Contents

Scope of CASA SF-3D

This is a brief introduction to structural analysis using CASA. There is inevitably a large amount of information required to define a three dimensional model of a structure. The engineer is required to think in three dimensions, not only for nodal coordinates but also for member orientations, load directions, and global stability of the structure.

CASA SF-3D is designed to help relieve the engineer of some of the effort in handling the structure and analysis through the use of a simple interface and natural abstractions. Please note, however, that it remains the responsibility of the user to provide models that are valid representations of the structure in context and thus both interpret and check the results accordingly. The software cannot replace engineering judgement.

| Capabilities | Limitations |

| Coordinate Systems | Basics of Modelling | Non-linear analysis | Natural modes |

| Footfall dynamics | Mass-cases |

Capabilities

CASA SF-3D has the following analysis capabilities:

- Linear elastic static analysis.
- P-Delta elastic static analysis.
- Natural mode dynamic analysis.
- Models represent:
    - Nodal (joint) coordinates
    - Members spanning between nodes
    - Nodes optionally restrained against movement (supports)
    - Frame members either rigid or pin-ended (moment releases)
    - Nodes and members are placed into groups
    - Linear point loads acting on members and nodes
    - Moment point loads acting on members and nodes
    - Linear distributed loads acting on members
    - Loads are placed into load-cases
    - Gravity-case is automatically generated
    - Sway-cases can be generated from load-cases
    - Load-cases can be factored and combined
    - Mass-cases can be generated from load-cases
- Computes for each load-case and combination:
    - Nodal reactions

- Nodal displacements
- Member deflections
- Member axial and shear forces
- Member torsion and bending moments
- Member bending stress
- Computes for each natural-mode-case:
  - Resonant frequency
  - Relative mode-shape; maximum displacement normalised to 1.0
  - Effective modal mass in each linear axis as a percentage of total mass
- Computes for each footfall-case:
  - Transient response factor and its critical node and frequency
  - Resonant response factor and its critical node and frequency

And makes the following assumptions:

- Members are straight between nodes.
- Member properties are constant throughout each member.
- All members and nodes are adequately restrained.
- Material properties are isotropic (i.e. same for each axis) and linearly elastic.
- The material remains within its elastic limit for all loading conditions.
- Deflections are relatively small.
- Members do not buckle.

CASA SF-3D is based upon the elastic stiffness method using matrix methods and a combination of direct and iterative equation solvers. The user need not be concerned with the technicalities of the mathematics, but they must, as an engineer, be aware of what the mathematics encompasses.

CASA SF-3D must not be regarded as a black box which will always return the 'right' answers for any problem given to it. Structural analysis provides an approximation of the actual behaviour based on an idealised mathematical model. It is therefore only as accurate and as meaningful as the model provided by the engineer, and as such due regard should be given to the assumptions and limitations of the mathematical procedures employed in matrix structural analysis. In addition, while every effort is made to ensure the use of valid algorithms and valid implementations of the mathematics, the software is provided in good faith and it remains the user's responsibility to check the software's output.

COMPARATIVE RESULTS

CASA SF-3D uses standard algorithms for much of the analysis, and as such should yield results comparable to those by other software. Slight variations are expected due to differences in implementation, and such variation is normal amongst all software.

Our P-Delta analysis uses an adaptive algorithm to enhance the model during analysis to obtain better deflections along long members than might otherwise be achieved using only the nodes provided by the model. Refinements to this algorithm might yield slightly different results in future versions.

ACCURACY & PRECISION

Internal calculations are performed using double-precision floating point arithmetic and are accurate to about 15 significant figures. However, if the input data contains a datum accurate to only 4 significant figures then the results which rely on this input cannot be more accurate than 4 significant figures either.

The output from the program is reported to a fixed number of decimal places for clarity of display while keeping a suitable degree of precision. Because of this a certain amount of common sense is required when interpreting the results, so when a reaction of 5.79kN is reported it might only be sensible to take this as 5.8kN if the accuracy and precision of the input data is low.

ILL-CONDITIONING

Ill-conditioning is a situation where the stiffnesses of members around the structure (ignoring releases and supports) vary drastically. The greater the variation in magnitude, the greater the rounding error in the results, and is a consequence of the finite capability of the computer processor. For the vast majority of structures this is not a problem and the rounding errors will not be noticed. However, if unusually large deflections are found without a mechanism being reported then severe ill-conditioning is likely to be to blame and the member properties, member releases, and nodal supports should be checked.

MECHANISMS

Mechanisms arise when a node is unrestrained in one or more of its six degrees of freedom. An example would be a cantilever pinned to the wall instead of being fully fixed.

Usually a mechanism will be detected by the analysis and the user notified. Sometimes mechanisms may not be detected if they do not undergo movement, symptoms of which are very large deflections elsewhere or a model which will sometimes analyse and sometimes will not.
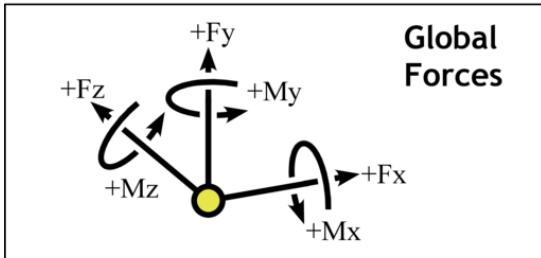
NATURAL-MODE DISCOVERY

The $\boxed{\text{natural mode analysis}}$ can only find modes of oscillation relating to nodes in the structure. That is, in reality a member may vibrate along its length, but if in the model there are no nodes along the member then those modes of vibration will not be found. Splitting critical members into shorter lengths may help to find additional modes. Conversely, omitting nodes along a span may help to remove insignificant modes and reduce analysis time. Note that it is engineering judgement as to what constitutes a significant frequency, mode shape, and mass; the software does not make such decisions.
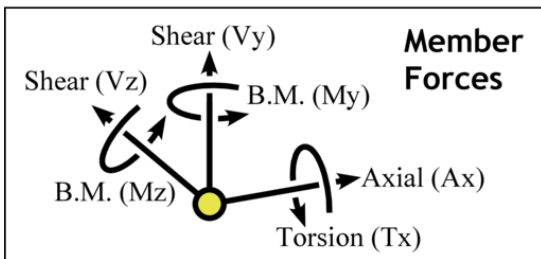
Coordinate Systems

Global Axes

Global Forces
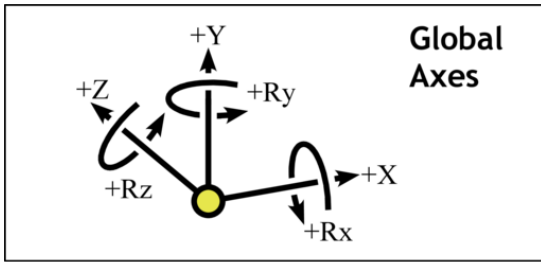
Member Forces

Member Axes

To define the structure and make the results useful, a coordinate system is required. In fact, two are used. Both are left-handed cartesian systems. One is used for the **Global Axes** and **Global Forces**; and the other is used for the **Member Axes** and **Member Forces**.
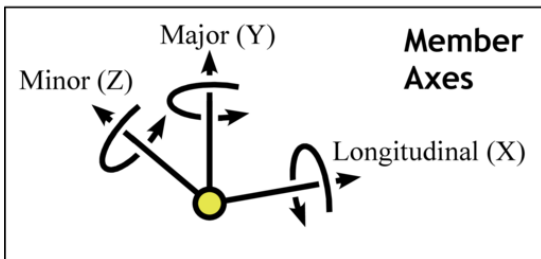
The **Global Axes** are set at an arbitrary but fixed origin and point in arbitrary but fixed directions. Nodes are given as offsets from this origin along the 3 directions, thus the origin can be taken to reside at the coordinates (0, 0, 0). As convention, the X-Y plane is usually taken as the "front" of the structure, consistent with most 2D plane frame analysis programs.

Node coordinates, supports, global loads, nodal displacements, and reactions are specified in the Global Axes.

The **Member Axes** are also referred to as **local coordinates**, as they are local to each individual member. The x-axis is aligned with the length of the member, the y-axis is aligned with the **major axis** of the member, and the z-axis is perpendicular to both x and y. The major axis of the member can be either the strongest or the weakest axis in bending, according to which the user chooses from the section library.

Local loads, pinned ends, member forces, and member deflections are specified in the Member Axes.

INTERNAL MEMBER FORCE SIGN CONVENTION

It has to be noted that within a member the internal forces are necessarily in balance; i.e. there are both +ve and -ve forces in equal magntiude at every point. This means that despite the adoption of the coordinate system stated above, the choice of +ve/-ve for the internal axial/shear/bending forces depends very much upon which way the user wishes to consider them.

For this reason it is difficult to provide a single convention that suits every user in every situation at every point along a member, and so CASA adopts a traditional "textbook" convention which attempts to provide signs and force diagrams similar to that used in hand calculations. Care must be taken when transforming these into other coordinate systems or interworking with other sign conventions.

Basics of Modelling

A structure to be analysed has primarily four types of components:

- **Nodes** define points in space between which span the members.
- **Supports** prevent movement of nodes, giving rise to reaction forces.
- **Members** provide rigidity to the nodes. Their stiffnesses limit the movement of the nodes and transmit forces around the structure.
- **Loads** are applied to the members, against which the reactions act, and giving rise to displacement of the structure.

The final positions of the nodes when all forces throughout the structure are in equilibrium represent the displacements around the structure. From these displacements are derived the member forces, and from these the member deflections and nodal reactions.

Degrees of Freedom and Nodal Restraints

There are six degrees of freedom at each node. All of them must be restrained in some way to ensure a stable structure. Most nodes will be restrained by the members attached to them. However, the structure as a whole also has six degrees of freedom, all of which must be restrained in some way. This is performed by applying supports to nodes.

The linear X, Y, and Z directions must all be explicitly restrained by supports. The rotations Rx, Ry, and Rz need not always be explicitly restrained if the rotational direction is restrained by two linear supports a finite distance apart along a line radial to the axis.

For example, a simply-supported beam spanning from X=0 to X=5 has its Rz axis restrained due to the vertical restraints at each end.

Member End Releases

Members may have pins applied at either end. Member pins release the member ends in rotation about the local Y and Z axes while retaining torsional resistance.

It is important to note that putting pins at the end of every member joined at a node may result in that node being insufficiently restrained, creating a mechanism for that node. Two ways to alleviate this is to either apply a support to the node which is restrained in rotation but free to move linearly, or to leave one of the members unpinned so that the node is fully attached to it.

Second-order static analysis

An iterative technique called "P-Delta" is used to extend the elastic static analysis to take into account second-order effects, in particular the effect of the applied forces ("P") when acting on the displaced ("delta") structure. By way of example, a standard first-order (linear) analysis may indicate that a structure will sway horizontally under a vertical load. A second-order (non-linear, P-Delta) analysis will then perform a second analysis with the loads now acting on the swayed structure, which may lead to an increase in sway and vertical deflection. Analyses are repeated until the structure reaches equilibrium.

Sometimes a structure may collapse due to a mechanism forming within the displaced structure which is not present in the undeformed structure; in such a situation a linear analysis may succeed but the second-order analysis will correctly fail. A common cause of such a mechanism is when a pinned member bends transversely to an applied load.

As an iterative method, it should be noted that for highly flexible or unstable structures convergence may not occur within reasonable time, though for a well-conditioned structure this is unlikely.

Natural mode dynamic analysis

The natural mode frequencies and mode shapes of a structure can be determined. These represent the unforced, undamped resonant oscillations of the structure. Because they are resonant frequencies, the actual displacement is unknown and arbitrary, and so while the mode shape indicates a deflected shape, it is scaled such that the largest displacement (whether linear or rotational) is given the value 1.0 (note that there are no units associated with this).

As well as the frequency and mode shape, the analysis also provides the percentage of structural mass moving in each of the linear axes (the effective modal mass). This is a useful guide as to the relative significance of each mode.

To obtain natural mode results, a mode-case can be created via the loadcase menu in the Loading screen. The mode-case defines the structure's self-weight plus an optional mass-case, along with the range of modes desired. The first mode (mode 1) is the lowest frequency of oscillation, with higher modes increasing in frequency. The solution time is dependent upon the highest mode specified to be found, not by the number of modes in the range requested.

MASS DISTRIBUTION

The self-weight of the structure is represented by distributed masses (sometimes called "consistent mass") and includes rotational inertias. However, independent masses applied as loads in a mass-case are represented by equivalent lumped-masses at the nodes at the ends of the members on which the loads sit and only act linearly and equally along each axis. See Mass-cases for more information.

Footfall dynamic analysis

A footfall analysis computes the transient and resonant response to idealised footsteps on the structure. The method used is in accordance with that described by the SCI and Arup for floors and stairs..

BRIEF DESCRIPTION

It is strongly recommended that the user familiarises themselves with the SCI and/or Arup sources before using this analysis feature. However, a brief explanation is given below.

Note that a footfall analysis does not model the progression of footsteps across a floor or up/down stairs; it models repeated footsteps on each node in isolation, as if one person was hopping on the spot, and determines the response of other nodes. The nodes on which footsteps are applied are called the *excited nodes*, and those for which a response is calculated are called the *response nodes*. Only one node is excited at any one time, and its excitation is considered independent of any other excitation.

For each *response node*, the *excited node* which causes the worst response is called the *critical node*. The response is calculated over a range of excitation frequencies (the rate of hopping), and the frequency at which the *critical node* causes its worst response is called the *critical frequency*.

The response of a node is represented by a *response factor*, as well as by peak and/or RMS velocities or accelerations as appropriate.

The footfall analysis considers only vertical excitation and response. Horizontal response is ignored. Any possibility of sway requires other investigation by the user.

The user has the option of a wide or narrow excitation frequency. For floors, wide considers 1.0~2.8Hz, and narrow considers 1.8~2.2Hz. Stairs are always over 1.2~4.5Hz.

A walking mass for floors can be specified. Typical values are 64.25kg/141.5lb (Arup) or 76kg/167.5lb (SCI). The walking mass on stairs is defined within the method and cannot be altered by the user.

The number of paces to consider for resonance, and the damping factor of the structure, can be specified. These values will vary from structure to structure and require engineering judgement.

Due to the processing intensity of the analysis, it considers only between the first 4 and 8 natural frequencies of the structure (depending on the range of modal frequencies). In some cases this may be insufficient and the response may be underestimated, particularly if the lowest modes have very little vertical participation. If this is suspected then an in-depth analysis on a desktop computer is recommended.

Mass-cases

While a static analysis is only concerned with the stiffness of a structure, a dynamic analysis involves also its mass. The self-weight of a structure is automatically included in CASA's dynamic analyses, but additional masses may exist which do not form part of the structure but which will move with the structure and thus affect its resonance but not its stiffness. For this it is possible to include a mass-case in the dynamic analysis.

A mass-case is derived from a combination of factored loadcases, in which the loads represent weights. These weights are divided by gravity ($9.81 m/s^2$) to obtain the mass. For example, a mass of 1000 kg is represented by a downward load of approximately 9.81 kN. Although the weight is applied only downwards, the mass acts in all directions.

It should be noted that while the structural self-weight is represented as a distributed mass which includes rotational inertia, any masses applied as loads in a mass-case are represented by equivalent lumped masses at the nodes at either end of the member on which they sit and do not include rotational components.

Getting Started

For the purposes of learning the program there are sample model files and "replay tutorials" built-in. *Replay tutorials* are annotated re-enactments of using the app. It is recommended that these be used to explore the use of CASA SF-3D before progressing onto creating a new structure.

If you have Airprint available then this documentation can be printed out.

HOW TO: LOAD SAMPLES & TUTORIALS

1. Go to the tab.
2. Tap on the **Demo** button.
   A list will appear containing samples and tutorials.
3. Scrolling down if necessary, tap on one of the names.
4. Tap on the **Load** button to confirm the action.

In the final step, if you do not tap to confirm within 3 seconds then the button will disappear and the name will need to be tapped again. This is a safety feature to prevent inadvertent loss of any current model data.

BEGIN EXPLORING

After watching some of the tutorials, try loading a sample model and then go to the tab. A view of the structure will be seen and the model can be explored to see what it contains and represents.

The User Interface section explains how to interact with the model.

Tabs	Viewpoints	Selections & Masters	Tips for quick modelling
	Dimensioning	Analysis Results	

Tabs

There are four tabs, shown along the bottom of the screen. These are:

Geometry
For creating and editing nodes, members, supports, and section types.

Loading
For creating and editing loads.

Results
For viewing reactions, deflections, bending moments, and shears.

Utilities
For choosing units system; on-line help; load, save, and clear files; and other functions depending on version.

MODEL VIEWS

The Geometry, Loading, and Results screens all show the same model for different purposes and have much in common.

Each screen has a set of buttons along its top. These are used for performing actions. Buttons containing a small white arrow are menu buttons. Tap on these to open a menu for more actions. Common buttons are:

Undo
Tap to undo the last action.

Clear Selection
Deselects all selected items. It does not delete anything but can cause consolidation of coincident geometry or loadings.

Open Selection Menu
The small white arrow indicates that this is a menu button, not an action button.

Delete Selected Items
Found in the selection menu.
Tap to *delete* the selection. In the Geometry screen it only deletes nodes and members; in the Loading screen it only deletes loads.

Viewpoints

HOW TO: PAN AND ROTATE

In the bottom-right of the screen is a button containing a triad indicating the axes. Tap on this to alternate between pan and rotate.

Put *two* fingers on the middle of the screen and slide them around to pan or rotate.
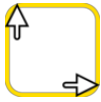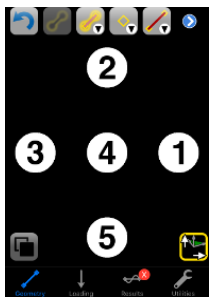
Pan
Two-finger drag will pan.

Rotate
Two-finger drag will rotate.

HOW TO: ZOOM IN AND OUT

Put *two* fingers on the screen and use a pinch gesture. This operates simultaneously with pan and rotate movements.

STANDARD VIEWPOINTS

To quickly move to a standard viewpoint, double-tap with *two* fingers in the following places on the screen.

① Rotate to the X-Y plane (frames).

② Rotate to look down (grids).

③ Rotate to the X-Z plane (side).

④ Fly away to a default distance and angle.

⑤ Fly out to a default distance without rotating.

STORE/RECALL VIEWPOINTS

Tap on this button in the Geometry screen to open the viewpoint selector. A viewpoint memory can be selected by tapping on it. Tapping on ⊕ will store the current viewpoint into the selected memory; tapping ❯ will recall that viewpoint and fly to it.

SOLID RENDERING

Tap on this button in the Geometry screen to open the viewpoint selector. The top-left thumbnail button will toggle the current view between solid and wireframe.

Selections & Masters

The concepts of the *selection*, the *master item*, and the *slave items* are central to the use of CASA SF-3D.

The fundamental method of operation is **noun—verb**; i.e. select items and then perform an action on them.

So to change nodes, members, or loads, you will select those you wish to change and then specify the desired action. Actions are described in the [actions] page.

The final item to be selected is called the *master* item and all other selected items are its *slaves*. The significance of the master item in many cases is to indicate relevant properties or parameters to be used by the action.

For an example see [Moving nodes].

HOW TO: SELECT AN ITEM

To select a node, member, or load, simply rotate or pan the view so you can see it and then tap on it. The final item to be selected will be the *master item*.

Selected items are conspicuously illuminated.

HOW TO: CHANGE THE MASTER ITEM

Tapping on an already-selected item which is not the master will cause it to become the master.

HOW TO: DESELECT AN ITEM

Tapping on a master item will cause it to be deselected. If the item you wish to deselect is not already the master then tap on it to make it the master and then tap again to deselect it.

When a master is deselected then the preceding selected item becomes the master.

HOW TO: DESELECT ALL OF THE ITEMS

Also known as *clearing the selection*.

Tap ⚷ at the top of screen. This is commonly invoked after other actions or prior to making a new selection for the next action.

HOW TO: SELECT OR DESELECT OVER AN AREA

A selection box can be displayed, the corners dragged to resize and position the box, and then the contents selected or deselected.

Toggle box
Turns the selection box on or off.

Deselect all items within the box
If there is a master item then only those items of the same kind as the master will be

deselected.

Select all items within the box
If there is a master item then only those items of the same kind as the master will be selected.

The white arrow in the top-left of the buttons indicate that if the button is given a long press then a menu of options appears to refine the action.

See the Selection Actions page for more information.

How to: Delete all selected items

Tap 🗑 within the selection menu. This will erase the selected items from the model.

Tips for quick modelling

The most common feature used when modelling is that of selecting nodes and members. Actions are then performed on the selected items, such as applying supports to nodes, setting member section types, applying loads, etc.

As such, it is good to get into the habit of clearing the selection between actions. Typically this would involve sequences of: select, act, clear; select, act, clear; and so on. This prevents accidentally applying an action to unintended items.

However, some actions automatically adjust the selection in preparation for the next usual action. This automatic adjustment is predictable, and if it is not desired then simply clear the selection and do something else. When it is desired then it can accelerate the modelling quite considerably, and it is worth getting into habits which utilise this behaviour.

Descriptions of how the selection is used and adjusted by actions is given in the [Actions] pages.

Dimensioning

Certain basic measurements across the structure can be obtained from within the geometry screen.

When selecting only nodes they will describe a line or polygon, information about which will appear at the bottom of the screen.

- For 2 or more nodes: the total length of segments from first to last selected node.
- For 3 nodes: the acute angle formed between them around the master selected node.
- For 3 or more nodes: the perimeter of the described polygon.
- For 3 or more nodes in a single plane: the area of the described polygon.

Note that the value of the area is only valid if the polygon is *non-intersecting*. Pale green lines are drawn from node to node, and none of them must cross any other.

Analysis Results

Analysis results can be viewed in the ⌣ tab.

The presented results relate to the choice of loadcase (accessible via ❯ in the top-right of the screen) and which data set is chosen from the menu at the top of the screen:

Reactions
Tap on a restrained node to view its reaction forces.

Static deflections
Tap on a node to view its global (X,Y,Z) displacement.
Tap on a member to add a sliding inspection cursor from which the local (Y,Z) deflections can be read off.

Natural mode shape
Tap on a node to view its unity-normalised displacement vector.
The mode number can be chosen at the top-left of the screen.

Bending moment envelopes
Tap on a member to add a sliding inspection cursor from which the local torsion and bending moments (Tx,My,Mz) can be read off.

Shear force envelopes
Tap on a member to add a sliding inspection cursor from which the local axial and shear forces (Ax,Vy,Vz) can be read off.

Footfall response.
Tap on a response node to display its response data and highlight its associated critical node. Buttons at the top-left of the screen provide a choice of transient (⋏) or resonant (⋀⋁) data.

Note that a subset of the described results may only be available according to the nature of the structure. For example, deflections along the Z-axis are not applicable to 2D frames.

In the case of deflections, bending moments, and shear forces, white dots may appear along the highlighted envelope of selected members. These indicate the "points of interest", such as maxima and discontinuities. Tap on a dot to move the cursor to its position.

Actions

Manipulating Nodes

Manipulating Members

Manipulating Loads

Manipulating Selections

Modelling Actions

Idioms

Node Actions

Create nodes	Delete nodes	Supports & Restraints	Move nodes

Slide along member

Creating Nodes

Procedure

Applies to Geometry screen.

1. Tap on ◇▾ (menu).
2. Tap on ◇⊕.

Changes to selection

Any previous selection is replaced by a selection consisting solely of the new node.

Significance of master items

If nodes are selected then the new node is created coincident with the master node.

If no nodes are selected then the new node is created at (0,0,0).

Deleting Nodes

Procedure

Applies to Geometry screen.

1. Select the nodes to be deleted.
2. Tap on ![menu icon] (menu).
3. Tap on ![delete icon].

Changes to selection

The selection is cleared if deletion is successful.

Significance of master items

No significance.

Notes

This will delete selected nodes and members, but not any selected loads.

If any nodes have unselected members attached then the deletion will fail and be cancelled. An option will appear asking if the necessary members are to be automatically selected so the user can try again.

If any selected members have loads attached, these must first be deleted separately in the Loadings screen.

Supports & Restraints

Procedure

Applies to geometry screen.

1. Select the nodes for which you wish to change the support conditions.
2. Tap on (menu).
3. Tap on .
4. Pick a support from the list which appears.

Changes to selection

The items remain selected.

Significance of master items

No significance.

Notes

Nodes with support conditions are drawn with an outer circle around them.

Moving Nodes

Procedure

Applies to geometry screen.

1. Select the nodes you wish to move.
2. Pick one to be the master node.
3. **Either**: drag the master node with your finger along one of the axes
   Or: double-tap on one of the co-ordinates at the top of the screen to open up a calculator .
4. All of the slave nodes will move by the same relative offset as the movement of the master node.

Changes to selection

The items remain selected.

Significance of master items

The movement or new position specified is applied to the master node.

Notes

All members and loads connected to the nodes will naturally follow the nodes; they do not need to be selected.

When dragging with your finger, the master node can only be dragged along those axes indicated by dotted lines on the screen. If an axis is too close to being in/out the screen then dragging is prohibited along it — rotate the view before dragging if necessary.

If moved nodes coincide with other nodes (i.e. have the same coordinates) then they will be consolidated when the selection is cleared.

Sliding Nodes Along Member

Nodes can be slid along the axis of a member. The procedure described here is the most basic. For a more sophisticated method refer to stretching a member.

Procedure

Applies to geometry screen.

1. Select the member, as the master, which is connected to the node.
2. Drag the end of the member along one of the axes.
3. All of the selected nodes (if any) will move by the same relative offset as the movement of the end of the member.

Changes to selection

The items remain selected.

Significance of master items

The movement is to the end of the master member.

Notes

This is a specialised version of stretching a member.

| Create members | Delete members | Local axis orienting | Section properties |
| End releases | Splitting | Stretching | Polarising |

Creating members

Procedure

Applies to Geometry screen.

1. Select two or more nodes between which the new members will span.
2. Tap on (menu).
3. Tap on .

Changes to selection

Any previous selection is replaced by a selection consisting solely of the new members.

Significance of master items

There is no significance of the master node.

If pre-existing members are selected then the new members are given the same axis and section type as the master member.

Notes

New members span from node to node in a chain linking consecutive nodes in the order in which they were selected.

If more than two nodes are selected then the loop is not closed. That is, no member is created between the last and first nodes.

Deleting members

Procedure

Applies to Geometry screen.

1. Select the members to be deleted.
2. Tap on (menu).
3. Tap on .

Changes to selection

The selection is cleared if deletion is successful.

Significance of master items

No significance.

Notes

This will delete selected members and nodes, but not any selected loads.

If any selected nodes have unselected members attached then the deletion will fail and be cancelled. An option will appear asking if the necessary members are to be automatically selected so the user can try again.

If any selected members have loads attached, these must first be deleted separately in the Loadings screen.

Setting member orientation

Procedure

Applies to geometry screen. When only one orientation is available (in 2D versions) this action is unnecessary as the orientation is automatically set during member creation.

1. Select the members for which you wish to change the orientation.
2. Tap on (menu).
3. Tap on .
4. Pick an axis from the list which appears.

Changes to selection

The items remain selected.

Significance of master items

If a master node is selected then the members may be orientated towards it.

Notes

The orientation of the member is determined by the direction along which its major axis points. This is coplanar with the longitudinal axis (between the end nodes). The minor axis is normal to this plane.

Selected members with a valid axis are drawn with a small tick at their centre point which indicates the direction of the major and minor axes.

If a member does not have a valid axis then it is drawn as a dotted line.

A member must have a valid orientation axis before loads can be applied to it.

Setting section types

Procedure

Applies to geometry screen.

1. Select the members for which you wish to change the section type.
2. Tap on (menu).
3. Tap on .
4. Pick a section from the library which appears.

Changes to selection

The items remain selected.

Significance of master items

No significance.

Notes

If the selected master member has a valid section type then its size is printed along it.

If a member does not have a valid section type then it is drawn as a dotted line.

Changing member pins

Within frames and grids, the end conditions of members can be set to either rigid or pinned. For pure-truss analysis, members are always considered pins and this action is unavailable.

Procedure

Applies to geometry screen.

1. Select the members for which you wish to set the end condition.
2. Select the nodes at the ends of each member at which the change is to occur.
3. Tap on ![menu icon] (menu).
4. Tap on ![icon].
5. Choose Rigid or Pinned from the list.

Changes to selection

The items remain selected.

Significance of master items

No significance.

Notes

Member pins are released in Y and Z but restraint in torsion remains.

Be careful when a selected node is shared between two selected members, as both adjacent end conditions will be toggled.

Splitting members

Procedure

Applies to geometry screen.

1. Select the members you wish to split.
2. Tap on (menu).
3. Tap on .

Changes to selection

The newly created members are added to the selection.

Significance of master items

No significance.

Notes

Members are split in the middle, resulting in two identical members and a node.

Loads applied to the member are split accordingly.

Stretching a member

A member's length can be changed, either by moving one end or both.

Procedure

Applies to geometry screen.

1. Select any nodes which you want to have move in sympathy with the ends of the member. See the notes below.
2. Select the member for which you wish to change the length and ensure it is the master.
3. **Either**: drag one end of the member with your finger along its axis
   Or: double-tap on the displayed member length and enter a new length in the calculator.

Changes to selection

The items remain selected.

Significance of master items

The master member is the one which is stretched.

Notes

If only one of the end nodes of the member is selected then only that end moves. All other selected nodes will move by the same relative amount.

If both or neither of the end nodes of the member are selected then the change in length is divided equally between each end. All other selected nodes will move in such a way that they part away from a dividing plane perpendicular to the member centre point.

Procedure

Applies to geometry screen.

1. Select the members to be polarised.
2. Select the node for the focus.
3. Tap on (menu).
4. Tap on .

Changes to selection

The items remain selected.

Significance of master items

The master node is the focal point away from which member axes are polarised.

Notes

Loads will be unaffected.

Loads

Choosing loadcase	Create member loads	Delete loads	Changing magnitude

Moving position	Setting direction

When tapping on a load to select it, the following rules apply:

- For point loads, tap on the tip where it touches the member.
- For couples, tap on the member at the point around which the couple acts.
- For distributed loads, tap on the tip at either end (not in-between) where it touches the member.

Choosing a loadcase

Procedure

Applies to Loading screen.

1. Tap on the disclosure button in the top-right of the screen.
2. Choose a loadcase from the list that appears.

Changes to selection

The items remain selected.

Significance of master items

No significance.

Notes

The chosen loadcase is displayed beside the disclosure button.

Any new loads will exist within the loadcase chosen by this method, regardless of how they are created.

Creating Member Loads

Procedure

Applies to Loading screen.

1. Choose the loadcase for the new loads.
2. Select the members on which new loads are to be applied.
3. Tap on ↓ (menu).
4. Tap on ↓, ↻, or ⬩, for point, couple, or distributed load respectively.

Changes to selection

Any previous selection is replaced by a selection consisting solely of the new loads.

Significance of master items

If a load is selected then the new loads are copies of the master load, if the master load is the same type (point, couple, distributed) as the new loads.

If no master load is selected then the new loads are created as 1kN downwards.

Notes

For a nodal point load or couple, apply it to the member and drag it fully to the relevant end.

Members must have valid orientation axes before loads can be applied to them.

Creating Brick Loads

Procedure

Applies to Loading screen.

1. Choose the loadcase for the new loads.
2. Select the bricks on which new loads are to be applied.
3. Tap on ⬇ (menu).
4. Tap on 🟨 or 🟥 for face pressure or volumetric load respectively.

Changes to selection

Any previous selection is replaced by a selection consisting solely of the new loads.

Significance of master items

If a load is selected then the new loads are copies of the master load, if the master load is the same type (pressure or volumetric) as the new loads.

If no master load is selected then the new loads are created as 1kN/m² or 1kN/m³ downwards.

Notes

Pressures act uniformly across the face.

Volumetric loads act uniformly throughout the brick.

Deleting Loads

Applies to Loading screen.

1. Select the loads to be deleted.
2. Tap on ![menu icon] (menu).
3. Tap on ![delete icon] .

Changes to selection

The selection is cleared if deletion is successful.

Significance of master items

No significance.

Notes

This will delete selected loads, but not any selected nodes or members.

Changing a load's magnitude

Applies to Loading screen.

1. Tap on the load you wish to adjust.
2. **Either:** drag the blue bars up and down to set the magnitude
   Or: double-tap on a blue number to open a calculator for numerical input.

Changes to selection

The items remain selected.

Significance of master items

The change is made according to the master load.

All selected loads of a similar type to the master load are adjusted in sympathy.

Notes

To invert the load, drag the blue bar to the other side of the member, or specify a -ve number in the calculator.

To cater for both large and small loads, the magnitude is drawn to a non-linear scale.

Procedure

Applies to member loads on Loading screen.

1. Tap on the load you wish to adjust.
2. **Either:** drag the green slider along the member to sets its distance from either end
   Or: double-tap on a green number to open a calculator for numerical input.

Changes to selection

The items remain selected.

Significance of master items

The change is made according to the master load.

All selected loads of a similar type to the master load are adjusted in sympathy.

Notes

A point load or couple applied to the very end of a pinned member will act on the nodal-side of the pin.

Setting a load's direction

Applies to Loading screen.

1. Tap on the load(s) you wish to adjust.
2. Tap on (menu).
3. Tap on .
4. Choose a direction from the list that appears.

Changes to selection

The items remain selected.

Significance of master items

Only loads of the same kind (point, moment, distributed) as the master load are affected.

Notes

All selected loads *of the same kind as the master load* are put in the chosen direction.

To reverse a load, negate its magnitude instead.

Clear	Auto-select references	Selecting within box

Clearing the selection

Clearing the selection is the act of deselecting everything that is currently selected.

Procedure

Applies to Geometry and Loading screens.

1. Tap on .

Changes to selection

Any previous selection is cleared.

Significance of master items

No significance.

Notes

Consolidation is performed when the selection is cleared.

Clearing a selection does not delete any part of the model, except what may be removed by consolidation.

Auto-Selecting references

This is the automatic selection of items referred to by the currently selected items.

Procedure

Applies to Geometry and Loading screens.

1. Tap on (menu).
2. Tap on .

Changes to selection

Items are selected if they are referred to by preselected items.

Significance of master items

No significance.

Notes

When applied in the Geometry screen it will select all of the nodes relating to the selected members, including those used for orienting any major axes.

When applied in the Loading screen it will select all of the members upon which selected loads lie.

Selecting/Deselecting over an area

This is the selection or deselection of items visible within a selection box.

Procedure

Applies to Geometry and Loading screens.

1. Tap on (menu).
2. Tap on .
3. Position the box by dragging the corners so that it intersects the items to select.
4. Tap on (menu).
5. Tap on (select) or (deselect).
6. To remove the selection box, perform the first two steps again.

Changes to selection

Enclosed and intercepted items may be subject to selection/deselection.

Significance of master items

If the master item is a node then nodes will be selected/deselected.

If the master item is a member then members will be selected/deselected.

If the master item is a load then loads will be selected/deselected.

If nothing is already selected then all enclosed or intercepted objects will be selected/deselected.

Notes

When applied in the Geometry screen it can select/deselect nodes and members.

When applied in the Loading screen it can select/deselect members and loads.

Modelling

Deleting	Copying	Rotating	Reflecting

Deleting selected items

Procedure

Applies to Geometry and Loading screens.

1. Select the nodes, members, and/or loads to be deleted.
2. Tap on (menu).
3. Tap on .

Changes to selection

The selection is cleared if deletion is successful.

Significance of master items

No significance.

Notes

This will delete selected nodes and members only if in the Geometry screen, and selected loads only if in the Loading screen.

If any nodes have unselected members attached then the deletion will fail and be cancelled. An option will appear asking if the necessary members are to be automatically selected so the user can try again.

If any selected members have loads attached, these must first be deleted in the Loadings screen.

Copying substructures

Applies to Geometry screen.

1. Select the nodes and/or members of the substructure to be copied.
2. Tap on (menu).
3. Tap on .
4. Enter the number of copies into the calculator.
5. Tap on one of the new nodes to make it the master.
6. Move the master node to suit.

Changes to selection

Any previous selection is replaced by a selection consisting solely of the newly created copy.

Significance of master items

No significance.

Notes

The copy function uses the selection only as the basis of the copy. It will automatically include any unselected nodes at the ends of selected members as well as any loads on the selected members. Thus, selecting only the members is sufficient to copy a substructure and all of its loads.

Any unselected major axis nodes are *not* copied. If they are to be copied then use auto-select before copying.

If more than one copy is created then when moving nodes the Nth copy is moved N times further than the first copy. This allows regularly spaced multiple copies to be created in a single step.

After copying, if the copied loads are not desired then go to the loading window and delete them. This does not require deselection of the nodes and members.

Rotating substructures

Procedure

Applies to Geometry screen.

1. Select the nodes to be rotated and a master member about which to rotate.
2. Tap on (menu).
3. Tap on .
4. Enter an angle in degrees in the calculator.

Changes to selection

The items remain selected.

Significance of master items

The selected nodes are rotated around the longitudinal axis of the master member.

Notes

A +ve angle will rotate anticlockwise around the member. To rotate clockwise, use a -ve angle.

Rotations apply only to nodal coordinates, not to any local axes of members or loads.

Only those nodes actually selected will be moved. Selected members whose end nodes are not selected will not move.

Reflecting substructures

Procedure

Applies to Geometry screen.

1. Select the nodes to be reflected and a master member across which to reflect.
2. Tap on (menu).
3. Tap on .

Changes to selection

The items remain selected.

Significance of master items

The selected nodes are reflected in the major plane of the master member (the plane formed by the longitudinal and major axes).

Notes

Reflections apply only to nodal coordinates, not to any local axes of members or loads.

Only those nodes actually selected will be moved. Selected members whose end nodes are not selected will not move.

To make a mirror copy, first select the nodes, then copy them, then select the master member, and then reflect.

Idioms

In CASA, an idiom is a sequence of operations to achieve a particular effect which is not provided as a special function. The speed and ease with which idioms can be performed allows the design of CASA to remain uncluttered.

The descriptions in the idioms assume a knowledge of how to perform basic actions .

Idioms frequently make use of *sacrificial* nodes and members. These refer to nodes or members created solely for the purpose of carrying out the operation, and can be deleted afterwards. Of course, if a suitable node or member already exists then that may be used instead.

The list below is not exhaustive, but gives an indication of what is possible.

| Bisect an angle | Trisect an angle | Slide substructure | Simply-supported beam |

| Distribute a load along members |

Bisecting an angle

Background

Consider a V-shaped substructure which is to be bisected with another member.

Method

1. Create a sacrificial member spanning the tips of the V
2. Split the member in half
3. Delete the two resulting members.

The new node will thus lie on the line bisecting the other two members.

Notes

If the new node is to be at a different distance (radially) from the base of the V, then create a sacrificial member from the base to the new node and change its length .

Trisecting an angle

Method

This is very similar to bisecting, but after splitting the sacrifical member (step 2), select only one of the resulting members and multiply its span by $^2/_3$.

Sliding a substructure along an arbitrary vector

Background

Usually a substructure is to be moved along the global X, Y, or Z axes, and that is straightforward. However, sometimes it is desired for a structure to move a specific distance along a non-obvious vector.

Method

1. Set up a sacrifical member with the desired vector
2. Select all of the nodes of the substructure to be moved.
3. Select one end only of the sacrificial member.
4. Select the sacrificial members.
5. Change the length of the sacrificial member by adding the distance to move.

Why this works

When a member length is changed, the end with the selected node is moved and the other end remains fixed. All selected nodes will move in sympathy with the moving end.

Simply-supported beams at arbitrary angles in 3D

Background

In the 2D CASA Apps, simply supported beams can be easily created at any angle with fully pinned supports at their ends. In 3D, however, such supports do not restrain against torsion and may cause the analysis to fail, and because support restraints are in global axes, it is not always possible to adequately add restraint in the supports.

Method

Use fully fixed supports and apply pins to each end of the member.

Why this works

Member pins act in the member's local axes and are restrained in torsion.

Applying a single member load along a chain of members

Background

Often what in reality may be a single beam may be modelled as a chain of shorter members, such as a floor joist spanning multiple supports. If a trapezoidal load is to be applied along the entire length then either a load has to be manually placed and its end magnitudes calculated for each segment, or this idiom can be applied.

Method

1. Begin with a clear selection.
2. Create a sacrifical member spanning the full length of the chain of members.
3. Set the axis so that the member can be loaded.
4. Apply the desired load along its length.
5. Select all of the nodes along the chain of members.
6. Clear the selection.

The outcome is that the load is automatically split and applied correctly to each of the individual members along the span and the sacrificial member is automatically removed.

Why this works

This idiom relies on the consolidation rule that deselecting a node which coincides with a member will split that member, followed by any selected coincident members (the sacrificial one) being eliminated.

Although CASA SF-3D provides graphical interaction through tap-and-drag, there are inevitably times when numeric input is required. A calculator is provided for this.

How to: Open a calculator

In general, whenever a user-defined numeric value is presented in the graphical views it can be double-tapped to open up a calculator.

The calculator is designed with a large cut-out so that interaction with the model can continue during a calculation.

Examples of editable numbers are:

- Geometry, Nodal coordinates:
  Those of the master (selected) node are displayed in white at the top of the screen.
- Loading, Load magnitudes:
  Those of the master load are displayed in blue.
- Loading, Load position:
  Those of the master load are displayed in green.
- Results, Read-off position:
  That of the slider is displayed in green.

Using a calculator

The calculator operates in the same fashion as a traditional pocket calculator.

When it first opens, the calculator contains the current value (i.e. the one tapped on). This value is used as the starting point of any calculation as if it had been typed in.

All trigonometrical functions operate in degrees.

Cancel
Closes the calculator *without* changing the model. This action must be confirmed by tapping the button twice.

Delete
Deletes the right-most digit when entering a value.

Clear / All Clear
When this button displays C then pressing it will clear the current value to zero and the button will change to AC. When it displays AC then pressing it will reset the entire calculation.

Evaluate
Evaluates the result so far.

Enter
Applies the final result to the model and closes the calculator.

Negation

± Changing the sign of the current value.

Functions

Opens or closes the advanced function panel.

Files & Storage

| Loading | Saving | Deleting a file | Export by E-mail |

| Import from E-mail |

All file handling occurs within the document browser, which can be accessed by going to the Utilities tab and tapping on **Documents** at the top-left of the screen.

As a deviation from standard iOS practice, documents are not edited directly. Instead, they are opened as "sessions", to which changes can be made. Only one session may be open at any one time, and must be closed or discarded before another can be opened. Changes made to a session are automatically preserved if the app is closed and reopened. However, changes are not saved back to the original document file until the user requests it through the session manager within the document browser.

Loading a model

Previously saved models can be reloaded at a later dat by simply tapping on them in the document browser.

If there is currently an active session then a message to this effect will be given and the file not opened. The current session must be closed or discarded before another file can be opened.

CLOSING A SESSION

To close the current session:

1. Tap on the **Sessions**.
2. Tap on **Close**.

If the session contains unsaved changes then you will be alerted and given the choice to either not close the session or to discard the changes.

Saving a model

Models can be saved to this device for later use. It is recommended that current work be saved prior to syncing the device or updating the app.

From within the document browser:

1. Tap on the **Sessions**.
2. Tap on **Save/As**.

A choice is presented whereby the session can be saved back to its currently associated document, or a new file name can be entered and the session saved to that.

NOTES

Filenames can only contain letters, numbers, spaces, and the following characters: + - , . @ _ ( )

Deleting a model

Previously saved models can be erased permanently from the device via the document browser.

WARNING: THIS OPERATION MAY BE IRREVERSIBLE.

1. Tap on the Utilities tab at the bottom of the screen.
2. Tap on the **Documents** button to go to the document browser.
3. Find the file you wish to delete and long-press on it.
4. A button marked Delete will appear. Tap on this to delete the file.

If you delete the file associated with the currently active session then it will reappear if you save that session.

Exporting a model via E-mail

Models may be exported via e-mail, either for backup or to transfer to another device with a similar version of CASA installed upon it.

1. Tap on the Utilities tab at the bottom of the screen.
2. Tap on the **Export** button.
3. An e-mail composer will appear with the model embedded within it. Enter an e-mail address and tap on **Send** to send it.

NOTES

The e-mail contains the model in two forms: one is an embedded URL and the other is an attachment. For tranferring models between two copies of CASA of the same edition we recommend using the URL because it preserves more information beyond the structural model itself. If transferring models between different editions of CASA then use the attachment.

Please bear in mind the inherent differences between the editions when transferring models. For example, transferring a 3D model into a 2D App is not meaningful. Conversely, transferring a 2D model into a 3D App may result in nodes being insufficiently restrained. Please check the model carefully and make appropriate edits before accepting any analysis results.

Importing a model via E-mail

BY URL

If a device containing a suitable version of CASA has within its Mail app an e-mail exported by the means explained above, then:

1. Open the device's standard Mail App and open the relevant e-mail.
2. Follow the instructions in the e-mail and tap where where indicated.
3. If the required version of CASA is correctly installed then it will be run with the model ready for use.

BY ATTACHMENT

If a device containing a suitable version of CASA has within its Mail app an e-mail containing a suitable CASA-X (.casax) file attachment, then:

1. Open the device's standard Mail App and open the relevant e-mail.
2. Make a long tap (press and hold for a second) the attachment icon.
3. From the pop-up list, choose the installed edition of CASA you wish to use.

NOTES

If the model cannot be imported (e.g. is incompatible) then CASA will start with an empty model.

Geometry Groups

Nodes and members can be associated together within groups. This allows regions of a model to be enabled or disabled for analysis (e.g. for annotation or "what-if" tests), as well as being colour-coded when displayed with solid rendering.

The use of groups is optional and nodes and members may be assigned to no group, by which they will always take part in the analysis and be rendered in a default colour.

The group list is accessed in the **Geometry** screen by tapping at the upper-right.

CREATING

A group is created by opening the group list and then tapping on the relevant add button at the bottom of the table.

EDITING

A group is edited by opening the group list and then tapping on the right-most disclosure button beside the group name.

The group can be either included in or excluded from the analysed model.

A colour can be set for the group. Members will be drawn in this colour when viewed with solid rendering.

Tap to delete the group.

DISPLAYING

A group is chosen for display by opening the group list and then tapping on its name.

When new nodes and members are created they are assigned to the group currently on display.

Geometry in groups other than that displayed will be drawn in faint, unless no group is displayed, in which case everything is drawn fully.

SELECTIONS

When a specific group is being displayed the following rules apply:

- Tapping on a node or member will select/deselect it regardless of which group it is in.
- Selecting/deselecting with the selection box will only affect nodes/members that are in the displayed group.

MOVING GEOMETRY TO A GROUP

Tap in the Selection menu to move all selected nodes or members into the currently displayed group.

SUPERGROUPS

While nodes and members can each only be in at most one group, it can be convenient if they can be

distributed amongst multiple groups but still be able to relate them all as a single group. For this, supergroups may be used.

A supergroup is a combination of groups. For example, "Floor 1", "Floor 2", and "Floor 3", could all be combined into a single supergroup called "Floors".

Note that when displaying a supergroup, nodes and members cannot be created or moved into it because they must reside in a standard group and not a supergroup.

Load-cases

Each load applied to members must be put within a base load-case.

When new loads are created they are assigned to the load-case currently on display.

The load-case list is accessed in the **Loading** screen by tapping ⊙ at the upper-right.

CREATING

A load-case is created by opening the load-case list and then tapping on the relevant add button at the bottom of the table.

EDITING

A load-case is edited by opening the load-case list and then tapping on the right-most disclosure button beside the load-case name.

A colour can be set for the load-case. Loads will be drawn in this colour.

Tap 🗑 to delete the load-case.

DISPLAYING

A load-case is chosen for display by opening the load-case list and then tapping on its name.

When new loads are created they are assigned to the load-case currently on display.

Only those loads in the load-cases chosen for display will be drawn, and all other loads will be hidden.

SELECTIONS

When a specific load-case is being displayed the following rules apply:

- Only the loads in the load-case on display can be selected or deselected.
- All selected loads will be displayed, regardless of which load-case they are in.

MOVING LOADS TO A LOAD-CASE

Tap 🔃 in the Selection menu to move all selected loads into the currently displayed load-case.

Combinations

Load-cases can be factored and combined in combinations.

Combinations are accessed in the load-case list in the **Loading** screen by tapping at the upper-right.

CREATING

A combination is created by opening the load-case list and then tapping on the relevant add button at the bottom of the table.

EDITING

A combination is edited by opening the load-case list and then tapping on the right-most disclosure button beside the combination name.

The load factor of each load-case can be set by tapping on the value to the right of each name.

The combination can be analysed for second-order effects by setting the P-Δ option.

Tap to delete the load-case.

DISPLAYING

A combination is chosen for display by opening the load-case list and then tapping on its name.

When a combination is chosen for viewing, then in the **Loading** screen the factored loads are displayed but cannot be selected or edited. If a load needs to be edited then it must be done so with that load's load-case on view.

Section Library

All members must have valid section properties in order to play a part in the analysis. The properties are chosen in the section library.

General use	Standard sections	Property generator

General use

The library consists of a short series of tables, beginning with countries and categories. When one of these is chosen it will move on to a table of standard sizes. When a standard size is chosen it will move on to presenting the dimensions and properties.

At the bottom of the country list is a custom-size generator for common cross-section shapes and the ability to change the material of selected members independently of their cross-sections.

The properties display

The top half of the screen displays a diagram of the cross-section and its dimensions. The lower half lists the relevant properties and the type of material.

The material can be changed by tapping on the blue disclosure button and picking from the menu. If the material name is highlighted in red then this indicates either that no material has been chosen or that it needs to be confirmed or changed before the section can be used. Confirmation is required in the situation where a standard library section has a different material from the members to which it is to be applied, and confirmation is made by tapping on the material name.

There is a **Rot** button and an **Apply** button.

The **Rot** button will rotate the section through 90 degrees and swapping its two axes. This allows either axis to be considered as the "major axis" for the purposes of orienting the member in the model, thus allowing the member to be rotated longitudinally without affecting its local coordinate system.

The **Apply** button will apply the section properties to the currently selected members in the model.

Standard sections

Library parts have a default material but a different one may be used by tapping on the blue disclosure button and selecting from the list of materials presented.

If a member is selected prior to opening the section library and it has a material different to that of the library part then material of the member will be used as the default, regardless of the part's library definition. It can be changed by tapping on the blue disclosure button.

The generator

Near the bottom of the country table is a group (labelled "custom") representing the various cross-sectional shapes available. Tap on one of these to move to the properties display.

The section dimensions can be adjusted numerically by double-tapping on the green numbers to open a calculator.

There is no presumed default material, so this must be chosen by tapping on the blue disclosure button and selecting from the list of materials presented.

When the section is the shape you want and the material has been chosen, then rotate if desired and then apply the properties to the selected members by tapping on the **Apply** button.

Material Library

All members must have valid section properties in order to play a part in the analysis. The properties are chosen in the section library.

Custom material	Applying

Creating materials

Additional materials may be created by the user. Within the **Utilities** tab (not the section library) is a blue disclosure button labelled "Materials". Tap on this to open a table of materials which may be browsed by tapping on them.

Tapping on **Create new material** will add a new material to the bottom of the list. It will appear in red to indicate that its properties are invalid. Tap on this to open a screen in which its name and properties may be edited.

Material properties are assumed to be isotropic (i.e. same for each axis) and linearly elastic. This implies that Poisson's ratio lies between -1.0 and +0.5, and the modulus of rigidity (G) must be greater than or equal to one third of Young's Modulus (E). That is, G >= E/3.

Applying materials

At the bottom of the country table is an entry labelled "Materials". Tapping on this will open a list of available materials. Choosing one of these will apply that material to all selected members without affecting their cross-section properties.

After tapping on a material an **Apply** button will appear. Tap on this to confirm the application of the material. If this is not tapped within 3 seconds then the operation will be cancelled with no change to the model.

Create	Preview	Export by E-mail

Creating a report

- Go to the Utilities (tap on the Utilities tab at the bottom of the screen).
- Tap on **Report**.

The Report Generation screen will appear. There will be at least two panels, one on the left containing a set of items which can be inserted into reports, and one representing the layout of reports. iPad devices will have a third panel in which a preview of the report can be seen; iPhone/iPod devices present previews in another screen on demand.

As a general rule, the contents of the report are based upon the currently selected geometry and loads (in the Geometry and Loading screens), currently displayed loadcase, and the current viewpoint. To create a report containing only a subset of the entire model, select that subset before previewing or exporting the report. To create a report covering the whole model, ensure that the selection has been cleared.

ADDING A NEW REPORT

The **Layouts** panel will initially be empty. To add a report, tap on the round blue add button in the panel title.

Reports are represented by a grey box with a title button. An empty report can be deleted by selecting the dark-grey **Report** button (tap on it) and then tapping on the waste bin which will appear at the bottom of the panel. If the report is not empty then it cannot be deleted and the waste bin will not appear, in which case the report's contents must be deleted first.

ADDING SECTIONS

The left-hand panel, entitled **Items available**, contains buttons representing sections which can be added to a report. To add one to a report, tap on the button to select it (it will turn yellow), then tap inside a report in the Layout panel. The section will be inserted at the point tapped.

Sections in a report can be deleted by tapping on them to select them, and then tapping on the waste bin which will appear at the bottom of the panel.

ADDING LOADCASE SECTIONS

By default, any results sections added to the grey region of the report will be generated only for the loadcase chosen in the Loadings or Results screen. To generate results for a given set of loadcases then in the **items available** panel select a loadcase and add it to the report. A blue box will appear in the report into which sections can be added.

Several loadcases can be added to a single blue box by selecting them and then tapping inside the blue box. When generating the report, the contents of the box will be produced for each loadcase in

turn.

A loadcase can be removed from a section by tapping on its button in the report to select it and then tapping on the waste bin that will appear.

If everything is removed from a loadcase section, including the loadcase buttons, then the loadcase section will be automatically removed from the report.

ADDING GROUP SECTIONS

By default, the reported geometry is based on the selection within the entire model (or all of it if no selection exists). If the geometry contains groups then the report can be restricted to these by picking a named group box from the **items available** and inserting it into the report.

Loadcase and group boxes can be nested inside each other.

Preview

If a report contains sections (i.e. is not empty) then a blue disclosure button will appear at the top of its grey box. Tapping on this will provide a preview of the report. The preview may be scrolled by swiping.

Export via E-mail

Reports containing analysis results can be exported as a PDF file via e-mail.

1. In the Geometry view, select any nodes and members you wish to have presented in the PDF report.
2. Tap on the Utilities tab at the bottom of the screen.
3. Tap on the **Report** button.
4. Create or choose a report and tap on the blue preview button.
5. Tap on the **PDF** button at the top of the screen.
6. Once the PDF file is generated an e-mail composer will appear with the PDF file as an attachment. Enter an e-mail address and tap on **Send** to send it.

Between steps 5 and 6 the PDF button will change to "Cancel" and a progress bar will appear marking off the pages as they are processed. The cancel button may be tapped to stop the PDF generation.

NOTES

If analysis is unsuccessful (a red cross will be on the Results tab), then the report cannot contain any results and the PDF may be empty.

PDF generation may take a little while for large documents.

The e-mail composer window will appear once the PDF generation is complete.

The PDF file may not be correctly displayed in all viewers. Please ensure that the Microsoft Arial font is available on the viewing computer.

Contact

Website: www.intesym.co.uk/mobile.php

E-mail: casa@intesym.co.uk