete the displayed element page.
Home	Go to the first element page.
End	Go to the last element page.
Enter	Next data box. If in the last box, go to the next page.

See Also: Library Elements Form

Bounds Form

PURPOSE:

The bounds form is used to define or edit all the bound nodes.

ENTERING AND EDITING BOUNDARY CONDITIONS:

Entering data: The bound nodes are entered by Node identification (typed or selected from the list box), X restraint, Y restraint, Z restraint, X_rotation_restraint, Y rotation restraint, Z rotation restraint. The node is either fixed or free which is set by the check boxes, or it can be entered as a specified_displacement. To enter a specified displacement you must first uncheck both the check boxes. Then you can enter the value in the adjacent text box.

The restraints for each bound node are entered on a page of the Bounds Form. Use the Next button or scroll bar to move to the next bound node page. In entry mode the next bound node is initially shown as completely free. In *entry mode* this additional boundary condition will be deleted if you exit (press ***OK***) without making changes. In *edit mode* CADRE will not progress beyond the last bound node page (use the ***New Page*** button to add new bound nodes).

CONTROLS AND OPTIONS:

Menu bar: The menu bar contains an ***Undo*** and a ***Help*** item:

Undo	will restore the current boundary condition to the last saved values.
Edit	**Copy** and **Paste** boundary conditions from one node to another.
	Replace all: Boundary conditions, previously copied from a node with the **Copy** command, are copied to all nodes matching the boundary conditons of the currently displayed node.
	Setup all: All structural nodes are set up as boundary conditions but with no restraints (free).
Help	will show this help page.

Scroll bar: The scroll bar is a fast way of moving through the boundary conditions. When moving off of a boundary condition with the scroll bar, the data for that boundary condition are written to memory.

Control Buttons: For entering and editing boundary conditions there are the following buttons:

OK Stores boundary data shown on the form to its place in memory and exits the Bounds Form. In entry mode CADRE progresses to the Loads Form, or the Mass Properties Form depending on the operation mode. In edit mode CADRE returns to the Main Menu.

Next Stores boundary data on the form to its place in memory and progresses to the next bound node page for entry. The next bound node page is initially displayed as free in entry mode. In edit mode the ***Next*** button will not progress past the last boundary condition.

Back Stores boundary data on the page to its place in memory and displays the previous bound node page.

Insert Stores boundary data on the page to its place in memory and sets up to insert a new bound node page before the current one.

Delete Deletes the bound node page currently shown on the form, renumbering the remaining pages.

Prev Scrn Only available in entry mode. CADRE returns to the previous screen (the Elements Form).

Cancel Only available in edit mode. The changes made after entry to the form will be ignored and the data will return to the state existing on entry to the form, or to the state of the last save if the user has pressed the ***Save*** button during the edit session.

Exit Only available in entry mode. CADRE exits the entry session allowing the user to return later and pick up at the same place. After prompting, CADRE saves the file. Later, when opening this file from the Main Menu, CADRE will return to the same place to continue the development of the new model.

New Page Only available in edit mode. Provides for adding a new bound node page.

Save Saves the input data file. If saved in *entry mode*, opening the file later will return you to the same point for entry of bound node data. If saved in *edit mode*, opening the file will return to the Main Menu.

Handlebar: In *entry mode*, pressing the handlebar to close the form is the same as using the ***Exit*** button. In *edit mode*, pressing the handlebar will returns to the Main Menu (same as the Cancel button).

Keyboard:

Key	Operation
F1	Help on the specific input form.
Escape	Edit Mode: Cancel & return to main menu.
	Entry Mode: Return to previous entry form.
Page Down	Go to the next bound node page.
Page Up	Go to the prior bound node page.
Shift-Insert	Insert a bound node page before the displayed one.
Shift-Delete	Delete the displayed bound node page.
Home	Go to the first bound node page
End	Go to the last bound node page
Enter	Next data box or if at the last box, go to the next page.

Loads Form

PURPOSE:

The Loads Form is used to define or edit all the external loads acting on the nodes in the finite element model.

ENTERING AND EDITING NODAL FORCES:

Entering load data: The nodal forces are entered by node identification (typed or selected from the list box), X load, Y load, Z load, X moment, Y moment, Z moment. The Loads Form, (similar to all the data entry forms) is set up to allow the entry of one loaded node per page. The pages are turned by the scroll bar on the right edge of the form or the ***Next*** and ***Back*** buttons (or ***Page-Up*** and ***Page-Down*** keys). In entry mode, a new page will automatically be added as you press ***Return*** or ***Enter*** key while in the last text box.

Pressing Return or Enter after each entry will advance to the next entry box. On the last entry (Z moment), CADRE advances to the next loaded node page, showing all zeros for loads.

In *entry mode*, if you exit without making any changes to these new zero loads, and if they are not stored by exercising some control, then this load condition will be deleted on exit (pressing ***OK***). In *edit mode* CADRE will not progress beyond the last loaded node page (use the ***New Page*** button to add new boundaries).

CONTROLS AND OPTIONS:

Menu bar: The menu bar contains an ***Undo*** and a ***Help*** item:

Undo will restore the current displayed load condition to their last values saved to memory.

Edit **Copy** and **Paste** properties from one node to another.

Replace all: Load properties previously copied from a node with the menu **Copy** command are copied to all nodes matching the load properties of the currently displayed node.

Setup all: All structural nodes are set up as loaded nodes with zero load.

Help will show this help page.

Scroll bar: The scroll bar is a fast way of moving through the loaded node pages. When moving off of a node page with the scroll bar the load data for that node are written to memory.

Control Buttons: For entering and editing loads, there are the following control buttons:

OK Stores load data shown on the form to its place in memory and exits the Loads Form. In either the entry mode or the edit mode CADRE progresses to the Main Menu

Next Stores load data on the form to its place in memory and progresses to the next page for entry. The next load page is initially displayed as all zeros in entry mode. In edit mode the ***Next*** button will not progress past the last page. (See **New Page**)

Back Stores load data on the form to its place in memory and displays the previous loaded node page.

Insert Stores load data on the form to its place in memory and sets up to insert a new page before the current one.

Delete Deletes the loaded node page currently shown on the form, renumbering the remaining pages.

Prev Scrn Only available in entry mode. CADRE returns to the previous screen (the Bounds Form).

Cancel Only available in edit mode. The changes made after entry to the form will be ignored and the data will return to the state existing on entry to the form, or to the state of the last save if the user has pressed the ***Save*** button during the edit session.

Exit Only available in entry mode. CADRE exits the entry session allowing the user to return later and pick up at the same place. After prompting, CADRE saves the file. Later, when opening this file from the Main Menu, CADRE will return to the same place to continue the development of the new model.

New Page Only available in edit mode. Provides for adding a new load page in edit mode.

Save Saves the input data file. If saved in *entry mode*, opening the file later will return you to the same point for entry of load conditions. If saved in *edit mode*, opening the file will return to the Main Menu.

Handlebar: In *entry mode*, pressing the handlebar to close the form is the same as using the ***Exit*** button. In *edit mode*, pressing the handlebar will returns to the Main Menu (same as the ***Cancel*** button).

Keyboard:

Key	Operation
F1	Help on the Loads Form.
F8	Copy
F9	Paste
Escape	Edit Mode: Cancel & return to main menu.
	Entry Mode: Return to previous entry form (bounds).
Page Down	Go to the next loaded node page.
Page Up	Go to the prior loaded node page.
Shift-Insert	Insert a loaded node page before the displayed one.
Shift-Delete	Delete the displayed loaded node page.
Home	Go to the first loaded node page.
End	Go to the last loaded node page.
Enter	Next data box. If at the last box, go to the next page.

Mass Properties Form

PURPOSE:

This form is used to enter the degrees of freedom and mass properties at the mass node points.

USING THE FORM:

Entering the data: This form is accessed when ***Dynamic*** is selected during entry of a new program or by selecting the ***Edit Operation*** item on the Main Menu, changing to ***Dynamic,*** and then selecting ***Edit Mass***. The mass nodes are identified according to the identification number for the node. The values for mass and inertia for a new model are initially set to zero and the values for mass c.g. location are initially set equal to the coordinate position of the identified node.

The Mass Properties Form, (similar to all the data entry forms) is set up to allow the entry of one nodal mass description per page. The pages are turned by the scroll bar on the right edge of the form or the ***Next*** and ***Back*** buttons (or ***Page-Up*** and ***Page-Down*** keys). In *entry mode*, a new page will automatically be added as you press ***Return*** or ***Enter*** key while in the last text box.

For each node that is to become a mass node, the node identification number must be typed, or selected from the list, and the applicable degrees of freedom must be checked in the check boxes.

Offset masses: A unique feature of CADRE is that the masses can have local rotational inertia about all three axes and can be offset from the actual node to which they are associated. This provides for a way of representing the static moment with respect to the node without using additional nodes in the problem. (The tradeoff is that this may increase the required degrees of freedom)

Degrees of freedom: The degrees of freedom (DOF) must be sufficient to define the inertial forces that can account for the offset masses. When the DOF are not sufficient to take the static moment and inertia of the offset into account, the c.g. location for the mass in that dimension will be assumed to be the same as the nodal coordinate in that dimension regardless of any numbers in these boxes. Therefore, the degrees of freedom should be set before the mass data are entered.

Mass data: The entry data required are the mass and mass moment of inertia (Ix, Iy, and Iz) where each of the inertial terms are about an axis through the center of the mass object parallel with the indicated axis. The user enters all inertial data relative to the mass object centroid, then CADRE determines all inertial data relative to the identified node.

Nodal Inertial Properties: CADRE calculates the total static moment, cross products of inertia, and moments of inertia, all about the referenced nodes, taking into account both the mass centroid and its offset from the node. Cross products of inertia of the mass about its own centroid are assumed to be zero. Mass, Inertia(I), cross products(P), and static moments(S) *about the node* are taken into account only when the appropriate degrees of freedom are selected according the following table:

Minimum DOF for prescribed inertia

Degree of Freedom	X	Y	Z	Xrot	Yrot	Zrot
Mass	#	or #	or	#		
Ix				#		
Iy					#	
Iz						#
Pxy				#	#	
Pxz				#		#
Pyz					#	#
Sxy	#				#	
Syx		#		#		
Sxz	#					#
Szx			#	#		
Syz		#				#
Szy			#			#

Units: The mass data must be entered in units consistent with the remainder of the data that has been input. The samples provided with the CADRE are in the inch-pound-second system.

System	Mass Name	Mass Units	Inertia Units
foot pound sec	slug	lb.sec^2/ft	slug.ft
inch pound sec	bug	lb.sec^2/in	bug.in
newton meter sec	kilogram	n.sec^2/meter	kg.meter
dyne cm sec	gram	dyne.sec^2/cm	gm.cm

Mass location: The location of the mass is entered in terms of global coordinates. Often the mass is located directly on the node itself. This is the default location. For offset masses the user must type the global coordinates of the masses directly in the text boxes. If desired, a button entitled ***Locate at Node*** can be used to set the center of gravity at the node.

Controls and options:

Menu bar: The menu bar has three items:

Undo Restores the current entries to the last values written to memory.

Edit **Copy** and **Paste** properties from one node to another.

Replace all: Properties previously copied from a node with the **Copy** command are copied to all nodes matching the properties of the currently displayed node.

Setup all: All structural nodes are set up as mass nodes with zero mass and zero degrees of freedom.

Help Shows this help page.

Scroll bar: Stores the data and scrolls to the next mass node page.

Buttons:

OK Stores the displayed data and returns to the Main Menu.

Next Stores the displayed data and advances to the next page.

Back Stores the displayed data and returns to the previous page.

Insert Stores nodal inertial data on the form to its place in memory and sets up to insert a new mass page before the current one.

Delete Deletes the mass properties page currently shown on the form, renumbering the remaining pages.

Cancel The changes made after entry to the form will be ignored and the data will return to the state existing on entry to the form, or to the state of the last save if the user has pressed the ***Save*** button during the edit session.

Prev Scrn Only available in entry mode. CADRE returns to the previous screen (the Bounds Form).

Exit Only available in entry mode. CADRE exits the entry session allowing the user to return later and pick up at the same place. After prompting, CADRE saves the file. Later, when opening this file from the Main Menu, CADRE will return to the same place to continue the development of the new model.

New Page Only available in edit mode. Provides for adding a new mass properties page in edit mode. Page Down key will perform the same function.

Handlebar: After pressing the handlebar CADRE returns to the Main Menu (same as the ***Cancel*** button).

Keyboard:

Key	Operation
F1	Help on the inertia input data form.
F8	Copy.
F9	Paste.
Escape	Edit Mode: Cancel & return to main menu.
	Entry Mode: Return to previous entry form (bounds).
Page Down	Go to the next mass node page
Page Up	Go to the prior mass node page
Home	Go to the first mass node page
End	Go to the last mass node page
Shift-Insert	Insert a mass node page before the displayed one
Shift-Delete	Delete the displayed mass node page
Enter	Goes to the next text box, if its the last text box, CADRE goes to the next mass node page.

Node Checking: A CADRE option in which nodes specified for elements, boundary condition, and loads, are checked for existence in the list of identified nodes. If they are unknown, CADRE allows the user to temporarily return to the Nodes Form to enter the new unidentified node.

Specified Displacement: A boundary condition which is defined by a specified displacement in one or more degrees of freedom.

Rotation restraint: X, Y, or Z rotation restraints are boundary conditions applied to a node so that the node is restrained in rotation about an axis parallel to the X, Y or Z (global coordinates) axes respectively.

Handlebar: The small dash [-] located in the upper left corner of a window in Windows 3.1. Double clicking on this dash will close the window. This is replaced by a small icon at the same location in Windows 95. Also Windows 95 provies a "X" in the upper right corner that performs the same function with just one mouse click.

Reference Node: The reference (or orientation) node used to establish the local x-y plane for the beam element and therefore give a sense of direction to the element stiffness properties and local element loads.

EIy: Bending stiffness property of a beam about an axis perpendicular to the beam axis and parallel to the Y axis.

EIz: Bending stiffness property of a beam about an axis perpendicular to the beam axis and parallel to the Z axis.

Origin Node: In the <u>local coordinate system</u> for a beam element this refers to the node at the end of the element at the origin.

Axis Node: In the local coordinate system for a beam element this refers to the node at the end of the beam farthest from the origin.

JG: Refers to the torsional stiffness about the beam axis (X axis).

Static moment: The product of a mass and its distance from a reference axis.

Degree of Freedom: The number of independent parameters needed to define the displacement state of a node. In CADRE each node has 6 degrees of freedom (X,Y,Z, and rotation about each axis). In reference to the full finite element model, the term is used to describe the total number of degrees of all nodes that make up the model.

Centroid: The mass centroid is the center of gravity (C.G.) of a mass object.

Unit Load: A force of one unit (i.e. 1 pound in the English system). Used to determine the displacement (influence) per unit load when defining influence coefficients.

Axial Rigidity: The stiffness (AE) in compression or tension of a beam element.

Beam Type - A designator for the type of beam used in CADRE (S for straight beam, T for tapered beam, and P for pinned beam.

Origin Node - The node that defines one end of the beam and the origin of the beam coordinate system. (See beam local coordinate system figure.)

Axis Node - The node that defines one end of the beam and together with the Origin Node defines the beam's Neutral Axis. (See beam local coordinate system figure.)

Reference Node - The node that together with the Origin Node and the Axis Node defines the X-Y plane and Y-axis of the beam local coordinate system. (See beam local coordinate system figure**.)**

AEo - The cross sectional area of the beam (A) at the origin node of the beam multiplied by Young's modulus for the beam material (E).

AEa - The cross sectional area of the beam (A) at the axis node of the beam multiplied by Young's modulus for the beam material (E)**.**

EIYo - Young's modulus for the beam material (E) multiplied by the moment of inertia of the beam cross section about the local coordinate system Y-axis (Iy) at the origin node.

EIZo - Young's modulus for the beam material (E) multiplied by the moment of inertia of the beam cross section about the local coordinate system Z-axis (Iz) at the origin node.

EIYa - Young's modulus for the beam material (E) multiplied by the moment of inertia of the beam cross section about the local coordinate system Y-axis (Iy) at the axis node.

EIZa - Young's modulus for the beam material (E) multiplied by the moment of inertia of the beam cross section about the local coordinate system Z-axis (Iz) at the axis node.

JGo - The polar moment of inertia of the beam cross section about the neutral axis of the beam (J) at the origin node multiplied by the shear modulus for the beam material (G).

JGa - The polar moment of inertia of the beam cross section about the neutral axis of the beam (J) at the axis node multiplied by the shear modulus for the beam material (G).

AE - The cross sectional area of the beam (A) multiplied by Young's modulus for the beam material (E).

EIY - Young's modulus for the beam material (E) multiplied by the moment of inertia of the beam cross section about the local coordinate system Y-axis (Iy).

EIZ - Young's modulus for the beam material (E) multiplied by the moment of inertia of the beam cross section about the local coordinate system Z-axis (Iz).

JG - The polar moment of inertia of the beam cross section about the neutral axis (J) multiplied by the shear modulus for the beam material (G).

YPINo - The condition indicator for the pin parallel to the Y-Axis at the origin node (0 - pin activated, 1 - for no pin).

ZPINo - The condition indicator for the pin parallel to the Z-Axis at the origin node (0 - pin activated, 1 - for no pin).

YPINa - The condition indicator for the pin parallel to the Y-Axis at the axis node (0 - pin activated, 1 - for no pin).

ZPINa - The condition indicator for the pin parallel to the Z-Axis at the axis node (0 - pin activated, 1 - for no pin).

Beam Maker (Registered version only)

The Beam Maker is a utility used for quickly making beams with a designated number of evenly spaced sections and nodes. The user designates the coordinates of the ends of the beam and the desired number of sections.

The beam is presented much in the same way as a DXF file is imported, that is, only the geometry is provided. The user must edit the file to add element properties and coordinates for the orientation node. Loads (or masses) and boundary conditions must also be added. Then the beam is ready for solution, or for saving as a merging element Merge Form.

DXF file conversion utility (Registered version only)

DXF ASCII files are standard AutoCAD(tm) drawing exchange files. They can be generated in many other drawing packages as well. The finite element model can be drawn with the special tools in those packages and imported to CADRE for analysis using the DXF inport feature. CADRE files can also be converted to DXF ASCII files for export to drawing applications to take advantage of the drawing application's special rendering and printing capabilities.

Conversion to, and from, DXF ASCII files is provided under the ***Convert*** Item on the Main Menu bar.

Importing: Only the geometry is imported. Practically all the 2D and 3D entities that can be constructed in AutoCAD(tm) and many other drawing packages can be interpreted by the converter. This includes Points, Lines, 2DPolylines, Polygons, CIRCLES, ARCS, Meshs, 3DPOLYLINES, 3Dmeshes, 3Dface, PFace, etc. Since most AutoCAD 3D objects (torus, sphere, etc.) are made of 3D polylines and 3D Polymeshes, they also can be converted to CADRE FEM files.

When a DXF file is imported, it is broken down into individual straight line elements, nodes and elements are identified, and then the model is displayed . One extra nodal coordinate is created and is used for the reference for all nodes. It is up to the user to add element properties, boundary conditions, and loads or inertia with the editing utilities in CADRE.

Face entities (3DFACE and PFACE) are interpreted as open frames bounded by elements. CIRCLE and ARC entities are approximated by segmented polygons. The DXF dialog box is presented when a DXF file containing ARC or CIRCLE entities is found. In this case the user can select the segment angle for ARCS and CIRCLES. The size of the segments is specified in degrees of arc. The range of values for the segment angle is from 0.011 to 180 degrees. The default value is 6 degrees.

The DXF conversion utility allows the drawing layers to be selectively imported. The DXF dialog box is presented when a DXF file with multiple layers is encountered. All the drawing layer names as well as the word "ALL" are presented in a dialog box list. The user can select any combination of drawing layers to convert. If the item "All" is selected, then all the layers will be converted irrespective of any other selections on the list.

The DXF conversion utility allows the drawing layers to be selectively imported. When the DXF file represents a drawing with multiple layers, all the drawing layer names as well as the word "ALL" are presented in a dialog box list.   The user can select any combination of drawing layers to convert.   If the item "All" is selected, then all the layers will be converted irrespective of any other selections on the list.

After importing, to minimize the bandwidth and improve solution efficiency, the nodes should reordered with the Re-order nodes feature.

Exporting: Conversion to a DXF file is done in the opposite manner.   Only the geometry is retained and each element is interpreted as a LINE entity.   Any extra reference nodes are interpreted as POINT entities.   Element properties, inertial properties, and reference node assignments are ignored.

Managing files

There are six file types used in CADRE:

The file for the input data	.fem extension
The file for the results or output data	.dta extension
The file for the structural influence coefficients	.sic extension
The file for saving bitmaps of the model	.bmp extension
The file type when printing to a file	.txt extension
The file type for drawing exchange	.dxf extension

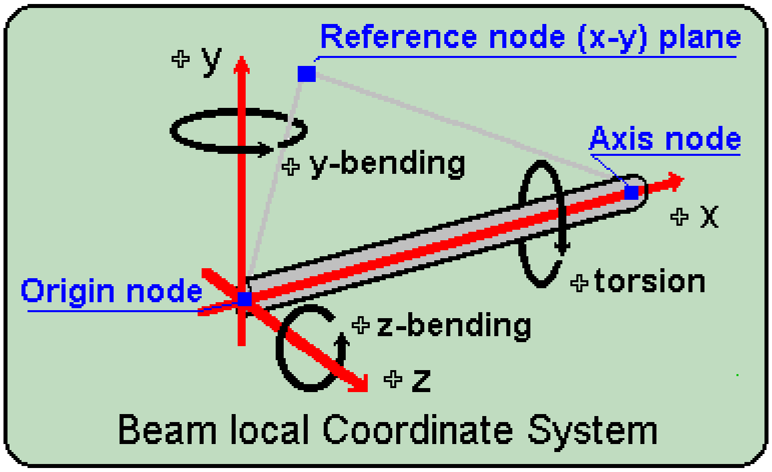
The input file (.fem) contains all the input data for the model. It is the file that is initially loaded when opening a file. This file extension can be set up as an active file extension so that the user can go directly to the file by clicking on it in the Windows File Manager. This option can be selected during initial setup or it can be set in the file manager under the ***Associate*** option under ***File*** on the menu bar in the Windows 3.1 File Manager.

The structural influence coefficient file (.sic) is only used when the operation is set to ***SIC***, or when the operation is set to ***Dynamic*** in order to calculate vibration modes and frequencies. It may need to be recalculated if changes to the structural model are made.

The results data file (.dta) is formed when the ***Save Results*** option is chosen from the CADRE Main Menu. This Main Menu item is enabled only after valid results are calculated or loaded. This file contains displacement data for the *Static* finite element model or mode shape and frequency data for a *Dynamic* model.

CADRE provides the following file management activities:

- Saving the model file
- Opening the model file
- Saving the results
- Loading the results
- Importing-Exporting DXF Files


Reference node (x-y) plane
+ y
+ y-bending
Axis node
+ x
+ torsion
Origin node
+ z-bending
+ z
Beam local Coordinate System

Reference Text: ***The Algebraic Eigenvalue Problem***, J. H. Wilkinson, Oxford University Press, 1988

Error Messages

CADRE displays error messages both during data entry as well as when running the solution.

Messages are displayed at the bottom of the Loads Form, The Bounds Form, The Mass Properties Form and on the Main Menu. Also, some errors (e.g. file errors) are placed in special message boxes that require a response to continue. The following error messages are displayed:

Bounded non-structural node xx

A boundary condition is specified for a node that is only used as a reference point.

Duplicate boundary in node xx

All restraints for a single node must be specified on one and only one page. This error indicates the same node identification number is specified on more than one page.

Duplicate load in node xx

All loads for a single node must be specified on one and only one page. This error indicates that the same node identification number XX is specified on more than one page.

Error in data file

May occur on an attempt to plot the model on the screen. Usually the result of an element connection problem.

Improper node in load condition xx

This indicates that the node referenced for the load condition, XX, is not an identified structural node. Perhaps it is unidentified in the nodal coordinates, or it may be a reference node instead of a structural node.

Improper node in boundary condition xx

This indicates that the node referenced for the boundary condition is not an identified structural node. Perhaps it is unidentified in the nodal coordinates, or it may be a reference node instead of a structural node.

Improper node in element xx

This indicates that a node reference by the element is undefined.

Improper reference node in element xx

This indicates that a reference node is collinear with the element and cannot be used to establish a local coordinate system for the element.

Loaded non-structural node xx

A load condition is specified at a nodal coordinate that is used only as a reference point.

Main model plane Improper

This indicates that the designated nodes in the main model do not prescribe a unique plane. Two or more nodes may have the same coordinates or the 3 nodes may be collinear.

Merging model plane Improper

This indicates that the designated nodes in the merging model do not prescribe a unique plane. Two or more nodes may have the same coordinates or the 3 nodes may be collinear.

Model is unstable

The restraints are not sufficient to resist rigid body motion (of the model or a part of the model) with respect to the applied loads or inertial forces.

Model too large

The model has too many nodes. The maximum is 5461 less the semi-bandwidth.

Out of memory

There is not enough memory to load or solve the model.

No restraints specified

There must be at least one node restrained in at least one degree of freedom for CADRE to run.

Not enough memory

There is not enough memory to load or solve the model.

Not enough structural nodes

At least two structural nodes must be available in order to show the model graphically. If less than two are found, the graphic will not be displayed. Instead this error will be displayed and the user will be shown the view screen for the current data configuration.

Taper error in element xx

A tapered beam cannot have a taper ratio of zero relative to either end.

Undefined element type

The element type is not supported by CADRE.

Unidentified node xx

The node identification number is not in the list of defined nodes.

Zero length in element xx

This indicates that an element is connected to the same node at each end, or that the different node identification numbers refer to the same coordinate location.

Merge Form

The purpose of this form is to enable the entry of the merging criteria for both the main model and the submodel.

Main model orientation plane. Orientation planes are required for both the main model and the submodel. A submodel is connected to the main structure much like any single structural element is attached. It is necessary to specify 3 nodal points in the main structure. Two of these nodes, or points, (A and B) specify where the submodel will be attached and the 3rd point specifies an orientation for the submodel.

Submodel orientation plane. It is also necessary specify 3 nodes in the merging model in order to define a plane for this submodel. This is also accomplished on the Merge Form. Two nodes are selected as the attaching nodes and one is defined as the orientation node. This defines an effective axis and reference node for the submodel so that it is analogous to any other beam element even though is can be a complex 3 dimensional structure.

Selecting orientation planes: The merge form provides list boxes with all the defined nodes for the main model and submodel. The 3 nodes to be selected (A, B, and Ref) are shown above each list box. The colored one indicates the currently active node box (A, B, Ref). Clicking on any number in the list box will place that number in the colored active node box above. When the list box has the focus the following keyboard and mouse actions are available.

Keys	**Action**
Arrow down	Selects the next item down in the list box.
Arrow up	Selects the next item up in the list box.
Page down	Scrolls the list box one page down.
Page Up	Scrolls the list box one page up.
End	Scrolls to the last item in the list box.
Home	Scrolls to the first item in the list box.
Shift Page Up	Changes the active node box up by one.
Shift Page Down	Changes the active node box down by one.
Mouse	**Action**
Left button	Selects the list item and places it in the active box.
Right button	Changes the active node box down by one.

You can also use the mouse to click on the appropriate option button or directly on the node box to make it active.

Attaching and scaling: The merging model is attached to the main model by connecting a designated node A and node B of the merging model to designated and nodes A and B of the main model. The merging model is automatically scaled to fit and oriented to align a designated submodel reference node with a designated reference node in the main model.

If more than two connections are desired this can be accomplished manually by entering the ***edit elements*** option and adding connecting elements. Nodes in the submodel can also be connected to the main model by;

1) Entering the ***edit nodes*** option
2) ***Delete*** the submodel node.
3) At the prompt to reassign any references, enter the node number in the main model.
4) The references will be assigned to the designated main model node and the submodel is then connected at this node.

Attaching to non structural nodes: The sub model can be merged into any specified plane in the main model even if it is prescribed completely with non-structural reference points in the main model coordinate system. Therefore, if the user wishes to merge without connecting the submodel, it is necessary to edit the main model and define some reference points for this purpose before performing the merge. *In this way, CADRE can solve two independent models at the same time*.

Full Merge: The submodel can be loaded along with any boundary conditions and load (or inertia properties). This is accomplished by checking the Full merge checkbox. If this box is not checked, only the nodal coordinates and element properties are merged and other information in the submodel are ignored.

Re-identifying: The planes described above are selected according to the nodal names present in the files as they currently exist. These may be duplicate node names at this point so CADRE must re-identify the new nodes from the submodel when the merging takes place in order to avoid having duplicates node names.

CADRE presents the node number for connection node A of the main model as a default starting number and calculates a small increment. Then CADRE renumbers the new submodel nodes accordingly. However, the user can prescribe how this renumbering will take place by entering values in the re-identifying text boxes.

After a file is merged, the nodes and elements for the entire model can be re-identified under the ***Tools*** item on the Main Menu

It is possible that the newly merged file will exhibit an excessive bandwidth resulting in an "out of memory" error if the model is solved without modification. This bandwidth may often be reduced considerably by re-ordering the nodes under the ***Tools*** item on the Main Menu.

See also Merging file

Merging a model

A model stored in a file can be merged with an open file to create a single new finite element model.   This is useful when a model is being constructed with repeated substructures.

Creating the submodel: The submodel can be created separately with any convenient scale and orientation.   It is no different than any other finite element model file and is created in exactly the same way.   In fact, to fulfill the CADRE rules, it must also have at least one load condition and one boundary condition.

The loads and boundary conditions in the submodel can be retained or discarded, at the user's choice, when merging the submodel with another model.   Any model can be treated as a submodel to any other model (including the model itself).

Merging the submodel: Merging is accomplished by way of the ***Merge*** option under the ***Tools*** item on the Main Menu.   The ***Merge*** item is only enabled when another file is already opened.   Clicking on this item will bring up the Merge File Selection Form where the merging file containing the submodel is selected.   After selection, the user is presented with the Merge Form where orientation, connection, and other merging criteria are specified.

Recovery: Reconstruction of the state of the main model before the merge can be accomplished using the ***Unmerge*** item under ***Tools***.   This item is only enabled just after a merge and before any other editing, solving, or saving is attempted.

See also Merge Form

Re-Identify Nodes

This option is selected under the ***Tools*** item in the main menu. Nodes can be re-identified according to a starting number and an increment. All nodes will be renumbered and all connections and other references will be automatically updated. The same rank order will be maintained so that the bandwidth will not be changed.

See Also: Re-order nodes

Re-ordering nodes

The Bandwidth Manager Form is used to re-order the nodes to minimize the bandwidth.

Re-ordering the nodes is different from re-identifying the nodes. Re-ordering involves revising the rank order of the nodes as well as renaming them with new identifying numbers while re-identifying the nodes just involves renaming them, keeping the same rank order.

The terms, semi-bandwidth and bandwidth are used interchangeably in CADRE. The bandwidth is proportional to the greatest difference between the rank order numbers of any two nodes that are connected by an element. Therefore, re-ordering to re-identify the nodes in a numerical sequence that minimizes this difference will minimize the bandwidth.

Minimizing the bandwidth can greatly reduce the memory requirement for the stiffness matrix so that much larger models can be solved with less RAM.

On entering the form the *Current bandwidth* is shown. Also shown is a blank space where the new bandwidth will be displayed after re-ordering. The ***Re-order*** frame is on the right side of the form.

First, select from the list box the node identification number where you wish to begin the re-ordering. This is the assigned node name in the current model (not the rank order).

Press ***Go*** to re-order the nodes. ***Stop*** is used to cancel the operation, while ***Reset*** is used to return all values on the form to their entry status. The current model is not changed until exiting this form, so be sure to refer to the original model in order to determine a new starting node identification if you wish to re-order from another node. (Hint: Before entering the Bandwidth Manager, display the node numbers with the ***Num*** checkbox so that you can refer to the model).

After pressing the ***OK*** button you will be requested to decide if you really want to re-order and re-identify the nodes. If you press ***OK***, the new node identification numbers will become integers equal to the new rank order.

After exiting the Bandwidth Manager, the nodes will be renumbered. In addition the element identifications will also be re-identified so that the digits to the left of the decimal point relate to the axis node and the digits to the right relate to the axis node.

Undo-Reorder is a menu command under the ***Tools*** item that is activated immediately after returning from the Bandwidth Manager. It provides a last chance to restore the model to the original numbering scheme. This item is only available until the first ***Save***, ***Solve*** or ***Edit*** command is executed.

See also Re-identify nodes