



Computer Aided Structural Analysis

Introduction

CASA-IV is a 3D structural analysis package aimed at engineers wanting sophisticated modelling and analysis combined with productive and smooth-flowing work.

Since 1990, the CASA family has been at the forefront of both the analysis methods and the graphical user interaction techniques, being a pioneer of techniques such as second-order analysis in general-purpose analysis software and having a fully-fledged multitasking desktop-based point-and-click, drag-and-drop interface, both at a time when most of the world was still using keyboards and many people believed second-order to be beyond the capabilities of mainstream computers. CASA-IV is the latest incarnation of the CASA family, developed anew, bringing its unique style to structural analysis.

Feedback

Intesym welcomes and encourages feedback with comments, criticisms, and ideas with which we can improve the software. We hope that all of our users can feel part of a group collectively aiming to enhance the job of the structural engineer.

For comments, feedback, and queries, please send e-mails to:

casa@intesym.co.uk



CONTENTS

Introduction	1	Groups	25
Feedback	1	Loadcases	26
Versions and features	3	Swaycases	26
Reader & Demonstrator	3	Mass-cases	28
Important notice	3	Natural modes	28
Basic principles	4	Combinations	29
General project workflow	4	Cameras	30
Modelling	4	Images	30
Coordinate system	5	Sheets	30
Loadcases	5	Expressions	32
Groups	6	Styles	32
Naming cf. Numbering	6	CAD (toolbox)	32
Sheets	6	Graphical views	34
Visual effects	6	Selections	36
Parametrics	7	Graphical selection	36
CAD	7	Bulk selection, filtering, & manipulation	36
On-line help	8	Analysis	39
User interface overview	9	Browsing results	40
Main window	9	Statics	40
Graphical interaction	11	Natural modes	41
Graphical editing	12	Printing reports	43
Common themes in editing windows	13	Operation	43
On-line help	13	Example	45
Item names & numbering	13	Cameras	46
Highlighting	14	Basic principles	46
Submitting data	14	Tips	46
Input locks	14	Advanced Section Generator	47
Parametric expressions	14	Tools and buttons	47
Point-and-click	14	Relationship to Geometer CAD	47
Drag-and-drop	15	Drawing	48
Choosing an item	15	Geometer CAD	51
Item status	15	General operation	51
"Take Me There"	16	Note of 3D/2D effects	52
Context-sensitive menus	16	Planes	52
Editors	18	Appendix A: Analysis considerations	53
Titles	18	Basic principles of static analysis	53
Nodes	18	First order analysis	53
Members	19	Second Order analysis	53
Section types	20	Member stress	54
DOFs	22	Thermal effects	54
Node loads	22	Lack of fit	54
Member loads	23	Swaycase determination	54
Settlements	24	Natural mode analysis	55
Lack of Fit	24	Relative Amplification Factors	56
Thermal effects	24		
Materials	25		

Appendix B: Mouse controls	57	Appendix E: Parametric Reference	67
Interacting	57	General	67
Cameras	60	Constants	70
Flying	60	Object types	70
Appendix C: Example report	61	Object characteristic suffixes	71
Appendix D: CAD reference	62	Appendix F: Miscellaneous hints	75
Click sequence tables	62	Appendix G: Trouble-shooting	76
		Installation	76
		Graphics / Video	76

Versions and features

This manual covers all versions of CASA-IV and so some features may not be present according to your purchased licence.

While effort is made to ensure that this manual is up-to-date, CASA is under continuous development and some visual changes or minor features may not be reflected in the following text or images.

Reader & Demonstrator

The Reader is intended for the engineer to pass on to their clients so they can directly view and interact with the model and print results without being able to modify it.

The Demonstrator is intended for a potential user to see CASA and practice using it. This demonstration version is provided via the Reader installation.

The Reader has a pale blue background and the Demonstrator a pale green background in their main windows, as well giving a message when loaded. This is to help reduce the chances of trying to use them accidentally instead of a full version.

Important notice

CASA 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 valid implementations of the mathematics and the software is provided in good faith, it remains the user’s responsibility to check the software’s output.

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.

Basic principles

General project workflow

When creating a model from scratch, the general method is:

- Set up titles for the project.
- Define the geometry and loadings, adding groups and loadcases as you go.
- Define loading combinations and load factors.
- Analyse.
- Review the results on-screen in tabular and graphical form.
- Optionally add in cameras and use multiple sheets to show different scenes.
- Generate printable reports containing tabular data and graphical scenes.

Templates

It is easy to create template files containing data common to many projects. Simply make a model containing any supports, groups, loadcases, combinations, reports, and so-on as desired, but not necessarily with any geometry or loadings, and save this as a file. The file can then be used as the starting point for future models.

Mouse controls

CASA makes extensive use of the mouse with which the user interacts with the graphical model. A table of mouse controls is given in the appendix.

Auto-analysis

Some editions of CASA have an automatic analysis feature. This provides updated results after each change of the model without needing to request a re-analysis.

Undo

Recent operations can be undone using the **Undo** button on the main window toolbar or in the **Edit** menu.

A large number of edits are recorded, but the oldest do get purged to prevent excessive memory consumption.

Modelling

The engineer will probably be familiar with CASA's modelling concepts. Nodes are defined at points in space, members are attached to the nodes, properties given to each member, supports attached to nodes, and loads applied to nodes and members.

To construct a model CASA provides the means to enter data either through the keyboard, by point-and-clicking at visual examples, and drag-and-dropping examples to move or copy them. There are also geometry generation and copying facilities which offer easy ways to create regular structures such as meshes and spiral staircases.

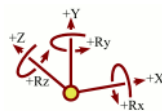
Point loads are applied to the structure at nodes.

Distributed loads are applied to the structure along members.

Nodes can be restrained or sprung in any direction and can have prescribed displacements (settlements) applied.

Coordinate system

The coordinate system is left-handed, so if +X is to the right and +Y is up, then +Z is pointing away. A +ve rotation is always anticlockwise when looking along the +ve axis around which it goes, and so this left-handed coordinate system is a consistent extension of traditional plane-frame models and 2D geometry conventions.



There are two ways of representing positions and vectors in space:

- *Global coordinates* are so-called because they are relative to a fixed origin at (0,0,0) and the direction of each global axis is fixed in space. Node coordinates are usually expressed in global terms.
- *Local coordinates* are so-called because they are relative to another datum and the axes may be rotated relative to the global axes. Member forces are usually expressed in local terms.

Loadcases

Loads and settlements are assigned to loadcases, and these loadcases then form the basis of combinations. A combination represents the summation of factored loadcases, either as a linear superposition or a non-linear summation. For example:

Combination = Sum of factored loadcases

Dead + Live = $1.2 \times \text{Gravity-Y}$ + $1.4 \times \text{Live}$

A linear combination can be changed after an analysis with the results updated immediately to reflect new combinations or factors. A non-linear combination is analysed with second-order effects taken into account but the loadcases and factors cannot be changed after the analysis without requiring a re-analysis.

Groups

Nodes and members can be assigned to groups. A group can be thought of as a means of indicating that items are in some way related, and can represent anything that the engineer wishes. A common use of groups is to indicate substructures. Another is to merely set cosmetic features, such as a colour, to a set of members.

However, a very important aspect of a group is its "usage". A group can be defined as being used for analysis, in which case any nodes and members within it will take part in the analysis. If a node or member is not in any such groups then they will be completely ignored by the analysis, and are typically used for annotation or camera trajectories, and is useful for depicting existing structures or the environs in the model.

Naming cf. Numbering

In conventional analysis packages nodes, members, loads, and so on are all numbered, often forcibly consecutive. CASA takes a different tack and allows everything to have arbitrary names. If the engineer wishes them to be called "1", "2", "3", etc. then he may do so, and that is the default scheme, but the facility is there for items to be meaningfully described through their names, e.g. "South-east base".

Sheets

A "sheet" represents a sheet of paper and contains a graphical scene of the model. When generating a printable report, graphics can be included by simply inserting the sheets.

The purpose of having multiple sheets is to allow a portfolio of scenes to be built up, each with its own choice of contents and viewing positions. Apart from making it easy to reproduce printable reports without having to repeatedly reset scenes, it gives ready-made scenes for clients to view with the Reader.

Each sheet is completely interactive and fully functional. They can be accessed by clicking on their thumbnail images down the side of the main window.

Visual effects

CASA incorporates sophisticated visual effects which are entirely optional from a structural analysis point of view, but which lend themselves to presentation of the model to a client.

Rendering & material textures

Members can be depicted in their intended shape as UB, RHS, CHS, etc., to the correct scale so that an image of the final structure can be seen. Sometimes this is even an aid to creation and editing of the model, but sometimes dense details can make it difficult to see clearly and so this can be turned off and the members depicted as simple lines.

When members are rendered in their full shape they can be decorated with images. Often such an image will be a material texture, making the member look like wood or stone, but it can equally be an advert or a picture of people or trees.

The colours and textures of members are dictated by the settings of the groups to which the members belong. If a member is in a group marked as "annotation" then that group's colour and texture takes priority over any other group to which the member belongs, otherwise the first group into which the member was put will be used.

Cameras

A camera allows either fixed viewpoints to be created or trajectories to be set up. It is easy to recall a camera view, allowing the user to go straight to a specific scene. Alternatively, a trajectory provides an animation allowing the user to be flown around and through the model.

Time base

To improve the visualising of the natural modes of oscillation, each sheet has a time base, the speed of which can be controlled, which when running will animate the oscillations.

Parametrics

Some versions of CASA support parametric expressions. These introduce spreadsheet-like functionality into the model, but instead of a table of cells there are variables and objects. Variables can be defined with formulae relating to other variables and objects within the model. Nodes and loads can be defined in similar terms too. Parametric forms of transreplication can also be applied.

This is a very powerful facility allowing large swathes of the geometry to be adjusted by simply changing the values of variables.

A reference chart of parametric functions is in the appendix.

CAD

Some versions of CASA include associative parametric CAD facilities, ideal for constructing model geometry, drawing details, and annotating the model. The interface for this is kept very simple; it has only half a dozen tools and the type of geometric construction is inferred from the way in which the user clicks on the drawing. For example, choosing the line tool and then clicking on the edges of two circles will create a line tangential to those two circles; clicking on two lines will create a line bisecting them; and so on.


As with the parametric expressions, this is a very powerful tool for describing the structure in geometrical and algebraic terms. For example, if a circle is made bigger then any tangential lines will move accordingly and all of the knock-on effects will be

calculated and the drawing adjusted. If any nodes are associated with the drawing then they too will move.

For convenient mark-up there are ink, dimensioning, and text tools. These can be combined with structural models for general annotation. Nodes can be used as points in CAD constructions, so a dimension can be drawn between nodes and along members.

A table of click sequences and the constructions they give is in the appendix.

On-line help

Context-sensitive help is available throughout the program with tooltip pop-ups and the "What's this?" tool – click on the question mark () in a window's title bar and then click on the item of interest in the window. Clicking on the background of the window will give generic help about the editing dialogue itself.

User interface overview

CASA takes a different approach to some other packages that measure functionality by the number of buttons and clutter in the program. In contrast, it is CASA's way to provide "power through simplicity"; often two or three simple and easy to understand tools can be used together to replace a sprawling toolbox.

Unlike many applications which try to contain everything within a single window, CASA uses many individual windows for different tasks. This gives the user freedom to arrange their work as they wish around the screen and also makes it easier to cope with overlapping windows when multitasking.

Main window

The main window is the hub of CASA and displays the graphical view of the structural model. The rendered view is interactive and is the default appearance, but can be turned off and a traditional stick-view used instead if preferred.

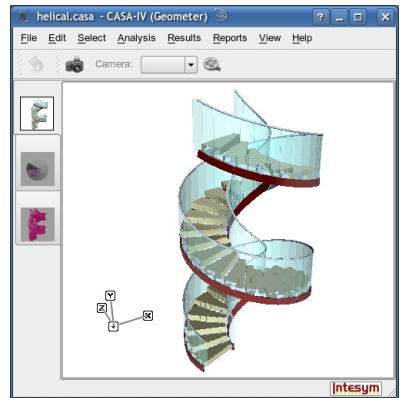
The menu bar along the top contains access to the various tools and facilities of CASA. The menus can be detached and made persistent by clicking on their top-most dotted line when opened.

A small toolbar presents the basic display controls.

Within the graphical view there is a triad at the bottom-left corner to show the directions of the axes.

Note that CASA uses a left-handed system, and so in the view shown here the 'X' axis is pointing to the right, the 'Y' axis is pointing upwards, and the 'Z' axis is pointing away into the screen.

More than one interactive graphical view can exist, shown on sheets selectable by tabs on the left containing representative thumbnail images. The sheets can be reordered by simply dragging the tabs up or down.




Toolbar



Undo the last edit of the model.



Take a "photograph" – creates a new static camera set at the current viewpoint.

 Choose a camera from the menu, and if it is a dynamic camera then it can be played by clicking on the film reel button. If **<none>** is chosen in the menu then the view is free to be moved and rotated.

File menu

Most of the entries in here are standard and behave similarly to other applications and require little explanation.

Merge... allows you to specify another file to have it loaded and merged with the current model. The new parts of the combined model will be automatically selected, ready to be moved to its intended position through the **Select – Manipulate...** window.

Print... and **Export to PDF...** relate to the current graphical sheet only. To print or generate PDFs of reports and numerical data, use the Reports viewer via the Reports menu.

Capture will take a snapshot of the current sheet and copy it to the clipboard. The image can then be directly pasted into other programs supporting that method, e.g. a word processor.

Record camera... which, if a dynamic camera has been chosen on the toolbar, allows the motion of the camera to be captured and saved in a Flash SWF file which can be played back in web browsers.

Edit menu (named "Browse" on Reader Editions)

Each entry here opens a dialogue window through which items can be individually edited or viewed. The list is laid out in a logical order from primitives to items dependent upon them but there is no requirement to follow this order and you are free to use them in any order and revisit them whenever you like. As many dialogues can be open as you wish, including many of the same type.

The last edit can be undone by choosing **Undo** from this menu, which is synonymous with the undo button on the toolbar.

Select

Bulk manipulation of a selected part of the structure, such as copying, is made through the Selection window opened through **Manipulate...** at the top of the menu. The other entries in this menu are simply shortcuts to some of the facilities available in that window and we direct the reader to the chapter on selections for further information.

Analysis

Choose **Analyse...** to open the analysis window in which the analysis can be performed. If an analysis has been previously performed then its log will remain for review, though if the model has been subsequently edited (and thus the results invalidated) the log will be shown faint to indicate this.

Results

Choose **Statics...** to open a results viewer for the static analysis.

Choose **Natural Modes...** to open a results viewer for the dynamic analysis.

As many viewers as you wish can be opened.

Reports

Choose **New Report...** to open a report generator.

View

Display settings... will open up a window with a scrolling set of pages containing checkboxes for indicating which types of items are to be displayed and slider bars for adjusting the drawing scales and render quality settings. Shortcut buttons to each page are shown down the left side.

Even if an item is ticked, what is displayed is subject to the choice of groups, loadcases, and combinations in the corresponding lists.


So for example, to remove all members from the view you should untick the **Members** entry, but to show only the members in group *roof* you should tick the **Members** entry and then in the **Groups** submenu tick *roof* and untick all other groups.

Note that the **Groups** check-boxes are not only on or off – they have an intermediate state in which any nodes and members in that group will be drawn semi-transparent.

Help

About... opens a window showing the version and serial number of your copy of CASA. When contacting technical support please quote the information from this window.

Manual... opens this manual in the computer's default PDF reading program. For many people this requires Adobe Acrobat Reader to be installed on their system, but there are numerous alternative programs that can be used.

Context-sensitive help is available throughout the program with tooltip pop-ups and the "What's this?" tool – click on the question mark () in a window's title bar and then click on the item of interest in the window. Clicking on the background of the window will give generic help about the editing dialogue itself.

Graphical interaction

To rotate, pan, move in and out, and generally fly around the graphical structure view you use the mouse. Drag with the left button to rotate and drag with the right button to pan. Move in and out with the scroll wheel. A complete list of the mouse controls is given in the appendix.

As you move the mouse over the graphical view it will 'snap' to items and they will be highlighted. If you hover over an item then a small box pops up with pertinent information about it. If you are editing an object, e.g. entering a node for one end of a member, and you point and click on a highlighted node in the graphical view then a reference to that node will be assigned to the member.

If you double-click on one of the X, Y, or Z points of the triad then the view will rotate to a front, side, or top projection. If you double-click on the base of the triad then the view will rotate to a default angled position.

If you press a SHIFT key while clicking with the right button onto a highlighted item then the viewpoint will focus on that item.

If you press CTRL while clicking with the left button onto a highlighted item then that item will be selected; with the right button it will be deselected.

If you click with the right-hand button over a highlighted item then a menu will appear and offer a means to open editing windows for the item and any related items. This is a convenient method of beginning an editing operation from a graphical encounter.

For further information on mouse controls please refer to the appendix.

Flying

If you press and hold down the left mouse button and scroll the wheel forward or backward you will begin to fly. Releasing the left button will stop the flight. While flying (left button held down), rotating the scroll wheel will increase or decrease the flight speed and moving the mouse left, right, forward, and back will bank and pitch.

Camera jogging

When playing a camera, the mouse scroll wheel can be used to control the speed and jog forwards and backwards.

Graphical editing

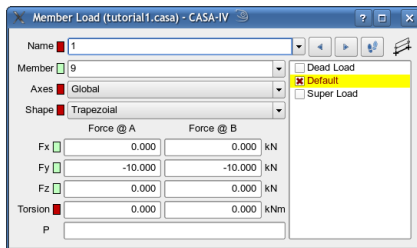
Whenever you are editing items or viewing results, if a relevant item is visible in the graphical view then you can point and click at it and a suitable action will be taken. For example, when editing a member you can specify the end nodes by simply clicking on the nodes in the graphical view.

Some items can be moved, copied, or applied by dragging them from an editing window and dropping them onto other items in the graphical view. For example, to apply a pin to the end of a member, choose a pin in the DOF editing window, drag it across into the graphical view, and drop it onto the end of a member.

Common themes in editing windows


All of the editing windows are designed to work in similar ways to keep a clean and elegant behaviour with predictable operation. CASA departs from many conventional analysis packages in that it uses separate editing windows in which individual items are chosen and edited. This might at first feel odd for users used to working in long tabular lists of nodes and members.

The *input focus* is the place in the window which responds to key presses or which will receive information when you point and click on the graphical view. Most input focuses are editable *fields*, the white boxes into which text and numbers can be typed, and in these a *caret* (I-shaped cursor) will appear.



On-line help

If the mouse is hovered over a field then a pop-up will appear giving more information about what it represents.

Use the **What's This?** feature for more comprehensive on-line help: click on the question mark () in the window's title bar and then click on the item of interest in the window. Clicking on the background of the window will give generic help about the editing dialogue itself.

Item names & numbering

Ordinarily there is no need to enter a name in the **Name** field when creating a new item – just enter data in the other fields and a name or number will be automatically generated.

If the name field is locked (the box next to it is red – see the subsection below on *input locks*), which is the normal state, then whether a name or a number is generated depends upon what the previously viewed item had; if the previous item had a simple number then the new one will be given a number, if the previous item had a name then the new one will be given a name following a sequence implied by the name.

If the field is unlocked and no name or number is entered by the user then the new item will always be given a number.

In the pop-up list of names, any created prior to the editing window being opened will be listed in alphabetical order; any created subsequent to that will be shown in the order in which they were created.

Highlighting

The item being edited in the window is highlighted on the graphical view in every place where it occurs. So for example, when editing a Section Type all of the members of that type are highlighted, and when editing a loadcase every load within that case is highlighted.

Submitting data

If you press TAB or ENTER (RETURN) then any changes will be submitted, the model changed to reflect this, and the focus will move to the next field. If you do so in the last field of a window then the focus will move to the first field and the data will roll over to the next item. Pressing SHIFT + TAB will move the input focus backwards up the window, rolling over to the previous item if you pass the top. Pressing SHIFT + ENTER will submit the changes without moving the input focus. If you have typed changes but not submitted them and wish to cancel them then press ESCAPE.

To undo a change which has been submitted, click on the **Undo** button (↶) which can be found on the main window toolbar or at the bottom of the **Edit** menu of the main window.

Input locks

Some fields have *input locks*. These are the red and green squares, where red = locked and green = unlocked, and can be toggled by clicking on them. An unlocked field is empty when a new item is created, a locked field has its value carried over from the previous item when a new one is created. In addition, locked fields are skipped over when the focus moves; to edit them you must click on the field with the mouse. Locking fields is a very convenient way to enter repetitive data. The fields which are most likely to benefit from being locked are locked by default when the window opens.

Parametric expressions

In some editions of CASA there are some kinds of items, most notably nodes and loads, which can be defined parametrically with reference to variables and values from other items. The relevant formulae can be entered in the fields labelled **P**.

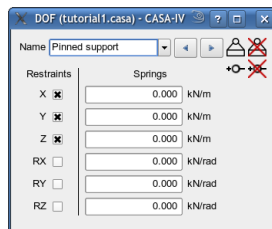
Point-and-click

When a field has the input focus and you click on an item in the graphical structure view then relevant data will be copied into the field. If you click with the left mouse button then the data will be submitted and the focus moved; if you click with SHIFT held down then the data will be submitted but the focus will not move (i.e. similar behaviour to typing in the data and then pressing ENTER or SHIFT+ENTER)..

Drag-and-drop

Some windows have icons in their top right which can be dragged across onto the graphical structure view (or into other input windows where appropriate) with the mouse to move or copy the currently edited item. In the DOF window, for example, drag and drop the support icons onto nodes or the ends of members in the graphical structure view to apply or remove supports, springs, and releases.

If you wish to cancel a drag then press ESCAPE before releasing the mouse button.



Choosing an item

The **Name** field shows the name of the current item being edited. If it is empty then this is a new blank item which will be automatically numbered and created if you start entering data elsewhere in the window, or named and created if you type a new name into this field. A pre-existing item can be chosen by opening the field's menu and picking from that, or the name *<new>* will be found at the end of the list and chosen if a new blank item is desired. Alternative methods are to type in a name or to click on the item in the graphical structure view.

In some versions of CASA, a search facility exists in that if you start to type in a name then a list will appear showing the pre-existing items whose names match what has been typed so far. One of these can be selected using the arrow keys and pressing ENTER or by clicking on it with the mouse.


Note that typing a new name into this field when on a pre-existing item will not directly cause a new item to be created. This is to guard against accidentally creating items through mistyping a name while searching. If a name is entered which does not correspond to any other existing items then the user is asked whether a new item is to be created, whether the current item is to be renamed, or to leave everything as-is.

When the name field has the input focus you can move to the previous or next items with the up/down cursor keys. If the focus is anywhere else then you can move to them with the page-up/page-down keys or by scrolling the mouse wheel over this field.


Item status

In the **Name** field a symbol will sometimes appear next to the name:

- ✗: The item is invalid, meaning that it lacks some necessary data before it is useful. Note that its absence does not mean that the given data is correct, simply that sufficient data has been given.

 : The item is a library part. It cannot be modified, renamed, or deleted because it will not be saved with a user's file, instead being always supplied by its library.

"Take Me There"

The *Take Me There* button () will move the graphical viewpoint across the structure to focus on the item currently being edited.

Context-sensitive menus

Clicking with the right-hand mouse button over an editing window will open a context-sensitive menu. Common operations are:

Where? is similar to *Take Me There* except that it does not move the viewpoint and only draws lines pointing to everything related to the current item. It is used when the one item may appear in a multitude of places, such as node supports.

Insert will rename/renumber the current and subsequent items, incrementing their numerical value by 1, and create a new item with the same name as the current item used to have. For example, choosing **Insert** on a node called "**Base 3**" will cause that node to be renamed to **Base 4** and a new node called **Base 3** to be created. If there were already a node called **Base 4** then that would be renamed to **Base 5**, and so on.

Remove parametric will appear for items which have a parametric expression defined (in their **P** field). If this is chosen then the fixed values of the item will be updated to reflect the combined result of themselves and the parametric expression, and the expression will be removed. It is thus used to convert a parametrically defined item into a conventional non-parametric item.

Delete will delete the current item. If any other items are dependent upon it (e.g. trying to delete a node to which a member is attached) then a warning will appear and the operation cancelled.

Rename allows the current item to be renamed.

Sort all will rearrange the names of all of the items so that they run in alphanumerical sequence according to some relevant geometrical arrangement. Names are only moved to items with similar names but different numbering, not between items with dissimilar names. So, for example, node *Base 4* may be swapped with *Base 2* but not with *Apex 2*.

Renumber all will rename items to give consecutive numbering starting from 1 within each set of items with similar names, but without rearranging their relative positions around the geometry. So, for example, nodes *Base 4*, *Base 2*, and *Apex 3* will become *Base 1*, *Base 2*, and *Apex 1*.

Consolidate all will compare pairs of items to see if they are equivalent, and if so they are merged into a single item. For example, when applied to nodes it will remove duplicate nodes residing at the same location in space with the same restraint.

Use this with care if some nodes are deliberately coincident yet intended to remain separate. In this situation you can prevent them being consolidated by creating null supports (i.e. released in every direction) and applying them to the nodes (each node being given a different support). The use of null supports has an added advantage of giving a visual indication of the presence of coincident nodes.

There is also often a **Selection** submenu offering facilities to manipulate the selection in the context of the currently edited item. Common operations are similar to:

- **Select:** Add the current item to the selection.
- **Deselect:** If the current item is selected then it will be deselected.
- **Apply:** Any items in the selection which can usefully have this item applied to them (e.g. nodes to which a restraint can be applied) will have it applied to them. The current item is not replicated; it is just referred to by the selected items.
- **Unapply:** Any items in the selection which have this item applied to them will have it removed from them. For example, if we are editing a "Pin" restraint then any selected nodes with that restraint will become unrestrained. The current item is not deleted; it is just no longer referred to by the selected items.
- **Copy to selected items:** The current item will be replicated with the copies referring to each of the selected items to which it could usefully refer. For example, copying a node load onto selected nodes.

Editors

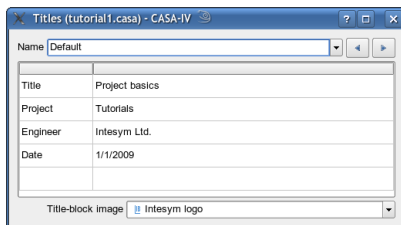
In this section we look at each of the editing windows in detail. Please refer to the previous chapter for details on the common features of editing windows. Only the salient unique features of each window are described here.

Titles

From the main window **Edit menu**, choose **Titles...**

Each row consists of two columns. The first four rows will appear in the headings of printouts, subsequent rows will not and can be used for holding additional information about the project.

If you have loaded a suitable logo into the image repository (**Edit – Images...** in the main window) then it can be chosen from the list at the bottom of the window and will be included in the headings of printouts.

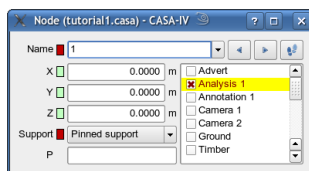


More than one set of titles can be defined and then used by different graphical sheets (**Edit – Sheets...** in the main window) so each graphic has pertinent titles.

Nodes

The **X**, **Y**, and **Z** fields represent the position of the node in global coordinates.

A DOF can be chosen for the **Support**. If left blank, or **<none>** is chosen from the menu, then the node is unrestrained. Any restraints and springs applied to the node occur in and around the global axes.



Note the **P** field which will appear on some editions of CASA. Into this field a formula or vector of three numbers can be entered to specify the coordinates in a parametric form.

Context-sensitive menu

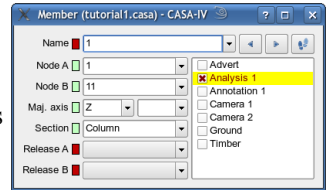
Sort all will rearrange the node names so that they run in alphanumerical sequence according to their spatial location along the axes X, Z, then Y.

Consolidate all will merge any nodes which are within less than 1mm of each other and have the same support conditions. If two nodes being merged each have loads then the loads are preserved and will exist together on the single merged node.

Members

Node A and **Node B** represent the ends of the member. The longitudinal axis and member length is the vector from A to B.

Maj. axis consists of two fields to indicate the major axis of the member. If the first field is set to **X**, **Y**, or **Z** then the second field is unused and the major axis is aligned along the global axis specified. If the first field is set to **Node** then the second field is used to indicate which node the major axis points towards.



Section is the section type that this member is made of.

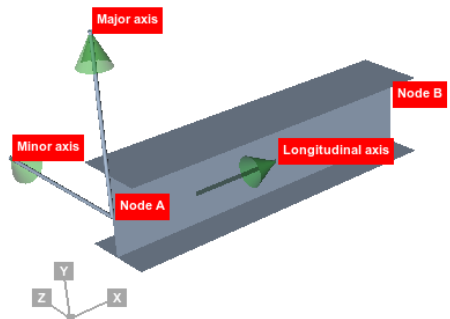
End releases can be applied at the node A or node B end (or both) in the **Release A** and **Release B** fields. Each end can be released independently of the other but the member as a whole must be sufficiently restrained to the rest of the structure.

★ *The major axis must not be colinear with the longitudinal axis because the pair form a plane. Usually the section is defined so as to be strongest in bending in this plane.*

★ *A member has its own local coordinates as shown in the diagram. The origin is at node A. The minor axis is computed automatically.*

When applying releases to a member, the following conditions need to be met:

- Torsion (R_x) must be resisted at least at one end.
- Each shear (X , Y , Z) must be resisted at least at one end, but they do not all need to be at the same end.
- A shear restraint in Y or Z is necessary at the opposite end to a corresponding R_z or R_y moment release.



Moment releases (R_y , R_z) can occur at both ends.

Context-sensitive menu

Split allows the member to be divided into parts, with any applied loads automatically split to ensure that the loading does not change with respect to the overall structure.

Swap ends will swap nodes A and B around, and also adjust any loads applied to the member to ensure that the loading does not change with respect to the overall structure.

Sort all will rearrange the member names so that they run in alphanumerical sequence according to the naming of their nodes A and B.

Consolidate all will merge any members which have the same end nodes, the same end release conditions, the same section type, and the same major axis. If two members being merged each have loads then the loads are preserved and will exist together on the single merged member. If two members being merged are antiparallel then the direction of the single merged member may go in either direction but the loading direction will be preserved with respect to the entire structure.

Section types

The material can be either chosen from the library (see the Materials editing window) or by entering values into the relevant fields. When a library material is chosen the fields cannot be edited here; you should choose *<none>* from the menu first.

The geometrical properties of the section are edited in a similar way as the material – choose from the section library or enter values individually. When choosing from the library the visual shape of the section is automatically set; when entering individual data you can choose a suitable shape.

★ *The visual shape chosen here is only relevant for rendering on the graphical structure view; the analysis only considers the numerical properties.*

★ *It should be noted that the x-x axis is the major axis, parallel to the web or nominal strongest axis, so in the case of a UB the web is vertical and the Ixx represents the highest inertia.*

★ *J is the torsional constant, not Ipp.*

The units are chosen to match those in common British usage. Note that unlike most other data used in CASA which are rounded to a number of decimal places, the section properties are rounded to a number of significant figures. This is because of the enormous range of values between the largest and smallest sections.

Some versions of CASA offer section generators. These are available in the additional tabs in the lower part of window. The generators can compute the geometrical properties given the shape of the cross-section.

If the material properties have been chosen from the material library menu then those fields will become uneditable and faded. To edit them, choose "<none>" from the material library menu.

If the geometrical properties have been chosen either from the section library or computed by one of the automatic generators then those fields will become uneditable and faded. To edit them, choose "<none>" from the section library menu. Note that doing so will remove the section generator's data.

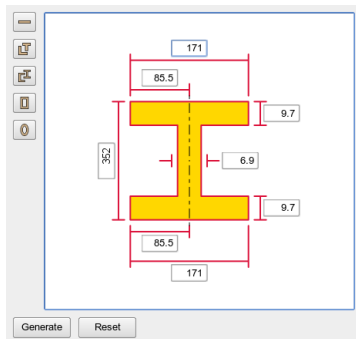
Drag-and-drop

There is a draggable icon in the top-right of the window for applying or removing the currently chosen section type to members in the graphical structure view.

Section generator (Standard)

A variety of common cross-sectional shapes can have their geometrical properties automatically calculated through the section generator tab in this window. Choose a shape by clicking on the buttons and enter the relevant dimensions in the boxes (vertical boxes act as if writing right-handed).

The standard generator uses common textbook formulae for computing the properties and so should be used within the limits of those methods. Sections should not have very high ratios of overall depth/width (except plates), each plate should be thin, and hollow sections should have thin walls relative to their depth and width.



Click on **Generate** to have the properties computed and submitted, or on **Reset** to put the dimensions back to their original values.

If you close the generator by changing tabs, closing the window, or changing to another section type, then the generator will be reset.

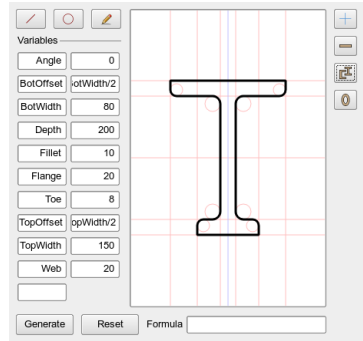
Section generator (Advanced)

The Advanced section generator allows any arbitrary cross-section shape to be graphically created with some simple associative parametric CAD tools.

The Advanced generator uses finite-element analysis and integration methods to compute the properties, and can be used for open, closed, and solid sections, as well as those with hollow cut-outs.

Use of this generator is explained in a later chapter.

Click on **Generate** to have the properties computed and submitted, or on **Reset** to put the drawing back to its original state.

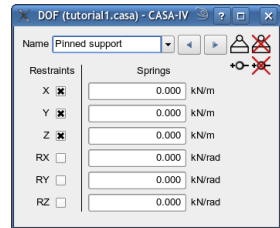


If you close the generator by changing tabs, closing the window, or changing to another section type, then the generator will be reset.

DOFs

DOF is an acronym for Degrees Of Freedom and represents the restraints or releases along and around each axis. When applied to a node it is a support, when applied to a member it is an end release.

Each DOF can also contain a spring in and around each *released* axis of a node. A spring in a restrained axis will be ineffective because of the restraint.



Drag-and-drop

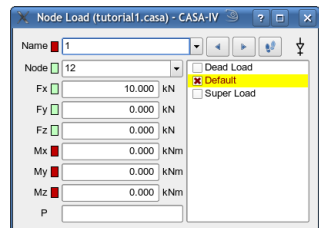
There are four draggable icons in the top-right of the window for applying or removing the currently chosen DOF to nodes or members in the graphical structure view.

Node loads

Node loads are forces applied to nodes. Their directions are along and around the global axes.

Drag-and-drop

There is a draggable icon in the top-right of the window for copying the current load onto other nodes in the graphical structure view.



Member loads

Member loads are forces applied to members. They are distributed along the whole member from the member's node A to node B. The magnitude at node A can differ from that at node B, with the force varying linearly along its length.

The direction of the linear forces (**F_x**, **F_y**, **F_z**) can be either in the global axes (default) or the local axes of the member, according to the **Axes** field.

The torsional force (**M_x**) is *always* in the local axes and a +ve torsion acts anticlockwise around the member looking along from A to B.

The **Shape** field is a convenience facility for providing a quicker entry if the forces at A or B can be inferred automatically. The choices here are:

- **Trapezoidal**: Independent force magnitudes at A and B.
- **UDL**: Enter a force at A. The force at B will be made the same.
- **Triangle (A to 0)**: Enter a force at A. The force at B will be set to 0.
- **Triangle (0 to B)**: Enter a force at B. The force at A will be set to 0.

Changing this field after a load has been defined may cause the forces at the inferred end to be changed. For example, changing a non-uniform load to a UDL will cause the force at B to be made the same as that at A.

★ *The numerical values apply per metre run, so the total load depends on the length of the member.*

Drag-and-drop

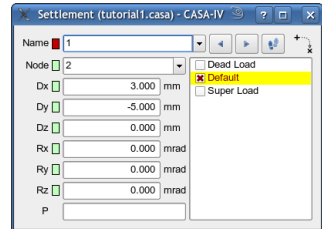
There is a draggable icon in the top-right of the window for copying the currently load onto other members in the graphical structure view.

Settlements

Settlements are prescribed displacements applied to restrained axes on supported nodes. Their directions are along and around the global axes.

Drag-and-drop

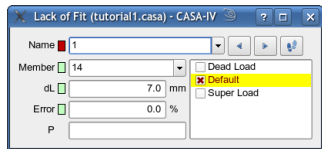
There is a draggable icon in the top-right of the window for copying the current settlement onto other nodes in the graphical structure view.



Lack of Fit

Lack of Fit describes the fabrication error of a member.

There are two ways of specifying a the error, one is as the difference in length (dL) and the other is as a percentage of the intended length. If both are specified then both are applied, added together. Positive errors represent a member fabricated longer than it should; negative errors represent a member being shorter than it should.



Note that lack of fits are placed into loadcases and so only take effect in combinations that include them, they are not considered part of the general geometry.

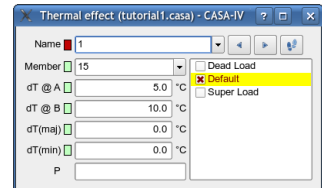
Thermal effects

Members can have thermal effects (heating) applied to them. Temperature differentials along all three local axes can be specified – from end to end, top to bottom, and front to back.

dT @ A and **dT @ B** are temperatures at each end of the member relative to the ambient temperature around the whole structure. This gradient can be considered as applying linearly along the axis of the member where at each point it is the mean temperature of the section around which the **dT(maj)** and **dT(min)** are centred.

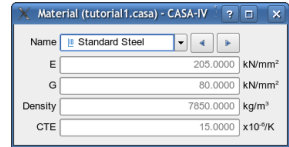
dT(maj) is the temperature differential between the top of the section and the bottom of the section (i.e. major axis). A +ve value means that the top is hotter than the bottom.

dT(min) is the temperature differential between the back of the section and the front of the section (i.e. minor axis). A +ve value means that the back is hotter than the front.



Materials

A material can be set up for easy reuse. These properties are unaffected by any geometry of the model. **E** is Young's Modulus, **G** is the modulus of rigidity, **Alpha** is the coefficient of thermal expansivity, and **P_y** is the yield stress. The units are chosen to match those in common British usage.



Material (tutorial1.casa) - CASA-IV

Name: Standard Steel

E: 205.0000 kN/mm²

G: 80.0000 kN/mm²

Density: 7850.0000 kg/m³

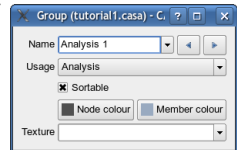
CTE: 15.0000 x10⁻⁶/K

Groups

Groups are used to contain nodes and members so that regions of the geometry can be more easily handled en masse. For example, an entire group can be selected in one operation.

The **usage** can take one of several values:

- **Analysis**: Any nodes and members within this group will be considered as structural components for the analysis.
- **Annotation**: Any nodes and members within this group are purely cosmetic and will be ignored for analysis (as will any loads applied to them).
- **Camera**: Equivalent to **Annotation**, but signifies the intent of any nodes within the group to be used for camera trajectories.



Group (tutorial1.casa) - C.

Name: Analysis 1

Usage: Analysis

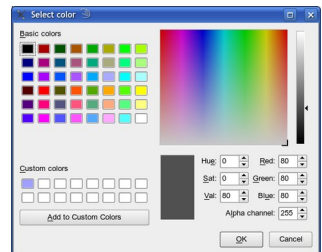
☒ Sortable

☐ Node colour ☐ Member colour

Texture: [dropdown]

A **sortable** option indicates whether nodes and members within the group are to be sorted or their positions preserved when the user requests an automated sort. If a node or member belongs to multiple groups then it will only be sorted if *all* of its groups are sortable.

The two buttons will open up colour selectors so that nodes and members can be drawn in different colours according to which groups they belong to.



Select color

Basic colors: [grid of 24 color swatches]

Custom colors: [grid of 12 empty color swatches]

Add to Custom Colors

Hue: 0 Red: 60 Sat: 0 Green: 60 Val: 60 Blue: 60 Alpha channel: 255

OK Cancel

The colour selector allows colours to be selected either from a set of basic colours, a set of user-defined custom colours, or any thing in terms of HSV or RGB values.

The **alpha channel** value represent translucency. A value of 255 means that the colour is completely opaque and a value of 0 means that the colour is completely transparent (invisible).

An example of using translucency is to simulate the appearance of glass – use a pale green colour with an alpha value of about 100~150.

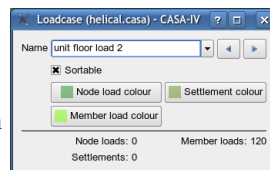
Loadcases

Loadcases contain node loads (point loads), member loads (distributed loads), and settlements (prescribed nodal displacements).

They are applied to the structure unfactored, with the intention of a factor being provided in the combinations. It is a popular choice to create loadcases with normalised loads (i.e. unit loads) and then specify a scaling factor in the combination, giving a clearer comparison between deflections and forces in the individual loadcase results and the applied loading.

A **sortable** option indicates whether loads within the case are to be sorted, analogous to that for groups.

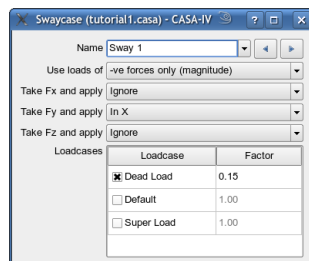
Each loadcase has individual control over the colour of each kind of load. Please refer to the description of the Groups window for more information on the colour selector.



Swaycases

Swaycases are a means by which all of the loads in a loadcase can be taken and a percentage of them applied in a different direction.

The Loadcases table at the bottom of the window is used to choose which loadcases are used as the basis of the sway and by what factor those loads should be multiplied. The direction of the sway can be reversed by applying a negative factor.



★ *If more than one loadcase is chosen then their resultants are independently factored and swayed and then the sways summed to provide the final swaycase.*

★ *Only linear forces are considered. Moments and settlements are ignored.*

★ *In the case of a second-order analysis, the sway is considered to be applied globally with the global components computed according to the original geometry and they do not change as the structure deflects.*

A swaycase can **Use loads of**:

- **All forces (signed):** -ve loads → -ve sway; +ve loads → +ve sway.
- **All forces (magnitude):** All loads → +ve sway regardless of their sign.
- **-ve forces only (signed):** -ve loads → -ve sway; +ve loads ignored.

- **-ve forces only (magnitude):** -ve loads \rightarrow +ve sway; +ve loads ignored.
- **+ve forces only:** +ve loads \rightarrow +ve sway; -ve loads ignored.

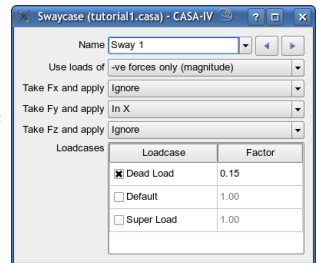
Each of the global Fx, Fy, and Fz components can be rotated, factored, and applied in the following global axes:

- **Ignore:** The component is ignored.
- **In +X:** The component is rotated and applied along the +X axis.
- **In +Y:** The component is rotated and applied along the +Y axis.
- **In +Z:** The component is rotated and applied along the +Z axis.

Example

Consider the screenshot to the right. The sway forces are computed in the following manner:

1. All of the -ve applied forces are extracted from the "Dead Load" loadcase. These include any -ve components of angled loads.
2. Only their magnitudes of these extracted forces are considered so all are considered "+ve".
3. These extracted forces are then multiplied by 0.15.
4. Those which act in the global Y direction are applied in the X direction; the others are ignored.
5. If more than one loadcase is set then they are accumulated in turn by repetition from step 1.



Thus the downward (-Y) Dead Load forces will be applied as a sway along +X.

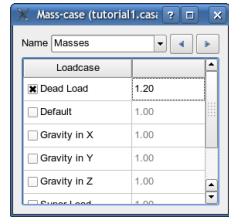
Note that the extraction of forces in step 1 is based on the applied forces and not subject to cancellation between opposing +ve and -ve forces.

Mass-cases

Mass-cases are used with natural mode (dynamic) analysis to determine the effects of mass on the vibration of the structure.

The **self-mass** of the structure is automatically taken into account in a dynamic analysis and should not be included as a mass-case.

One or more additional loadcases can be chosen, with each factored and then summed. The resultant forces are then interpreted as masses lumped at the nodes, applied *in addition to* the self-mass of the structure.



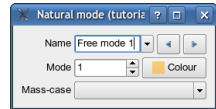
★ *Additional loading: only linear forces are considered as weight. Moment forces are ignored.*

★ *Additional loading: loads distributed along members are applied as masses lumped at the nodes.*

Natural modes

Some versions of CASA support the determination of natural modes of oscillation.

In this window, modes can be chosen and given suitable names. The same mode can be used under multiple names. Only the modes defined will be determined in the analysis.



A factor can be given to scale the **self-mass** of the structure. For true mass, use **1.00**. The self-mass of each member is taken as being evenly distributed along its length (as opposed to being lumped at its end nodes).

★ *Modes of a lower number correspond to lower frequencies; modes of a higher number correspond to higher frequencies. **Mode 1** is the lowest frequency.*

★ *While any arbitrary modes can be chosen, it will take longer to find high modes than low modes.*

★ *The analysis requires that all members have a non-zero mass so that the self-weight of the structure can be determined.*

Combinations

One or more loadcases can be chosen, with each factored and then summed.

If the **Second order** box is ticked then the combination will be analysed with second-order effects (if your version of CASA supports it), otherwise it will be analysed with first-order linear effects only.

In the case of a first-order combination the rules of superposition apply and the combination can be altered after the analysis with the results being updated immediately.

In the case of a second-order combination the rules of superposition do not apply and any change requires a re-analysis.

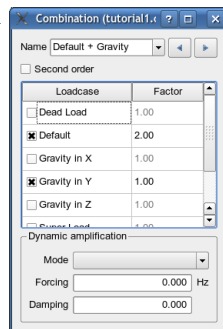
Second-order analysis takes much longer to compute and you lose the convenience of superposition of results.

When using an edition of CASA which supports dynamic analysis, combinations may include a dynamic amplification factor based upon a specified natural mode, forcing frequency, and damping coefficient.

★ *Gravity forces act towards the origin, i.e. in the -ve (negative) direction. Often sway loads will act in the +ve (positive) direction. When combining automatic Gravity in X or Gravity in Z with swaycases, please note the sign and ensure that the applied factors are correctly signed.*

★ *If Gravity in X (or Gravity in Z) is to be considered a swaycase of Gravity in Y, then it would be a swaycase with the following configuration:*

- *Using -ve forces only (signed)*
- *Take F_y and apply in X (or Z for Gravity in Z) – all others ignored*
- *Loadcase Gravity in Y with factor 1.00*



Cameras

A camera can be either *static* or *dynamic*.

A static camera at coordinates has two fixed positions each specified as X, Y, and Z coordinates. The camera sits at one position and looks towards the other.

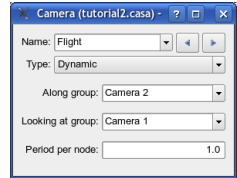
A static camera at nodes has two nodes specified: **At node** and **Towards node**, which indicate where camera is to be placed and where it is focused on. If the nodes change position then the camera will move with them.

A dynamic camera has two groups specified: **Along group** and **Looking at group**, which indicate which two groups contain the nodes for the trajectory of the camera and the trajectory of the camera's focus. When computing the trajectories, nodes are traversed in alphanumerical order. If the contents of the groups change then the camera trajectories will change accordingly.

A **period per node** is specified for dynamic cameras, indicating how quickly the camera should move from one node to the next along its trajectory. The smaller this number, the faster the camera will move.

★ *Trajectories containing three or more nodes follow a curve automatically computed from the relative vectors between each pair of nodes.*

★ *If the number of nodes in the focus group is not equal to the number of nodes in the position group then the period per node applies to the path of the camera position, and the speed of movement along the path of the camera focus is scaled to compensate.*

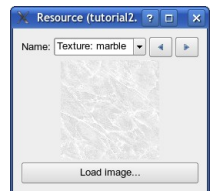


Images

From the main window **Edit menu**, choose **Images...**

This is a central repository for images that can be loaded and named and used elsewhere in the project, such as in title blocks or for decorating members with textures or pictures.

Image files of PNG or JPEG (.png, .jpg) can be loaded, either by clicking on the **Load image...** button or by dropping it onto the picture panel.



Sheets

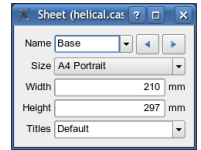
Multiple views can be created by adding sheets. Each view is accessible through the tabs in the main window and have their own independent view points and display settings.

Titles

Each sheet can have the contents of its title block specified. If nothing is specified then the title block will be empty.

Printing

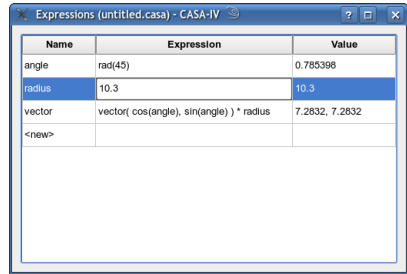
To print multiple graphical scenes from the report generator, first build up a set of sheets each portraying the desired scenes and then add these where appropriate in the report.



Expressions

Expressions are the basis of parametric calculations. The Expression editing window allows variables to be named and an expression given. The expression can refer to other variables, parametric objects (lines, circles, etc.), and structural items (nodes, members, loads, etc.), and can call functions and operate on scalars and vectors.

Variables, objects, and items are referred to by name and can be interrogated for their relevant properties by adding the property name as a suffix to the object name. For example, to find the length of member 1, the expression `member"1".length` can be used.



Name	Expression	Value
angle	rad(45)	0.765398
radius	10.3	10.3
vector	vector(cos(angle), sin(angle)) * radius	7.2832, 7.2832
<new>		

Styles

Styles are used to define the colours and fonts used by the parametric CAD tools.

CAD entities can be moved between styles by selecting them and then applying the desired style via the context menu in this window.

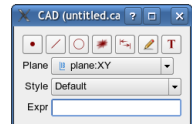


CAD (toolbox)

The CAD window provides the set of tools required for constructing, inking, annotating, and dimensioning a drawing.

Note how few tools are required. This is because CASA can infer the geometrical construction desired by analysing the mouse clicks. For example, to draw a circle simply click on the drawing where the centre will go and then click on a point on the circumference. To draw a circle tangential to three other circles, click on the circumferences of the three circles roughly in the area you expect the tangents to be. You do not need to be precise about the locations because CASA computes them accurately and uses your clicks only as a guide to select the best fit where multiple choices exist.

Because the CAD is associatively parametric, distances, radii, and angles can be provided in the form of expressions or variables, i.e. calculated according to formulae given by the user. Also, when any part of the geometry changes, any corresponding intersections and tangents will be recalculated causing other geometry defined in such terms to be adjusted accordingly.



Constructions are built up with clicks with the left-hand mouse button. Clicking with the right-hand mouse button will cancel the entire construction (as will changing tools).

A pre-existing construction can be edited by highlighting it in the graphical view and then right-clicking to open a context menu and choosing **Edit...** Initially the final click is undone, ready to be redefined. Clicking with the right-hand mouse button over empty space will cancel the edit and restore the construction to its original state. Using SHIFT+right-click over empty space will cause the construction to be unwound step by step so the construction can be redefined by varying degrees or even completely; if unwound completely then it is cancelled and restored to its original state.

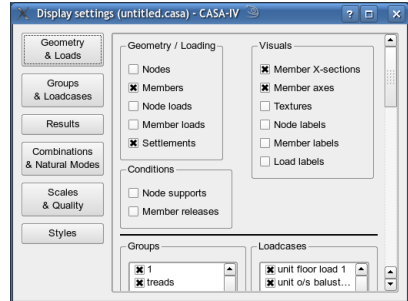
When editing text, initially the text itself is editable. To change its position, unwind the construction with SHIFT+right-click and redefine it with new clicks.

The CAD operates in 3D, but often it is desirable to have it constrained to a plane. Planes can be defined and chosen from the **Plane** menu for this purpose.

Graphical views

From the **View** menu in the main window, **Display settings...** will open up a window with a set of pages containing checkboxes for indicating which types of items are to be displayed and slider bars for adjusting the drawing scales and render quality settings.

Use the buttons on the left of the window or the scrollbar on the right to move between each set:



- **Geometry & Loads:** Checkboxes indicate which items are to be displayed. If *Member X-sections* is ticked then the members are rendered according to their section shape, otherwise they are drawn as simple sticks. If *Member axes* is ticked, and members are drawn as sticks (*X-sections* turned off) then little 'T' shapes will appear at the middle of the members to indicate the major and minor axes directions.
- **Groups & Loadcases:** Of the items ticked in **Geometry & Loads**, only those which exist within the chosen groups and loadcases in this page will be visible. In the case of groups, the box can be half-ticked (faded), for which members in that group will be drawn translucent rather than completely invisible.
- **Results:** Checkboxes indicate which results envelopes and data are to be displayed. *Dynamics – Oscillations* shows the mode shapes normalised such that the largest amplitude is shown equivalent to 10mm. The mode shapes can be animated via the *Scales & Quality* page.
- **Combinations & Natural modes:** Of the results chosen to be displayed in the **Results** page, only those belonging to the combinations and natural modes ticked in this page will be visible.
- **Scales & Quality:** Various scales can be adjusted here. The *Loads* scale is for not only the applied loading but also nodal reactions, member shears, bending moments, etc. The *Displacements* scale applies to both static displacements and dynamic mode shapes.
- **Styles:** Parametric CAD display settings and styles are chosen here.

Notes on Scales & Quality

The *Zoom* factor is different to moving in and out with the mouse wheel. The former is a simple magnification of the image, comparable to using a zoom lens on a camera to get closer to the subject, whereas the latter is a movement in the viewpoint, comparable to moving towards/from the subject with a fixed lens.

The *Loads* scale is logarithmic. The ratio between the largest likely forces and the smallest forces is around 1000000:1, and it is impractical to offer that kind of scaling range on a linear scale.

Time speed states how quickly time passes relative to real time for the purposes of animating the mode shapes of dynamic oscillations. When set to 0.0 the animation is stopped, when set to 1.0 it is running at real speed. However, it is of most benefit when running in slow-motion, often around 0.1.

Visibility sets the distance from the viewpoint that can be seen. Depending on the edition of CASA, anything beyond this point is either invisible or shrouded in fog. The fog is not a gimmick; it gives a subliminal cue to the surrounding geometry which is not present with a simple visible/invisible cut off.

Render quality allows the complexity of the rendering to be adjusted to suit the power of the computer and video card. Low gives the simplest rendering, whereas Highest gives the most sophisticated with smoother curves and antialiasing. Not all video cards (or drivers) support all of the visual features, e.g. some integrated video chipsets (IGPs) do not perform some anti-aliasing effects, and in those cases the Highest setting may not be discernibly better than High or Medium.

Note that fog effects are disabled at quality settings of Medium or lower.

Selections

Graphical selection

Individual items can be selected by CTRL + left-clicking on them in the graphical structure view. They can be removed from the selection by CTRL + right-clicking on them.

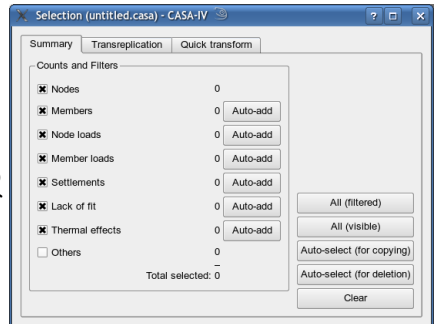
Similarly, CTRL + left-drag will create a rubberband box within which any items that fall will be selected. CTRL + right-drag is used for deselecting items.

In both these cases, an item is only allowed into the selection if it passes the filters set in the Selection window (see next section).

Bulk selection, filtering, & manipulation

In the **Select** menu of the main window, choose **Manipulate...**

A multi-tab window will appear. The first tab is used for filtering and controlling the selection; subsequent tabs are for manipulating the selected items in bulk. There is a field at the top of the window which shows the total number of objects currently selected.



Summary and Filters

The first tab shows an itemised summary of how many of each kind of object are current selected. This is useful for checking that you have not missed any intended items or picked up unintended items.

To the left of each item type is a check-box, and these form a filter. Any items of a type which is ticked are allowed into the selection; any items of a type not ticked are left out of the selection.

Note that changing these check boxes will not alter any selection already made; it only affects future selections.

There are a set of buttons for controlling the selection:

All (filtered) will select everything in the model subject to the filters set by the check boxes.

All (visible) will select everything that is visible in the graphical structure view, again subject to the filters set by the check boxes.

Auto-select (for copying) will look at the currently selected items and see if anything else needs to be selected for it to be possible to copy the current selection. For example, a member cannot be copied unless its end nodes are copied too.

Auto-select (for deletion) will look at the currently selected items and see if anything else needs to be selected for it to be possible to delete the current selection. For example, a member cannot be deleted unless any loads on it are deleted too.

Clear will remove the selection, but will *not* delete anything.

Beside each set of items is an **Auto-add** button. This will select those items which refer to those already selected. For example, auto-add for members will add all members connected to the already selected nodes.

Context-sensitive menu

Delete whole selection will delete as many of the items within the selection as it can, subject to it being possible. For example, a member cannot be deleted if a load applied to it is not also selected for deletion. Use the **Auto-select (for deletion)** facility if you want to be sure of deleting everything.

Split members allows all of the selected members to be divided into parts, with any applied loads automatically split to ensure that the loading does not change with respect to the overall structure.

Transreplication

The transreplication feature allows nodes, and thus any items dependent upon them, to be geometrically transformed and replicated.

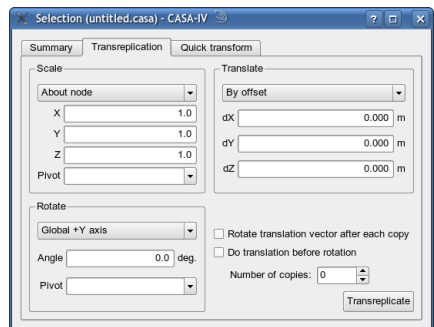
Translation can be either by a specified offset or parallel to the vector between two nodes.

Rotation can be either by a specified angle around a global axis, or by a specified angle around an axis parallel to a member.

The option of **Rotate translation vector after each copy** means that when making multiple copies, the translation vector is rotated after each copy instead of continuing linearly away from the original copy.

The option of **Do translation before rotation** decides the order of applying the transformations.

Number of copies specified how many copies are to be made. If this is set to 0 then the selection is transformed and no copies made; if it is not 0 then the original selected items are left unchanged and only the copies are transformed.



After transreplication, the nodes and members (but not loads) in the original selection and in the copies are consolidated to remove extraneous copies. See the sub-chapters on nodes and members for more details of how they are consolidated.

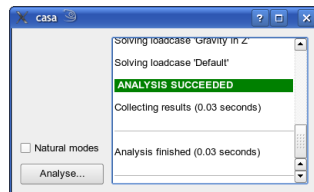
Quick transform

Some common pre-defined transforms are available in this tab for quick and easy use.

Analysis

In the **Analysis** menu of the main window, choose **Analyse....** A window will appear. To begin the analysis, click on the button marked **Analyse...**

Some editions of CASA allow automatic analysis to be enabled, which causes a re-analysis to occur when the model changes without further request from the user.



A log will appear. If the analysis is completed without error then a message to that effect will be given. If errors or warnings have occurred then you will be notified and an explanation should be available in the log.

First-order static analysis

A *first-order* analysis is one which assumes linearity between the applied forces and the resulting displacements, and is thus also called a *linear* analysis. This means that combinations are simply superpositions of factored loadcases, and if a combination is changed in terms of which loadcases it contains or the factors thereof, the new results can be determined immediately and it does not require a re-analysis.

Second-order static analysis

A second-order analysis takes into account induced forces caused by the deflection of the structure. This introduces non-linearities between the original forces and the final displacements, meaning that combinations are no longer superpositions of factored loadcases. If a second-order combination is changed in terms of which loadcases it contains or the factors thereof, it requires a re-analysis to determine the new results.

★ *Second-order analysis is automatically performed for any combinations defined as second-order.*

★ *Second-order analysis is much slower than first-order analysis because it is an iterative process. Usually good convergence occurs after only 3 or 4 iterations, but sometimes when a structure is very flexible or unstable it may not converge. CASA will stop the analysis and issue a warning if no convergence has occurred after 9 iterations.*

There are generally benefits from splitting up members to add nodes between major joints in the structure because this improves the approximation of the bending on each iteration.

Natural frequency analysis

The natural modes of oscillation can be computed with versions of CASA which support it. The time required for this analysis depends upon the highest mode required, not the number of modes. So requesting modes 1, 2, 3, 4, and 5 will take less time than requesting solely mode 10. The lower the mode number, the lower the frequency.

As this analysis is not normally required even when modes have been defined, it is disabled by default. To perform this analysis, tick the box marked **Natural modes**.

What is analysed

All loadcases, swaycases, and gravity loads are independently analysed first-order.
Any combinations marked as being second-order are independently analysed as such.
Any combinations not marked as second-order are therefore first-order superpositions and are computed separately after analysis.
Intermediate forces and deflections along a member are computed according to standard beam theory, and are always first-order calculations.

Auto-analysis

For many structures, analysis is sufficiently quick to make auto-analysis a convenient feature to enable. It is enabled by default (if available on the edition being used). For very large structures, or when running on a laptop with limited battery time, it can be desirable to disable automatic analysis to reduce the workload for the computer. The feature may be enabled or disabled by ticking the relevant box in the analysis window.

Browsing results

Statics

From the **Results** menu in the main window, click on **Statics...**

A loading can be chosen and a series of tabs gives access to tables displaying the member end forces, nodal reactions, nodal displacements, and results along the length of individual members.

Within each tab, as you move the mouse over the rows or select a row by clicking on it, the corresponding item in the graphical structure view will be highlighted.

In addition, point-and-clicking on a relevant node or member in the graphical structure view will make the currently visible table scroll and select the corresponding row.

Double-clicking on a row operates as a Take Me There function and the graphical view will move across to the clicked item.

Results Viewer (tutorial1.casa) - CASA-IV

Choose a loading: Combinations: Default + Gravity Data sorting: By column only

Member end forces		Nodal reactions		Nodal displacements		Member intermediates			
mems	Section type	Nodes A - B	Fx (kN)	Fy (kN)	Fz (kN)	Mx (kNm)	My (kNm)	Mz (kNm)	Stress (N/mm ²)
1	Column	1	90.626	7.692	10.640	0.000	0.000	0.000	3.021
		11	84.850	7.692	10.640	0.000	-26.599	19.231	24.404
2	Column	12	90.626	7.692	-10.640	0.000	0.000	0.000	3.021
		13	84.850	7.692	-10.640	0.000	26.599	19.231	24.404
3	Rafter	3	22.036	20.280	0.448	0.007	0.547	-53.198	87.055
		13	21.487	19.183	0.448	0.007	-0.704	1.954	14.028
4	Rafter	5	20.938	-18.086	-0.448	-0.007	-1.955	54.039	102.915
		14	21.437	-19.183	-0.448	-0.007	-0.704	1.954	14.028
5	Column	6	90.626	-7.692	10.640	0.000	0.000	0.000	3.021
		15	84.850	-7.692	10.640	0.000	-26.599	-19.231	24.404
6	Rafter	7	22.036	20.280	-0.448	-0.007	-0.547	-53.198	87.055
		16	21.487	19.183	-0.448	-0.007	0.704	1.954	14.028
7	Rafter	8	20.938	-18.086	0.448	0.007	1.955	54.039	102.915
		17	21.487	-19.183	0.448	0.007	0.704	1.954	14.028

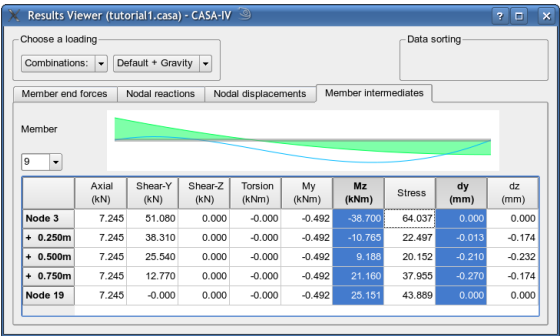
The rows can be sorted according to the values in each column by clicking on the column header. Subsequent clicks will reverse the order of sorting, e.g. from low-high to high-low. Some versions of CASA allow member forces to have their sorting constrained within each section type by choosing **Grouped by section type** from the data sorting menu in the top-right of the window.

A loading can be chosen in the box in the upper left. This gives a choice of:

- **Combinations:** Summation of factored cases, either first-order or second-order as defined within the combination.
- **Loadcases:** Individual cases, first-order, unfactored.
- **Swaycases:** Individual cases, first-order, unfactored.
- **Specials:** Individual gravity, first-order, unfactored. 'g' is taken as 9.81ms^{-2}

The **Member Intermediates** tab contains a view for interrogating the forces and deflections along the member.

As each column is highlighted by clicking on it, a graphical representation is drawn above. More than one column can be chosen by clicking with the CTRL key pressed.

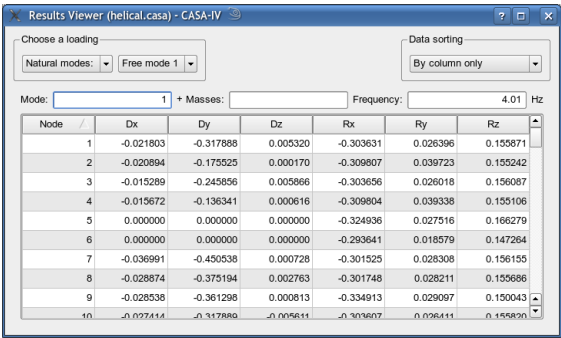


Natural modes

From the **Results** menu in the main window, click on **Natural modes...**

A mode can be chosen and the relative amplitudes for each node will appear, along with the natural frequency of the mode.

The amplitudes are normalised such that the largest magnitude in each mode has the value of either +1.0 or -1.0. Since the actual size of the deflections depends upon the initial conditions in reality, only the relative amplitudes are be given.



If one node has an amplitude of +1.0 and another has an amplitude of +0.5 then the latter moves only half the distance of the former. If one node has an amplitude of +1.0 and another has an amplitude of -1.0 then they move the same distance but in opposite directions. Note that this comparison can only be done within each mode, not across different modes.

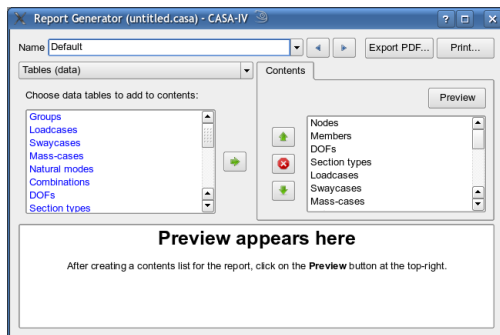
All oscillations throughout the structure within a single natural mode are either in-phase or anti-phase. That is, all nodes are at their maximum displacements along or around each axis at the same time on each cycle and at zero displacement at the same time on each cycle.

Printing reports

In the **Reports** menu in the main window, choose **New report...**

Operation

There are three main areas in this window. The right-hand pane shows a list of contents for the report. This list can be built by selecting entries from the left pane and transferring them across by clicking on the green arrow button. The contents of the pane on the left are controlled by a drop-down menu just above it. The menu is described in the following subsections.



The order of the entries in the contents list can be changed by selecting entries in the right-hand pane and then clicking on the green up and down arrows.

Entries can be removed from the contents list by selecting them in the right-hand pane and clicking on the red cross button.

Entries in either pane are selected by clicking on them. More than one can be selected by clicking with SHIFT or CTRL pressed.

When the contents list is how you like it then click on the **Preview** button to have it compiled and displayed in the bottom pane. For long reports it may take a few seconds for the formatting to be completed.

When you wish to print or export to PDF, click on the buttons in the top-right of the window.

Beside the Preview button are a drop-down list and a numerical input from which the page numbering format can be chosen and the starting page number set. This makes it easy to set the numbering to fit in with other documents or to omit page numbering altogether.

As with other items, multiple reports can be created and named and later recalled, reused, and edited. This makes it easy to create a set of report formats for various purposes.

Tables (data)

This menu item shows a list of tables which can be inserted into the contents list. They are colour-coded according to the nature of the table – blue for containers and attributes, black for geometry, green for loading, red for static results, yellow for dynamic results, and grey for general report formatting.

A page break can be forced by copying *New Page* over into the contents at the appropriate point.

Sheets (graphics)

This list contains all of the sheets in the document, allowing graphical views to be included into the report wherever desired by simply selecting them and moving them across to the contents list.

Note that any sheets using dynamic cameras will have their cameras in their starting positions. Any sheets with a running time base will be seen as a snapshot of how they are at the time of generating the report.

Titles

Sets of titles can be inserted into the contents list. Pages in the report subsequent to the occurrence of a new title will use that title in their title-blocks.

Group filter

This shows a list of the groups which contain nodes and members. One or more groups can be chosen and transferred across to the contents list.

For the tables in the contents list that follow a set of groups, any showing data that is dependent upon nodes or members will be filtered so that only the data relating to the set of groups is shown. The default is for all groups to be set.

Groups can be set anywhere and as often as you like in the contents list. They take effect on the tables following them.

Loadcases

This shows a list of the loadcases which contain loads and settlements. One or more loadcases can be chosen and transferred across to the contents list.

For any consecutive loading tables following a set of loadcases, they are sub-divided into the loadcases. The default, if none are specified, is for loading tables to be generated for all loadcases.

Natural modes

This shows a list of the natural modes. One or more modes can be chosen and transferred across to the contents list.

If the report is to contain amplitude tables then this list indicates which modes are reported. The default, if no list is specified, is for amplitude tables to be generated for all modes.

Combinations

This shows a list of the combinations contain results. One or more combinations can be chosen and transferred across to the contents list.

For any consecutive results tables following a set of combinations, they are sub-divided into the combinations. The default, if none are specified, is for results tables to be generated for all combinations.

Example

An example of a report contents list and how it relates to the printed output is shown in the appendix.

Cameras

Basic principles

A camera can be either static or dynamic.

Static cameras are useful as modelling aids, allowing quick movement from one view to another, as well as being ideal for setting up scenes for presentation to clients.

Dynamic cameras are primarily of interest for presentation to clients. A typical use would be to provide a fly-through of a model, highlighting significant structural or architectural details.

A static camera sits at either a fixed coordinate and looks towards another fixed coordinate, or sits at a single node and looks towards another single node.

A dynamic camera moves on a point along a path while looking towards the corresponding point on another path. The paths are described by sets of nodes within groups. For example, if a camera moves along group A and looks at group B, then the path for A is computed from the positions of every node in group A, following each node in alphanumerical order. The path for B is computed in the same way, using the nodes in group B.

If there are 3 or more nodes in a group then the path trajectory is smoothed out with a curve-fitting algorithm.

Tips

To look along the path being followed

If you wish for a dynamic camera to look along the same path as that being travelled, i.e. looking straight ahead, then do the following (assuming 5 nodes in the path):

- Create nodes C1 to C5 for the path to follow.
- Create node C6 for the final node to focus on.
- Put nodes C1 to C5 in group Path.
- Put nodes C2 to C6 in group Focus (so C2 to C5 are in both groups).
- Create a camera to follow Path and focus on Focus.

In this way the camera is always pointing towards the next node along as it moves.

Advanced Section Generator

The advanced section generator is suited to obtaining the properties for almost any arbitrary section shape, whether it be open, closed, solid, or hollow.

It uses an associative parametric CAD interface for the engineer to draw the cross-section of a member, and the properties will then be determined automatically through a combination of finite-element analysis and integration techniques.

In the editions of CASA which offer this feature, it is found in the Advanced tab of the Section Type editing window.

Tools and buttons

In the top-left of the pane are three buttons. These are the tools – lines, circles, and ink. Simply click on the relevant tool as required when constructing the drawing.

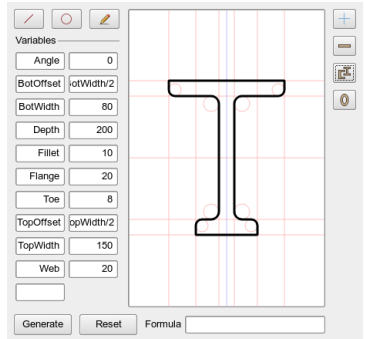
Below the tools is a list of variables and their expressions. Hovering over the expression with the mouse will bring up a tooltip containing the numerical result of the expression.

In the middle of the window is a large area in which the drawing takes place.

In the upper-right is a set of buttons providing short-cuts to pre-defined section constructions. Click on these to have the drawing set up accordingly. The top-most button provides a clean sheet.

Below the drawing area is a field into which formulae can be entered. This is used when creating a construction and is used to express things such as individual angles, distances, or radii.

In the bottom left is two buttons; one to reset the drawing back to its original state, the other to generate the properties. Due to the nature of the calculations, generating properties for complex cross-sections may take a few moments on slower machines.



Relationship to Geometer CAD

The general style of the advanced section generator is the same as that for the 3D CAD offered by the Geometer edition of CASA, but it is not as comprehensive. The CAD and parametric references in the appendices apply with the following limitations:

- The plane is implicitly fixed in the global X/Y axes. As such any reference to planes does not apply and no constructions outside of the plane can be made.

- The parametrics only support references to variables defined within the section generator for the section under consideration. Conversely, the variables defined therein can not be referenced outside of the generator.
- The parametrics do not support references to objects.
- The parametrics do not support vector operations.
 - However: the *innerProduct()*, *length()*, *min()*, and *max()* functions support any number of arguments.
- The paths of inked lines are automatically closed if the end point does not match the start point.

Drawing

The basic principles are described here by way of example. For more advanced information, please refer to the section relating to the Geometer CAD. In particular, we recommend watching the video tutorial for a visual demonstration.

Mouse pointing

When the mouse points at a line or circle then it is highlighted, and that item provides construction information if the mouse is then clicked.

Intersections of lines and circles can also be picked up and used to create constructions dependent upon the positions of the intersections. In contrast, clicking in empty space will position a construction at an arbitrary point which will not move when other constructions change.

Lines

To draw a line, first choose the line tool (click on the line button in the top-left of the pane). Most lines require two clicks to complete. How the line is drawn depends upon on what you click and the order in which you click.

For example:

- Click on an empty part of the drawing (or an intersection) to create a line pivoting around that point. Clicking again on an empty part (or typing a number into the formula field and pressing ENTER/RETURN) will set the line at a given angle.
- Click on a line. This creates a line parallel to the first at a distance. Clicking again on an empty part (or typing a number into the formula field and pressing ENTER/RETURN) will set the line at a given distance from the first.
- Clicking on two lines will create a new line bisecting them.
- Clicking on a circle will create a line tangential to it.

There are many more possibilities. Please see the appendix for a full list.

Circles

To draw a circle, first choose the circle tool (click on the circle button in the top-left of the pane). Most circles require two or three clicks to complete. How the line is drawn depends upon on what you click and the order in which you click.

For example:

- Click on an empty part of the drawing (or an intersection) to create a circle centred on that point. Clicking elsewhere or entering a number in the formula field will set the circle with a given radius.
- Click on a circle. This creates a circle concentric to the first, from which the radius can be set.
- If the SHIFT key is pressed on the first click then it indicates that a three-click construction is to be made. Each click can be on a line, circle, empty space, or an intersection, and the circle will pass through the three points or the nearest relevant tangents as appropriate.

There are many more possibilities. Please see the appendix for a full list.

Ink

Ink is important because each inked line defines the outline of an area of the section. Whole areas should be inked around in a single continuous line; i.e. do not draw individual line segments as that would define the section as being composed of individual lines instead of areas.

Once begun, an inked line continues from click to click and ends when a point is encountered twice or the mouse is double-clicked. If the path drawn by the ink is left open then it is automatically closed by a line between the end and start points. This ensures that areas are always bounded.

Areas can exist within other areas. The outermost area defines a solid region; an area within this defines a hollow region.

Selecting

Selection is performed by holding down CTRL while clicking on a line, circle, or ink. A left-click will select, a right-click will deselect. Clicking into empty space will clear the selection.

If a single item is selected then if it has a user-editable parameter (e.g. an angle or radius) then it will appear in the formula field. Changing the formula will change the item.

Editing & deleting

To edit or delete a line or circle, point to it with the mouse and right-click to open a context menu. Choose Edit or Delete as appropriate. If a selection is present then Delete will apply to the whole selection, not just the item clicked on.

When editing, the final original click of a construction is removed and a new one made by clicking again (or entering a suitable new formula).

When deleting, if other lines or circles are dependent upon the item being deleted then a warning will appear and the deletion will not occur.

★ *To delete an inked line, first select it and then delete the selection.*

Geometer CAD

The Geometer edition of CASA-IV offers associative parametric CAD facilities. These may find many uses, but are particularly useful for the construction and adjustment of complex geometry without manually calculating the locations of nodes.

Even without using the constructional capabilities, the casual user may find the inking, dimensioning, and text tools useful for annotating a conventional structural model.

General operation

Drawing in CAD occurs in the same manner as using the other editing windows – ensure that the CAD tools window has the input focus then click on the graphical view as necessary.

There seven tools, each chosen by clicking on their respective button in the CAD tools window:

- **Points, Lines, & Circles:** These are the primary constructional tools.
- **Planes:** A plane constrains a 2D construction so that it can be easily made within the 3D model.
- **Dimensions:** Linear, angular, and radius/diameter dimensions, both internal and external, can be drawn between points, nodes, intersections, etc.
- **Ink:** This is analogous to drawing on paper. Once constructions have been created, inked lines can be drawn over them. When printing a drawing, turn off the display of constructions to leave only the inking.
- **Text:** Labels and annotation can be pinned in space. They are always plotted in the plane of the screen but may rotate if aligned with a construction.

Constructions are made by clicking with the left-hand mouse button in the relevant places or on the relevant objects to describe the desired method.

To edit a pre-existing construction, point to it with the mouse and right-click to open a context menu. Choose Edit... from this menu. The construction can then be re-described by replacing its final click (or further deconstructed for more extreme editing).

The right-hand mouse button will cancel a new construction if clicked during its creation (i.e. before it is completed); if used with SHIFT during editing of a pre-existing construction it will step backwards, deconstructing, allowing part or all of the construction to be re-described. Right-click and SHIFT+right-click in this context should be done only when pointing at empty space.

Note of 3D/2D effects

CASA operates in 3D but interaction is necessarily restricted to the 2D nature of screen and mouse. This can sometimes cause unintuitive behaviour when trying to point to arbitrary locations in space, such as when positioning text or dimension bars.

When the mouse pointer moves over the 3D space it represents a ray from the cursor on the screen into and through the scene. CASA tries to guess the user's intended position along this ray to pick a 3D position, according to the current focal distance, any chosen planes, and sometimes the first clicks of the construction being made.

For constructions intended to be at particular locations in space relative to other constructions, such as the placement of text, it is recommended that it be confined to a plane or suitable lines/circles be constructed to properly determine the 3D location.

Planes

There is a notion of a "*current plane*". When set, any inherently 2D constructions are confined to the plane to make them simple to create. Any inherently 3D constructions are not affected by the current plane.

To set a current plane, either pick one from the drop-down menu in the CAD tools window or do a SHIFT + *middle*-click (the wheel) with the mouse on a plane, line, or circle in the graphical view. A plane defined solely by a line is such that the line is perpendicular to the plane.

Classes

Planes fall into two categories: 1st class and 2nd class.

A 1st class plane is one whose definition includes a line within the plane, this line forms an "x-axis" of the plane's internal 2D coordinate system. The "z-axis" is perpendicular to the plane and inferred by the construction and the "y-axis" is computed automatically. An example of a 1st class plane is one defined in terms of two coplanar lines.

A 2nd class plane does not have such a line within its definition and as such lacks a well defined internal 2D coordinate system. An example of a 2nd class plane is one defined as parallel to a circle.

In the majority of 2D constructions it does not matter whether a plane is 1st or 2nd class. Where it does make a difference is when something within the plane is arbitrarily placed, such as an angle not expressed relative to anything else. Normally the arbitrary placement is computed relative to the internal coordinate system, but since a 2nd class plane lacks this then such constructions should only be made with 1st class planes.

Appendix A: Analysis considerations

Basic principles of static analysis

CASA uses standard stiffness matrix formulae. The general steps in the analysis are:

- Calculate stiffnesses of every member.
- Calculate equivalent nodal forces.
- Calculate nodal displacements from forces and stiffnesses.
- Calculate member forces from nodal displacements.
- Calculate nodal reactions from member forces.

First order analysis

A first order analysis is described as a linear analysis because the displacements of the nodes are linearly proportional to the magnitudes of the applied forces – double the force, double the displacement. This is a useful property because it allows simple calculations with respect to load factors and combining loadfactors.

However, it cannot take into account the secondary effects of the structure bending under the forces. A structure which works well under first-order conditions may bend too much or collapse under second-order conditions.

Second Order analysis

Second-order analysis is an iterative process which takes into account the deflection of the structure under loading which causes a redistribution of force leading to a change in deflection, and is thus described as a "non-linear" analysis because the final deflections are not linearly proportional to the original applied forces.

Effects on member force computation

Textbook first-order (linear) analysis uses the member vectors from the undeformed structure when calculating the local member forces from the global nodal displacements. For small displacements the error incurred is negligible.

Second-order analysis, by contrast, uses the member vectors from the deformed structure for these calculations.

Note that the intermediate results are first-order calculations and may differ from the member end forces for second-order combinations.

Member stress

The stress in a member, as reported in the static member force results, is calculated as:

$$\sigma = \frac{|F_{axial}|}{A} + \frac{|M_y|}{Z_{yy}} + \frac{|M_z|}{Z_{xx}} \text{ and is negated if } F_{axial} \text{ is negative.}$$

Thermal effects

Thermal effects, i.e. a member's temperature relative to ambient, is modelled through the use of forces induced by its expansion as if it were originally fabricated at ambient temperatures. An axial force only appears if the member is restrained against the expansion.

The analysis determines the forces induced by the temperature difference on a steady-state basis and does *not* consider transfer or diffusion of heat through the structure.

Currently CASA-IV does not support thermal effects on members with end-releases.

Lack of fit

Lack of fit, i.e. a fabrication error in the length of a member, is modelled through the use of forces induced by the error. An axial force only appears if the member is restrained.

The length of the member is defined according to the distance between its end nodes, and it is this which is used for calculating its stiffness and the total forces from distributed loads applied to it.

Currently CASA-IV does not support lack of fit on members with end-releases.

Swaycase determination

Sway cases are derived from loadcases and apply fractions of the loads in another direction. For example, all vertical dead load can be taken and a percentage of it applied horizontally.

The following should be noted:

- The directions considered by the swaycase are always along global axes.
 - Loads applied along other axes (e.g. local member loads) are transformed into their global-axis components, with the components only being considered for the swaycase.
- The *resultant* applied node loads within *each* loadcase are taken, multiplied by a factor, and then added with those of any other loadcases.

- Individual applied node loads within a loadcase may cancel out with others, leaving a smaller or negative resultant.
- The *resultant* applied member loads within *each* loadcase are taken, multiplied by a factor, and then added with those of any other loadcases.
 - Individual applied member loads within a loadcase may cancel out with others, leaving a smaller or negative resultant.
- Loadcases are processed individually before being added together; the resultants used are *not* from the summation of all loadcases.
- Moments are disregarded.
 - In the situation of distributed loads, it is the load envelope that is considered for the sway, not the equivalent end forces.
- Induced forces from temperatures and lack of fit are disregarded.

Natural mode analysis

It may be reasonably intuitive with simple structures to consider their distinct natural frequencies and modes, but as models become more complex it becomes more difficult to interpret them. It is assumed that the engineer understands the meaning of oscillation modes when interpreting the results of this analysis.

The amplitudes given by CASA are normalised such that the largest amplitude in each mode is represented by the value 1.0, as this simplifies the comparison of movements between nodes. The actual amplitude depends upon the initial conditions in reality and can be imagined similarly to a pendulum – it has the same frequency and shape of oscillation independently of the height of its swing, the height being determined by an externally applied initial displacement or force.

Self-weight

The self-weight of the structure is determined by the sizes and densities of the members and section types. All members must have a non-zero size and density otherwise the analysis will not proceed.

The self-weight is considered in the analysis with a consistent mass matrix, i.e. the masses of the members are considered distributed along the member, not lumped at nodes.

Mass-cases

Masses additional to the self-weight of the structure can be specified through mass-cases. A mass-case is a combination of factored loadcases, the *resultant* forces being interpreted as weights, divided by g (9.81) to give a mass. These masses are lumped at nodes and have no rotational inertia. If a distributed mass is to have a consistent mass

matrix then it can be simulated by attaching a suitably sized member to the structure (though this will also introduce additional stiffness, which may not be desirable).

The masses derived through mass-cases can be expressed by forces in any direction, the *length of the resultant vector* being considered as the weight. This allows for centrifugal weight as well as gravitational weight to be applied. For example: a sole 9.81kN in Y at a node will give a 1000kg mass, as will a sole 9.81kN in X. If you apply 9.81kN in X, in Y, and also in Z, all at the same node, then you will have a mass of 1732kg (the resultant). However, a mass will exist in all directions regardless of the directions of the forces which define it.

Note that if two forces cancel out at a node then their equivalent mass will be zero.

Moment forces are disregarded; only linear forces are considered as equivalent mass.

Member end-releases

Member end releases introduce a discontinuity in the stiffness of the structure; i.e. the stiffness is zero along or around an axis at the end of the member, preventing forces from passing to or from the associated node and allowing the end of the member to move independently of the node.

As of version 1.147, end-releases also create a discontinuity in *mass*, so the mass of the member at a released end could manifest itself on the unreleased end.

Due to the purity of a mathematical release compared to physical realisation, their interpretation in a dynamic analysis is the subject of academic debate. Some researchers recommend that end releases be removed. However, it remains the user's engineering judgement as to whether the presence or removal of a release is valid for their purposes.

Relative Amplification Factors

The results of a natural mode analysis can be used to determine the relative amplification factor (RAF) for a static loadcase combination. The following points should be noted:

- The RAF considers only a single natural vibration frequency (f_n) and a single forcing frequency (f_f).
- An overall structural damping coefficient can be included in the determination of the RAF, but bear in mind that the natural vibration frequency is calculated without damping.
- The chosen natural mode and its mass-cases should be appropriate for the combination loading.

- $$R_{AF} = \frac{1.0}{\sqrt{(1.0 - r^2)^2 + (2 \cdot c \cdot r)^2}} \quad \text{where } r = \frac{f_f}{f_n} \text{ and } c = \text{damping coefficient.}$$

Appendix B: Mouse controls

Interacting

When pointing to empty space in the graphical view

		+ <i>SHIFT</i>	+ <i>CTRL</i>
<i>Hover</i>	-	-	-
<i>Left click</i>	-	-	-
<i>Middle click</i>	-	-	-
<i>Right click</i>	Cancel construction (CAD only)	Unwind construction (CAD only)	-
<i>Left double-click</i>	-	-	-
<i>Middle double-click</i>	-	-	-
<i>Right double-click</i>	-	-	-
<i>Scroll wheel</i>	Move in/out of the model	Move in/out faster	-
<i>Left drag</i>	Rotate elevation and heading	-	Select objects within a rubberband area
<i>Middle drag</i>	Rotate around viewing axis	-	-
<i>Right drag</i>	Pan	-	Deselect objects within a rubberband area

When pointing to an object in the graphical view

		+ <i>SHIFT</i>	+ <i>CTRL</i>
<i>Hover</i>	Highlight the object and pop up information	-	-
<i>Left click</i>	Put a reference to the object in the current editing window and advance the input focus	Put a reference to the object in the current editing window but do not advance the input focus	Select the nearest object
<i>Middle click</i>	-	Set plane (CAD only)	-
<i>Right click</i>	Open context-sensitive edit menu for the object	Focus on the object	Deselect the object
<i>Left double-click</i>	As for click	-	Select the object and its dependencies
<i>Middle double-click</i>	-	-	-
<i>Right double-click</i>	As for click	-	Select the object and its dependencies
<i>Scroll wheel</i>	Same as when pointing to empty space		
<i>Left drag</i>	Same as when pointing to empty space		
<i>Middle drag</i>			
<i>Right drag</i>			

When pointing to the X, Y, Z, or base of triad

		+ <i>SHIFT</i>	+ <i>CTRL</i>
<i>Hover</i>	Same as when pointing to empty space		
<i>Left click</i>	Same as when pointing to empty space		
<i>Middle click</i>			
<i>Right click</i>			
<i>Left double-click</i>	Move to corresponding standard viewpoint	-	-
<i>Middle double-click</i>	Same as when pointing to empty space		
<i>Right double-click</i>			
<i>Scroll wheel</i>	Same as when pointing to empty space		
<i>Left drag</i>	Same as when pointing to empty space		
<i>Middle drag</i>			
<i>Right drag</i>			

Cameras

When a camera is running

		+ <i>SHIFT</i>	+ <i>CTRL</i>
<i>Hover</i>	-	-	-
<i>Left click</i>	-	-	-
<i>Middle click</i>	-	-	-
<i>Right click</i>	-	-	-
<i>Left double-click</i>	-	-	-
<i>Middle double-click</i>	-	-	-
<i>Right double-click</i>	-	-	-
<i>Scroll wheel</i>	Jog fast & slow, forward, rewind, or stop	Jog normal speed forward, rewind, or stop	-
<i>Left drag</i>	-	-	-
<i>Middle drag</i>	-	-	-
<i>Right drag</i>	-	-	-

Flying

At any time except when a camera is running

<i>Left pressed + scroll wheel spun</i>	Begin flight and/or adjust speed of flight
<i>Left held pressed</i>	Maintain flight
<i>Left held pressed + mouse moved</i>	Pitch up/down and bank left/right
<i>Release left button</i>	Stop flight

Appendix C: Example report

Contents list	Actual report layout
--> Insert sheets Main view	"Main view" graphics
--> Filter by groups G1 + G2	
Nodes	Nodes in groups G1 + G2
Section types	Section types
Members	Members in groups G1 + G2
--> Insert sheets Geometry Front elevation	"Geometry" graphics "Front elevation" graphics
--> Arrange loading by loadcases LC1 + LC2	
Node loads	Node loads in LC1 on nodes in G1+G2
Member loads	Member loads in LC1 on members in G1+G2
	Node loads in LC2 on nodes in G1+G2
--> Arrange results by combinations CM1 + CM2	Member loads in LC2 on members in G1+G2
Reactions	Reactions for CM1 on nodes in G1+G2
Displacements	Displacements for CM1 on nodes in G1+G2
	Reactions for CM2 on nodes in G1+G2
	Displacements for CM2 on nodes in G1+G2
Settlements	Settlements in LC1 on nodes in G1+G2
--> Filter by groups <all>	Settlements in LC2 on nodes in G1+G2
Member forces	Member forces for CM1 on all members
	Member forces for CM2 on all members
--> Insert sheets Results CM1 – Deflections Results CM1 – B.M. Results CM2 – Deflections Results CM2 – B.M.	"Results CM1 – Deflections" graphics "Results CM1 – B.M." graphics "Results CM2 – Deflections" graphics "Results CM2 – B.M." graphics

Appendix D: CAD reference

Click sequence tables

For each tool there are sequences of clicks which determine how the geometry is constructed.

Key: U = A user-entered value, usually a 3D coordinate (e.g. 3,4,5) or a single number representing a distance or angle.

P = Click on a point, in free-space, or at intersection/tangent of lines and circles. Nodes may be used as points for the purpose of constructions.

L = Click on a line.

C = Click on a circle.

M = Click on a member.

^ = Hold down the SHIFT key while making the click.

Sometimes an object and a user value are entered simultaneously. This is denoted by, e.g., "LU". In this example, either hover the mouse over the line and type in the value and press ENTER, or type in the value without pressing ENTER but then click on the line.

Points can be created at arbitrary positions on lines, circles, and planes, but note that they may drift or move unpredictably if the lines, circles, or planes are subsequently transformed through rotations or translations.

Some constructions are inherently 2D, and as such require a current plane to be set (in the drop-down menu within the CAD editing window). These are indicated with ticks in the tables in the Planar column.

There are two kinds of plane: those that are defined only in terms of a normal vector (2nd class), and those that are defined with at least one in-plane vector (1st class). The latter are deemed to have a coordinate system in which the in-plane vector defines the X-axis, with either the Y (in-plane) or Z (normal) calculated according to whichever is not defined. Where a 1st class plane is required for correct geometrical constructions (such as those requiring arbitrary angles), it is indicated by "(1)" in the planar column.

Constructions which work in 3D will ignore any currently chosen plane.

Choosing a plane

Planes can be chosen either from the drop-down menu (which lists the defined planes) in the CAD editing window or by holding down SHIFT while clicking with the *middle* mouse button (press down on the wheel) on either a plane, line, or circle.

Point tool

<i>Sequence</i>	<i>Constructs new point at</i>	<i>Notes</i>
U	Coordinate U	U must be a 3D coordinate
Free space	Arbitrary position in space or plane	Acts in current plane, if any
P – P	Mid-way between both points	
P – L	Distance along L from P	The distance is determined by the position of the click.
P – LU	Distance U along L from P	U is a single number
L	Arbitrary position on L	
C	Arbitrary position on C	
^C	Centre of circle	

Line tool

<i>Sequence</i>	<i>Planar</i>	<i>Constructs new line as</i>	<i>Notes</i>
P – U	✓ ⁽¹⁾	Through P at angle U	
P – P		Through both points	
P – L		Through P perpendicular to L	
P – C		Through P tangential to C	Both must be co-planar
L – P		Parallel to L through P	
L – U	✓	Parallel to L at distance U	
L – L		Bisect two lines	
L – ^L		Normal to both lines (cross-product)	Both must be co-planar
L – C		Parallel to L tangential to C	Both must be co-planar
^C		Axial through centre of C	
C – U	✓ ⁽¹⁾	Tangential to C at angle U	U is in degrees
C – P		Equivalent to P – C	
C – L		Tangential to C perpendicular to L	
C – C		Tangential to both circles	Both must be co-planar

M		Co-linear with member	
^M		Co-linear with member major axis	

Circle tool

<i>Sequence</i>	<i>Planar</i>	<i>Constructs new circle as</i>	<i>Notes</i>
$\wedge P - U$	✓	Centre P radius U	
$\wedge P_1 - P_2$	✓	Centre P_1 circumference through P_2	
$P - L$		Centre P tangential to L	
$P - C$		Centre P tangential to C	Both must be co-planar
$P - \wedge C$	✓	Centre P same radius as C	
$C - U$		Centre C radius U <i>relative</i> to C	U is the difference in radii.
$C - P$		Centre C circumference through P	Both must be co-planar
$C - L$		Centre C tangential to L	Both must be co-planar
$C_1 - C_2$		Centre C_1 tangential to C_2 or bisector	Both must be co-planar If concentric, they are bisected.
$C_1 - \wedge C_2$		Centre C_1 same radius as C_2	
$\wedge +$ any 3-click combination of points, lines, or circles		Circumference through points and/or tangential to lines and circles.	All must be co-planar. The final value may be a user-entered radius or $\wedge C$ for equal radius.

Planes

<i>Sequence</i>	<i>Constructs new plane as</i>	<i>Notes</i>
$P - P - P$	Through three points	1 st class
$P - L$	Coplanar with point and line	1 st class
$L - P$	As $P - L$	1 st class

L – L	Coplanar with both lines	1 st class
Intersect of 2 lines	Coplanar with both lines	1 st class
L – C	Plane of circle, x-axis along L	1 st class
L	Normal to line	2 nd class (no coordinate system)
C	Coplanar with circle	2 nd class (no coordinate system)
M	Normal colinear with member minor axis, local x-axis along member	1 st class

Dimensions

<i>Sequence</i>	<i>Constructs new dimension as</i>	<i>Notes</i>
$P_1 - P_2 - P_3$	Distance P_1 - P_2 with bar through P_3	
$P_1 - P_2 - L$	Distance P_1 - P_2 projected along L	
$L - L - P$	Angle between lines through P	Inside/outside is determined by position of P
$C - P$	Size of circle C through P	Radius/diameter and inside/outside are determined by position of P

Ink

<i>Sequence</i>	<i>Constructs new inked line as</i>	<i>Notes</i>
$P - P - \dots$	Straight between points	
$P - C - P - \dots$	Between points following around C	
Any sequence of the above	Polygon	Polygon is closed if end point is same as start point, otherwise polygon is open
A sequence ends when one point is encountered twice, such as closing a polygon or double-clicking.		

Text

<i>Sequence</i>	<i>Constructs new text as</i>	<i>Notes</i>
-----------------	-------------------------------	--------------

P – U	Text "U" at P	Plotted in plane of screen
L – P – U	Text "U" at P parallel to L	Plotted in plane of screen

Appendix E: Parametric Reference

General

Expressions are mathematical formulae which can also make reference to other expressions. Often they are used in named variables to allow the engineer to build up mathematical relationships within the structure.

The format is similar to that used by most spreadsheet programs. For example:

$$\cos(\text{angle}/2)*\text{radius}$$

will yield an answer of *radius* multiplied by the cosine of half of *angle*. Functions that take angles will consider them in radians, so be careful when mixing them with degrees.

Vectors can be used with most operators and functions. Be careful when expressing vectors directly in numerical form – functions taking two or more arguments must have any vector arguments enclosed in braces.

e.g. if(a<b, {1,2,3}, {4,5,6}) is correct, whereas if(a<b, 1,2,3, 4,5,6) is wrong, because the former has three parameters and the latter has seven.

The values of variables and functions yielding vectors need not be enclosed in braces.

Operators

Note that, unless specified otherwise, all operators allow X to be either scalar or vector though Y must be scalar. Vector operations occur element-wise and yield a vector.

The table below shows the available operators grouped in order of priority.

<i>Operator</i>	<i>Description</i>	<i>Notes</i>
(X)	Parentheses for priority of evaluation	
{ X, Y, ..., Z }	Vector of elements X, Y, ..., to Z	Vector can be any length
+ X	Positive X	Identical to X
– X	Negates X	Element-wise with vector
X [Y]	Pick index Y from vector X	Y must be scalar
X ^ Y	X raised to the power of Y	X & Y must be scalars
X * Y	X multiplied by Y	
X / Y	X divided by Y	
X % Y	Remainder of X / Y	

<i>Operator</i>	<i>Description</i>	<i>Notes</i>
$X + Y$	X plus Y	
$X - Y$	X minus Y	
$X = Y$	Test X equal to Y	All tests yield a scalar: 1.0 if TRUE 0.0 if FALSE
$X < Y$	Test X less then Y	
$X \leq Y$	Test X less than or equal to Y	
$X > Y$	Test X greater than Y	For vectors, every pair of corresponding elements must test TRUE for the result to be TRUE
$X \geq Y$	Test X greater than or equal to Y	
$X \diamond Y$	Test X not equal to Y	
$X \sim Y$	Test X approximately equal to Y	About 6 significant figures

Functions

All functions take one or more parameters. Any single-input scalar functions can operate either on a scalar or element-wise on a vector unless stated otherwise.

<i>Operator</i>	<i>Description</i>	<i>Notes</i>
abs (X)	Absolute magnitude	
acos (X)	Inverse cosine	
acosh (X)	Inverse hyperbolic cosine	
angle (X,Y)	Angle (in radians) of X,Y vector	X & Y are scalars
asin (X)	Inverse sine	
asinh (X)	Inverse hyperbolic sine	
atan (X)	Inverse tangent	
atanh (X)	Inverse hyperbolic tangent	
cbrt (X)	Cube root	
ceil (X)	Round up to nearest integer	
cos (X)	Cosine	
cosh (X)	Hyperbolic cosine	
deg (X)	Convert radians to degrees	

<i>Operator</i>	<i>Description</i>	<i>Notes</i>
exp (X)	Base-e exponential	For other bases, use B^X
floor (X)	Round down to nearest integer	
if(X,Y,Z)	If (X \neq 0) then Y else Z	X must be a scalar or relational
innerProduct (X)	Inner product of vector	X is vector
length (X)	Length of vector (resultant)	X is vector
ln (X)	Base-e logarithm	
log10 (X)	Base-10 logarithm	
max (X)	Maximum value in vector	X is vector, result is scalar
min (X)	Minimum value in vector	X is vector, result is scalar
rad (X)	Convert degrees to radians	
round (X)	Round off to nearest integer	
sin (X)	Sine	
sinh (X)	Hyperbolic sine	
sqrt (X)	Square root	

Constants

The value π is predefined and can be used directly in expressions.

Object types

All objects are accessed in an expression with the syntax:

type " name "

Variables can be accessed simply by their name (i.e. without the type prefix or quotes) if they begin with a letter and use only letters, numbers, and the underscore character.

The following types are recognised:

Variables	var" ... " (or just the name)
Points	point" ... "
Lines	line" ... "
Circles	circle" ... "
Planes	plane" ... "
Ink	ink" ... "
Dimensions	dim" ... "
Text	text" ... "
Nodes	node" ... "
Members	member" ... "
Node loads	nload" ... "
Member loads	mload" ... "
Settlement	settle" ... "
Lack of fits	lof" ... "
Thermal effect	thermal" ... "

Object characteristic suffixes

Objects can be simply referred to by their type and name when, for example, passing them to a function that expects such an object as a parameter. However, for other uses most objects have properties which are used with the following dot-suffix syntax:

type " name " . property or type " name " . function (arguments)

Functions which take arguments have them listed within brackets, each separated by a comma. Functions which do not take arguments have no brackets.

An example of suffix usage would be the expression:

member"rafter1".length

which would return a single number equal to the length of the member called rafter1.

Variables

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
hypot	Calculates square-root of inner product (hypoteneuse of vector)	Variable should be a vector
innerProduct	Calculates inner-product	Variable should be a vector

Points

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
pos	Coordinate of point	

Lines

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
nearestToPoint(P)	Coordinate on line nearest to P	P is a point
through	A coordinate through which line passes	
vector	Normalised vector of line	

Circles

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
axis	Normalised axial vector	
centre	Coordinate of centre	
radius	Radius	Always +ve

Planes

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
normal	Normalised vector perpendicular to plane	

Ink

There are no properties for ink.

Text

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
toNumber	Returns the numerical value(s) represented by the text	Separate values with commas for a vector

Nodes

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
pos	Coordinate of node	

Members

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
end	Coordinate of node B	
length	Length of member	
major	Normalised vector of major axis	
minor	Normalised vector of minor axis	
start	Coordinate of node A	
vector	Normalised vector of member	

Node loads

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
force	X, Y, & Z linear forces	
moment	Mx, My, & Mz moment forces	

Member loads

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
forceA	X, Y, & Z forces at node A	Global or local, as defined
forceB	X, Y, & Z forces at node B	Global or local, as defined
length	Length of run	
torsionA	Torsion at node A	Local coordinates
torsionB	Torsion at node B	Local coordinates

Settlements

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
offset	Linear offset	
rotation	Rotational offset	

Lack of fits

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
delta	Delta error	Absolute distance
fraction	Fractional error	Relative to member length
total	Total error	May depend on member length

Thermal effects

<i>Property suffix</i>	<i>Description</i>	<i>Notes</i>
dTmajor	Temperature difference across major axis	
dTminor	Temperature difference across minor axis	
tA	Temperature at node A	Relative to ambient
tB	Temperature at node B	Relative to ambient
mean	Mean temperature along member	

Appendix F: Miscellaneous hints

This appendix lists various miscellaneous hints, tips, and trivia regarding CASA.

- Files can be loaded and/or merged with the current file by dragging them into the main window.
- Double-clicking on a line in a results viewer table will move the current graphical view as necessary to focus on the relevant node or member (similar to "Take me there").
- Library items are protected against modification.
- When moving the graphical view with the mouse, the speed depends upon the focal distance. If it is too slow or too fast, point at an object (e.g. node, member) near the region of interest and start the drag with it highlighted.
- Shift + Page Up/Down in an editing window will move to the first item or to the *<new>* item.
- Input locks record the data from the last viewed item in the editing window.
- Multiple editing windows for the same kind of item can be open simultaneously, each with their own set of locked data.
- A quick way to begin editing an object is to point to it in the graphical view, click with the right-hand mouse button to open a menu, and choose "Edit...".
- If the name field is locked then automatically generated names will follow the pattern set by the last viewed item.
- The lists of names in an editing window are alphabetical for all items created before the window was opened, followed by later items in creation-order.
- Sheets can be re-ordered in the main window by dragging their thumbnails up and down.
- If the results after an analysis have been invalidated (e.g. by changing the model) then the report will remain in the analysis window for perusal but it will be faded to show that it is no longer current.
- Printable reports are items just like any other – once one has been generated it can be reused by choosing its name from the menu at the top of the window.

Appendix G: Trouble-shooting

If you encounter any problems with CASA-IV then please let us know, no matter how trivial you think they may be, because we always aim to put right any deficiencies and improve the usability of our software.

Installation

CASA-IV will happily run for users without administration rights (i.e. “normal” users), but installation requires some of these rights (an “administrator”). This is common to the majority of programs installed on Windows, so if you have installed programs before then carry on the same for CASA as you have for those others.

You can be logged on as any user and you will be prompted for authorisation as needed.

Graphics / Video

nVidia and AMD graphics are recommended. Integrated Intel graphics should be okay for smaller models, but may have some lag on larger models.