

GS Plan 5.7 Designer's Manual



Keymark

6707 Winchester Circle
Boulder, Colorado 80301

(303) 443-8033

10/6/2014

v.0.21

Table of Contents

The GS Plan Interface.....	6
The GS Plan Window.....	6
Menu Bar.....	6
Drawing Area.....	7
Toolbars.....	7
Status Bar.....	7
Right Click Menu.....	8
The GS Plan Menu Bar & Toolbars.....	10
The File Menu & Toolbar.....	10
The Tools Menu & Toolbar.....	17
Bottom Snap Points.....	19
Command Buttons.....	19
Command Buttons.....	22
Command Buttons.....	23
Start Point.....	24
End Point.....	24
Scan.....	27
[Actions].....	28
Command Buttons.....	28
Create/Edit Option.....	29
Command Buttons.....	30
Select Options.....	31
Conflicting Options.....	32
Copy/Xfer Options.....	33
Command Buttons.....	33
Copy Selected Entities.....	33
Option Materials.....	36
The Modify Menu & Toolbar.....	37
Entities Menu.....	45
Command Buttons.....	47
Display Menu & Toolbar.....	62
Macros Menu.....	67
Top Track Elevation.....	73
Bottom Track (Walls, Headers, Posts).....	73
[Other].....	73
Command Buttons.....	74
Place new veneer wall at.....	74
New Veneer Wall Properties.....	74
Command Buttons.....	75
Outside.....	76
Inside.....	76

Slopes	77
Seat Cuts	77
Command Buttons	77
Ceiling Joists	79
Command Buttons	80
Command Buttons	81
[Framing Supports]	81
Command Buttons	82
Command Buttons	84
How to Place Entities	84
The Page Up Key	84
The Page Down Key	84
The Home Key	85
The End Key	85
Macros Menu (Continued)	85
Lateral Menu	86
Materials Menu	87
Command Buttons	87
Material Category Descriptions & Measures	88
Design Menu	99
GS Build Menu	100
Help Menu	101
Plan Toolbars	102
The Annotations Toolbar	102
The Header Macros Toolbar	102
Command Buttons	105
The GS Build Toolbar	107
The Options Toolbar	108
The Panel Toolbar	108
The Roof Planes Toolbar	108
The Selection & Snap Toolbar	108
The Standard Toolbar	108
The Structure Commands Toolbar	111
The Toggle Macros Toolbar	115
The TOP Toolbar	117
The Wall Panel Toolbar	117
General Information Tabs & Settings	118
Job Info Tab	118
Job Settings Tab	119
Extend Planes	122
Plot Program	122
Command Buttons	122
Load Info Tab	123

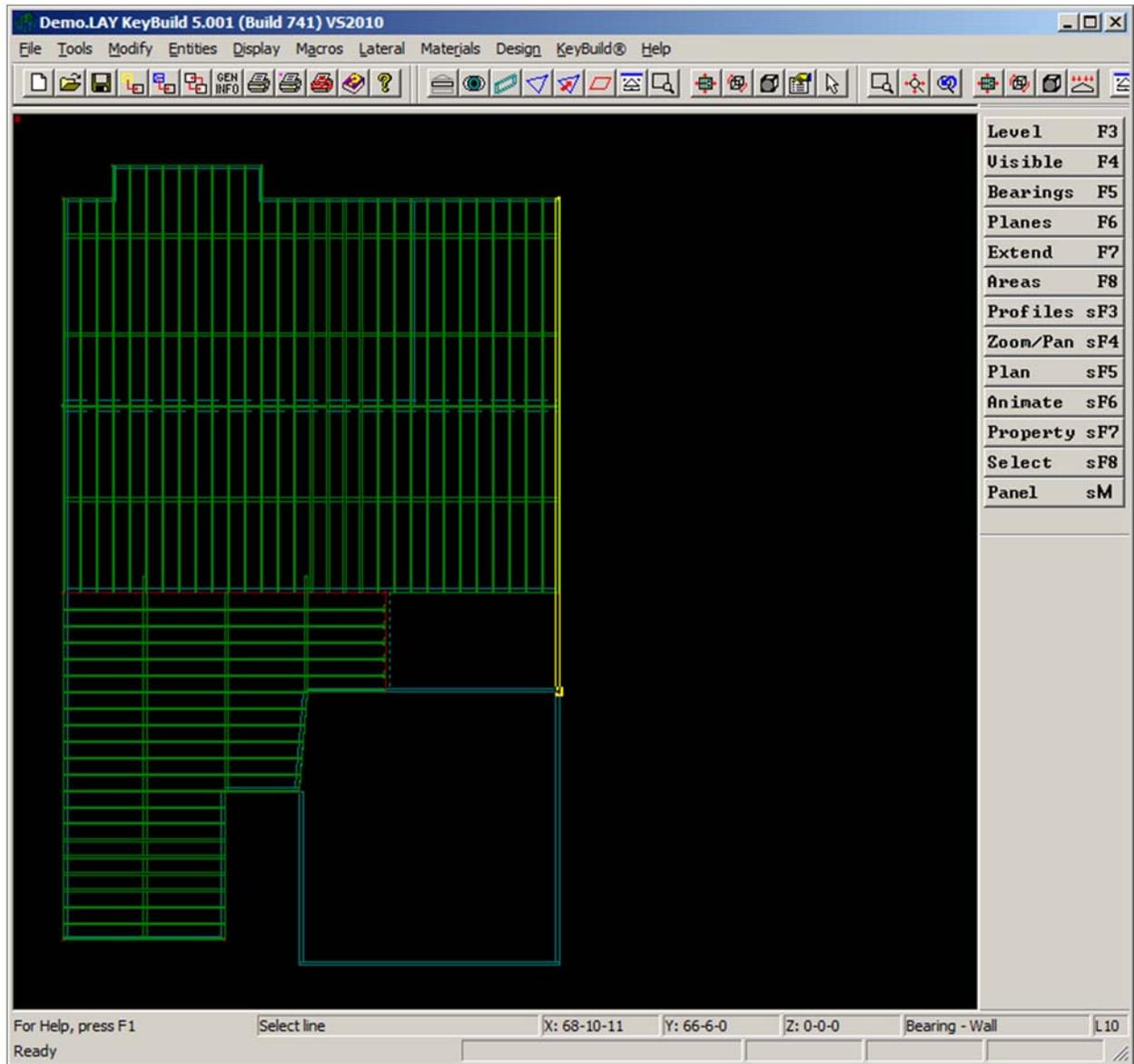
Floor Loads.....	123
Wall Loads.....	123
[Other]	124
Roof Loads.....	124
Tributary Area.....	124
Command Buttons	125
Framing Members Tab	126
Blocking Panels	126
Command Buttons	127
Other Materials Tab	128
Command Buttons	129
Options Tab.....	130
GS Wall Settings Tab	131
Command Buttons	131
Design Settings Tab	132
Allowable Spacings	132
Roof.....	132
Floor.....	132
Ignore Warning Messages	133
Command Buttons	133
Deflection Criteria Tab	134
Floor Joist.....	134
Floor Bearing.....	134
Flat Roof.....	135
Sloped Roof.....	135
Command Buttons	136
Joist Accessories Tab	137
Floor Joist Framing.....	137
Roof Joist Framing.....	139
Ceiling Joist Framing	139
Command Buttons	140
Blocking & Rim Tab	142
Blocking Settings.....	142
Joist Rim Settings	142
Mid-Span Blocking Settings	143
Ladder Settings	143
Command Buttons	143
Material Lists Tab	144
Command Buttons	145
Material Table Formatting.....	145
Cut Lengths Tab.....	146
Color & Line Types Tab.....	147

Line Types	147
Colors	147
Line Styles	148
[Other]	148
Plot Labels Tab	149
Settings	149
Plot Label Formatting.....	149
Label Size.....	151
Working with Options	152
Drawing an Option	152
Adding a Sub-Option	154
Excluding Options from Each Other	154
Using Macros with Options.....	155
Glossary of Terms.....	156
Set Bearing Properties for a Wall.....	157
Set Bearing Properties for a Girder.....	158
Set Bearing Properties for a Beam.....	159
Set Bearing Properties for a Header	161
Set Bearing Properties for a Truss Girder	162
Set Bearing Properties for a 'Girder by Others'	163
Set Bearing Properties for a 'Beam by Others'	164
Set Bearing Properties for a Flush Wall	166
Set Bearing Properties for a Cut Joist	167
Set Bearing Properties for a Ridge Board.....	168
Set Bearing Properties for a Ledger	169
Set Bearing Properties for a Strut	171
Command Buttons	174
Selecting Door & Window Components	175
Appendix C: GS Plan Menus & Toolbars	176

THE GS PLAN INTERFACE

The GS Plan Window

The *Main Window* in GS Plan contains the *Drawing Area*, the menu bar, toolbars, and the status bar.



MENU BAR

Entries in the menu bar, located at the top of the GS Plan window, can be accessed one of three ways:

1. Move your mouse cursor over the name of the menu that you want to access and then click the left mouse button. Then select the command you want from the menu by clicking on the command name with the left mouse button.

2. Start by pressing the F10 key. This will activate the *File* menu. Then use the right arrow key to move to the menu needed, then the down arrow to move to the command needed. When the proper command is highlighted, press the *Enter* key.
3. Hold down the *Alt* key and then press the letter that is underlined in the menu name. For example, in *File*, the “F” is underlined, so press *Alt* and *F* at the same time to bring up the *File* menu. Then press *P* to print (because the P is underlined letter in *Print Drawing*).

DRAWING AREA

The *Drawing Area* is the central portion of the *Main Window* in which layouts are displayed. All input, editing, and viewing of the model takes place within the *Drawing Area*. When in the *Drawing Area*, there are three different cursor modes. Depending on the mode that you enter, your cursor will change appearance:

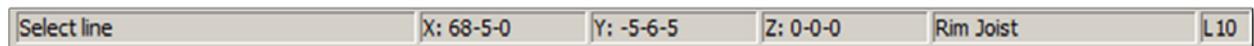
1. If you are in the *Select Mode*, the arrow cursor will appear. (This cursor will also appear if you leave the *Drawing Area*.)
2. If you are in *Drawing Mode* or *Input Mode*, the crosshair cursor will appear.
3. If you are in *Animate Mode*, the animate cursor will appear.

TOOLBARS

Toolbars can be added or subtracted from the *Main Window* in two ways:

1. Click on the *Tools* menu and scroll down to the *Toolbars* option. On the sub-menu that appears to the right, click on the toolbar that you want to add or remove from your view. (The toolbars that are currently displayed will have a checkmark next to them in the sub-menu).
2. Right-click in a gray area of the GS Plan Window (for example, the gray area to the right of the Help menu). On the menu that appears, click on the toolbar you want to add or subtract from your view. (The toolbars that are currently displayed will have a checkmark next to them.)

STATUS BAR



The status bar displays information about the function currently being performed. Below is a sample status bar.

- *Prompt* – The *Prompt* informs the user about the next step that should be performed in the application. (For example, "Set first point" when starting to draw a bearing.)
- *X Coordinate* – This is the current location of your cursor along the X-axis (left – right, horizontal in *Plan View*).
- *Y Coordinate* – This is the current location of your cursor along the Y-axis (up – down, vertical in *Plan View*).
- *Z Coordinate* – This is the current location of your cursor along the Z-axis (height).
- *Current Entity* – If you are in a *Drawing Mode*, this is the type of entity you are currently drawing or are going to be drawing. If you are in a *Select Mode*, then this is the type of entity that you last selected.

- *Current Snap Modes* – These are the snap modes that you currently have activated.
 - ◆ Z represents XYZ Snap
 - ◆ P represents Point Snap
 - ◆ G represents Grid Snap
 - ◆ represents Ortho
- *Level* – This is the current active level in GS Plan. To change the levels that are active and/or visible, choose the *Set Levels* option on the *Tools* menu.

RIGHT CLICK MENU

The right-click menu provides shortcuts to some of the most often used commands in GS Plan. The commands that are available at any given moment are contingent upon the actions currently being performed by the user. Following are some of the commands commonly available from the right-click menu:

Deselect – The *Deselect* command allows you to deselect selected (highlighted) items in the model. No items will be selected after this command is executed.

General Information – Select *General Information* from the *File* menu to open the *General Information* window. This window allows users to set up default information for the current layout. It is the same window that appears when you start a new job. For more information, see the section within this document entitled “*General Information Tabs & Settings.*”

Set Levels – The *Set Levels* command allows users to create new levels, and to view and edit information about all existing levels in the current model. **For additional information, see the section within this document entitled...**

Set Visibility – The *Set Visibility* command allows users to select the entities to be displayed in the *Drawing Area*, and those to be hidden. If the name of a member type has a check mark next to it in the *Set Visibility* window, it will be displayed in the *Drawing Area*. If the member is not checked, it will not be displayed. To check (or uncheck) a member type, click in the box to the left of the name. **For additional information, see the section within this document entitled...**

Plan View – The *Plan View* command displays a model from above, as large as possible on the screen, so that it fills the entire *Drawing Area*.

Animate – The *Animate* command allows users to view and rotate a model in 3D space. After selecting *Animate*, your model will be displayed in a preset 3D view. Use the arrow keys on your keyboard (not the keys on the numeric keypad on your keyboard) or your mouse to move the model. To exit the *Animate View* press the *Escape* key. If you move to a 3D view that you wish to keep, either click the left mouse button or press the *Home* key on your keyboard.

Solid Model – This feature allows users to view jobs with a *Solid Model View* in a modeless window. The *Solid Model View* window can be resized and/or repositioned at any time. Size and position information are stored when the viewing window is closed and restored when the window is opened. Pressing the *Escape* (*Esc*) key while in the viewing window will close the window and return the user to the standard *Model View*.

Undo – The *Undo* command reverses the action of the last single-line *Modify* command.

Redo – The *Redo* command reverses the action of the last *Undo* command.

Tools – The right-click *Tools* option opens a *Tools* menu with a subset of commands that are available from the *Tools* option on the menu bar. For more information about the features and functions available on the *Tools* menu, see the section within this document entitled _____. Note, the five options not available from right-click *Tools* include: 1) *Set Levels*, 2) *Deselect All*, 3) *Select*, 4) *Select All Visible*, and 5) *Enter Point*.

Entities – The right-click *Entities* option opens an *Entities* sub-menu with a list of commands functions that are identical to those available from the *Entities* option on the menu bar. For more information about the *Entities* menu, see the section within this document entitled _____.

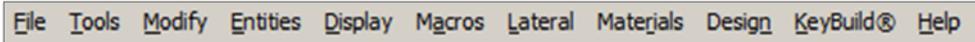
Display – The right-click *Display* option opens a *Display* sub-menu that contains a subset of the commands available from the *Display* option on the menu bar. The commands available from the right-click *Display* sub-menu include:

1. Zoom Window
2. Zoom Full
3. Zoom Last
4. Show Wall Profiles
5. Show Planes & Overhangs

Macros – The right-click *Macros* option opens an *Macros* sub-menu with a list of commands functions that are identical to those available from the *Macros* option on the menu bar. The single exception is the *Re-Elevate Roof Members* function, which is not available from the right-click *Macros* sub-menu. For more information about the *Macros* menu, see the section within this document entitled _____.

Set Properties Mode – The *Set Properties* command allows users to view and change the properties of a selected member. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the entity that you want to modify. Select *Set Properties* from the *Modify* menu. The property window for the selected entity type will appear. Review or change any information necessary, then click on the *OK* button.

The GS Plan Menu Bar & Toolbars



GS Plan toolbars contain a subset of the most commonly used commands and functions available from the GS Plan menu bar. Taken collectively, the menu bar contains all the commands and functions available within the GS Plan application. The available listings on the menu bar change, based upon where the user is within the application and the functions he or she is performing at any given time.

THE FILE MENU & TOOLBAR

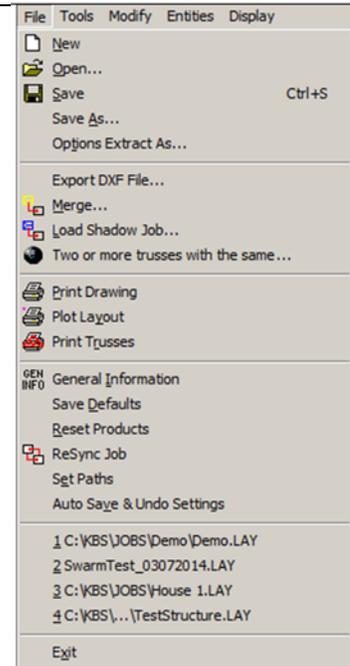
The *File* menu and the *File* toolbar provide users with commands for opening, saving, merging, extracting, exporting, and printing GS Plan files. They also provide commands for accessing a variety of file-related presets.

 **New** – When you select *New* from the *File* menu, or click the *New* button when first starting GS Plan, you will be prompted to enter default information for the job that you are beginning. The first window to appear is the *General Information* window. Fill in any default information necessary, and then click on the *OK* button. This will bring up the *Set Levels* window. Define the level that you want to start working on and then click on the *OK* button. (Note: Define only the level that you want to work on, since empty levels do not save.)

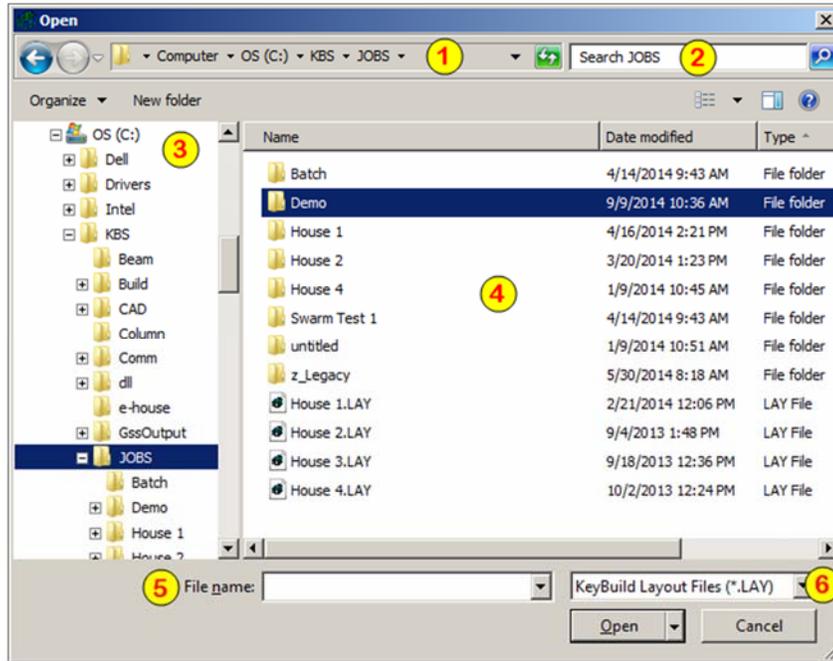
If you have a model open when you start a new model, you will be prompted to save the existing model before a new model is started.

 **Open** – When you select *Open* from the *File* menu, or when first starting GS Plan, you will be prompted to select the model that you want to open, using the *Open File* dialog box. You can use this option to open either .lay files or .may (options) files. Use the *Open File* dialog to locate and select the model that you want to open, and then click on the *Open* button. When a model is first opened, all levels will be visible and the lowest level will be active. Once you have a model open, and attempt to open a different model, you will be prompted to save the current model.

Note: To open a job that contains *Options*, change the *File Type* from *GS Plan Layout Files* to GS Plan **Base House** files and then select the .MAY file you wish to open. The preview box will display the base house plus all options – even though simultaneous option selection may be structurally impossible. Don't worry about this slight exaggeration. The preview is based upon snap-points stored with the model and does not differentiate between base house and option entities.



The *Open File* window is shown below. Read below for more information.



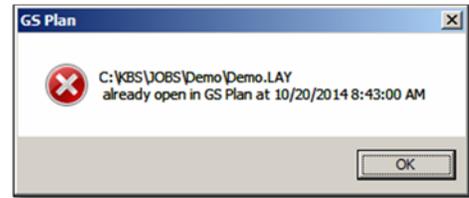
1. **File Path** – This field, at the top of the *Open* window, displays the full path to the folder selected below in the *Navigation Pane*. You can use the *File Path* field to navigate to a new location in the folder hierarchy, or you can navigate to a new path using the *Navigation Pane*. (See below.)
2. **Search Field** – This field is used to search for files and/or folders within the currently selected folder and its sub-folders. Simply type in all or part of the file or folder name you are seeking, and then click on the *Search* icon to the right of this field to initiate your search.
3. **Navigation Pane** – As its name implies, the *Navigation Pane* is used to navigate to the desired folder within your directory structure.
4. **Details Pane** – The *Details Pane* displays the contents of the folder currently selected in the *Navigation Pane*.
5. **File Name** – When you click on a file in the *Details Pane*, its name is displayed here in the *File Name* field. You can also type the name of the desired file into this field, or select the file from the field's drop-down list.
6. **File Type** – You can choose a specific File Type to search for from this field's drop-down list. The list of available options includes:
 - *All Files (*.*)*
 - *GS Plan Layout Files (*.LAY)*
 - *GS Plan Layout Files (*.MAY)*

Command Buttons

Open – Opens the selected file.

Cancel – Cancels the *Open* process and closes the *Open* window.

Note: GS Plan programs now check to see if the job you are attempting to open is already open by another user. If it is determined that the job you are attempting to open has already been opened by someone else, you will get an error message and will not be able to open the job. If it is determined that you have the job open and are attempting to open it again, you will be warned and then, if you click *OK*, the program will open the job. Please note that having a GS Plan job open by multiple users will cause file corruption, and could result in the loss of all job information.

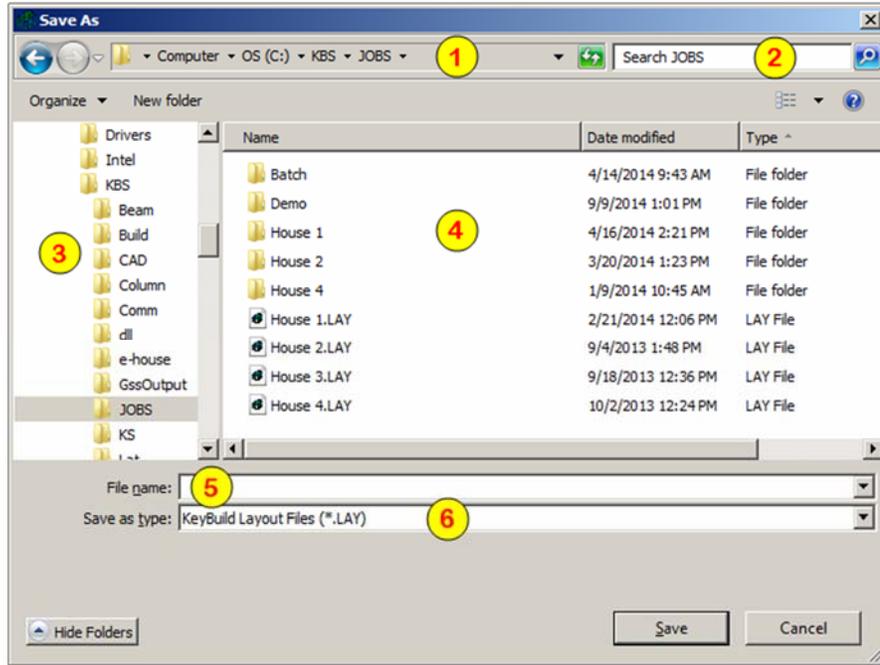


Save – Select this command from the *File* menu to save the changes to the job that is currently open. If you have never saved this model before, you will be prompted by the *Save As* window to define a name. (See below for more information about the *Save As* window).

Shortcut: Press the *Control* and *S* keys on your keyboard at the same time.

Save As – The *Save As* command will bring up the *Save As* window, allowing you to specify a name and directory for the current job. Jobs that do not contain options will be saved as GS Plan layout files (.Lay files). If you are saving a job that contains options, change the *File Type* to *GS Plan Base House* files, so that the job will be saved as a .may file).

The *Save As* dialog box is displayed below.



1. **File Path** – This field, at the top of the *Save As* window, displays the full path to the folder selected below in the *Navigation Pane*. You can use the *File Path* field to navigate to a new location in the folder hierarchy, or you can navigate to a new path using the *Navigation Pane*. (See below.)
2. **Search Field** – This field is used to search for files and/or folders within the currently selected folder and its sub-folders. Simply type in all or part of the file or folder name you are seeking, and then click on the *Search* icon to the right of this field to initiate your search.
3. **Navigation Pane** – As its name implies, the *Navigation Pane* is used to navigate to the desired folder within your directory structure.
4. **Details Pane** – The *Details Pane* displays the contents of the folder currently selected in the *Navigation Pane*.
5. **File Name** – Use this field to type the name you wish to use when saving your file.
6. **Save as Type** – Select the File Type you wish to use when saving your file. The list of available options includes:
 - *All Files (*.*)*
 - *GS Plan Layout Files (*.LAY)*
 - *GS Plan Layout Files (*.MAY)*

Command Buttons

Save – Click on the *Save* button to save your file as the specified *File Type* and to the specified location.

Cancel – Click on the *Cancel* button to close the *Save As* window without saving your file.

Options Extract As – The *Options Extract As* command allows users to extract .lay files from a job that contains options (.may files). This option operates in conjunction with the *Select Options* command on the *Tools* → *Options* sub-menu, to generate a set of .lay files containing the base house plus all selected options. (If no options have been selected, only base house entities will be extracted). This command is used whenever you want to export an options job to one of GS Plan's downstream modules (such as Structure, which cannot analyze a .MAY file).

To use this command, first use the *Select Options* command to select the options you wish to export into a .lay file. Then choose the *Options Extract As* command from the *File* menu. A dialog box similar to the *Save As* screen will appear. (See above for an explanation of each field). Choose a file name and click on the *Save* button to generate .lay and level files based on the base house and options currently selected.

Export DXF File – This command allows you to export the current layout as a DXF file.



Merge – Select *Merge* from the *File* menu to import a single level of an existing job into the current one. After selecting the command, the *Open File* window will open. Use this window to locate the file that you want to merge into the current job, and then click on the *Open* button. (For more information on the *Open* window see the *Open* help above). The level selected will be placed into the active level, but will retain its original height (the top of plate height at which the level was input) as well as its original location.

All of the members on the imported level will be highlighted after it is imported. This allows you to use the commands on the *Modify* menu to move the imported members to where they are needed.



Load Shadow Job – This command allows you to import a level of any existing layout as a shadow to be used for tracing. Choose the appropriate level (jobname.l##) from the *Open File* window and click *Open*.



GS Build –



Re-Synchronize the GS Wall-Side Data with the GS Plan-Side –



Print Drawing – Selecting *Print Drawing* from the *File* menu will send a printout of the view in the *Drawing Area*.



Plot Layout – Select this option from the *File* menu to plot the layout of the currently active level in *Plot*. When you choose this option, GS Plan opens *Plot* with your floor layout, a material list, and title box. The labels drawn with the individual floor members correspond with the material list.

Note: The labels will change if you make changes to your job and regenerate the plot. This feature is intended for quick bid and quick output, for NON-ENGINEERED layouts.



Print Trusses – This command is used to print a *Truss Profiles Report*.



General Information – Selecting *General Information* from the *File* menu opens the *General Information* window. This window is used to set up default information for the current job. It is the same window that appears when you start a new job. For comprehensive information about General Information settings, please see the section of this manual entitled “*General Information Tabs & Settings*” on page ___.

Save Defaults – This command stores the layout and information on the main window to be used the next time you start GS Plan. Selecting the *Save Defaults* command from the *File* menu will store the current settings of the following to be used each time GS Plan is run:

- Snap modes (Z, P, G, and O)
- Snap grid settings
- Current color settings
- Current entity type
- Current view (animated or plan)

Reset Products – Selecting *Reset Products* from the *File* menu will set the product list for all design materials to the default (defined in the *General Information* window (see above)).



ReSync Job – The *ReSync Job* command is used to re-synchronize panel-side data with model-side data within GS Plan. Users can elect to force panel updates (which updates all of the sticks/wall studs for a panel each time an update is performed) or, they can process panel updates as a group by clicking *ReSync Job*. This latter action determines what panels have been modified and leaves any custom framing on all walls, except for those that are out of sync with the model side. Walls that have been modified or added to the model-side are reframed.

Set Paths – The *Set Paths* command allows users to specify the location of the KeyMan executable file (usually C:\KMW\Man or C:\KMS\Man). This is the location from which GS Plan accesses header, skylight, post and geometric categories. Set this path to the location of your KeyMan directory (by typing it in or using the *Browse* button to find the *Man* directory) and then click on the *OK* button.

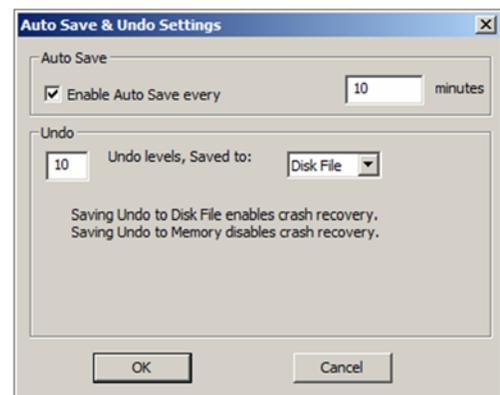
Note: Even if you store your Inventory and KeyMan jobs on a network, you must point this path at your local *Man* directory – not the server location.

Auto Save & Undo Settings – Clicking on this command opens the *Auto Save & Undo Settings* window pictured at right.

Enable Auto Save Every – When this checkbox is activated, GS Plan will automatically save your open files according to the timeframe defined in the adjoining *Minutes* field.

Minutes – This field defines how often the *Auto Save* function is executed. If the *Enable Auto Save* checkbox is deactivated (unchecked), then the *Minutes* field is grayed out and uneditable.

Undo – The *Undo* field defines how many consecutive user actions can be cancelled (undone). **The maximum number of actions that can be undone is ___.**



Undo Levels Saved To – The drop-down list in the *Undo Levels Saved To* field provides two options:

Disk File – If this option is selected, undo levels are saved to the user’s local hard drive. This is considered non-volatile storage, and the *Undo* information will be retained even if the user’s computer loses power.

Memory – If this option is selected, undo levels are saved to the user’s Random Access Memory (RAM). This is considered volatile storage, and the *Undo* information will be lost if the user’s computer loses power.

Exit – Choose this option to close GS Plan. If you have made changes to the current layout you will be prompted to save it before the program closes.

THE TOOLS MENU & TOOLBAR

The *Tools* toolbar provides users with commands to...



Point Snap – The *Point Snap* icon is used to toggle the *Point Snap* function on and off. If *Point Snap* is turned on, the user can snap to the XY coordinates of existing snap points within a model. In this mode, when the crosshairs of a user's pointer move past an existing snap-point, a rubber-band line (a line running from the center of the crosshairs to the nearest snap point) will appear. If you click the left mouse button while the rubber-band line is visible, your cursor will jump to the exact XY location of that snap point. If *Point Snap* is not turned on, the rubber-band line will not appear regardless of how close you are to the snap-point. You will not be able to get your crosshairs to the exact location of the point.

It is important to note that you stay in the current Z plane when using *Point Snap*. If you are snapping to a point that is at a different height, the crosshairs will move only to that point's XY coordinates, while staying at the current Z height. If you want to jump to the X, Y, and Z location of a point, use the *XYZ Snap* tool described below.

You can tell if *Point Snap* tool is turned on or off in two ways. If the icon to the left of the command name on the *Tools* menu is highlighted, then *Point Snap* is on. **The second is to look at the status bar. If a P is displayed in the lower right, then *Point Snap* is on.**

Shortcut: Press the *Control* and *P* keys on your keyboard at the same time. This will toggle *Point Snap* on and off.

Note: *Point Snap* cannot be on at the same time as *XYZ Snap*. Only one of the two can be active at a time.



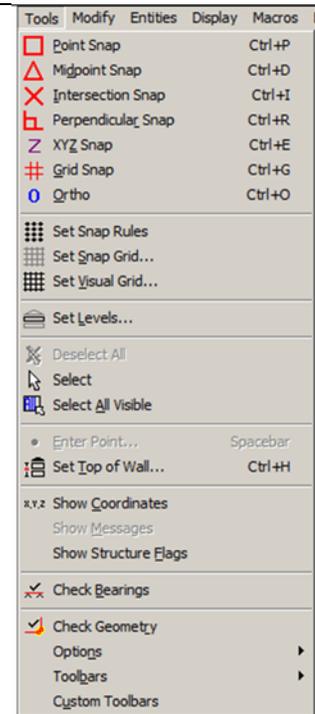
Midpoint Snap – When activated, the *Midpoint Snap* function creates snap-points at the center of each member and at the middle of each square of the *Snap Grid*. If *Midpoint Snap* is turned on, you can snap to the XY coordinate of existing snap-points in your model. In this mode, when you move your crosshairs past an existing snap-point, a rubber-band line appears. A rubber-band line is a line that runs from the center of the crosshairs to the nearest midpoint snap-point. Click the left mouse button while the rubber-band line is visible, and the cursor moves to the exact XY location of that midpoint snap-point.

You must stay in the current Z plane when *Midpoint Snap* is on. If you snap to a point that is at a different height, the crosshairs move to that point's XY coordinates, while staying at the current Z height. If you want to move to the XYZ location for a point, use the *XYZ Snap* command from the *Tools* menu.

If *Point Snap* is turned off, the rubber-band line does not appear and you cannot get your crosshairs to the exact location of the point.



Intersection Snap – When activated, the *Intersection Snap* function creates snap-points at the every point member intersects another member. It also creates a snap-point at each location where a vertical line of the snap grid intersects a horizontal line of the snap grid.



In this mode, when you move your crosshairs past an existing snap-point, a rubber-band line appears. A rubber-band line is a line that runs from the center of the crosshairs to the nearest intersection snap point. Click the left mouse button while the rubber-band line is visible, and the cursor moves to the exact XY location of that intersection snap point.

You must stay in the current Z plane when Intersection Snap is on. If you snap to a point that is at a different height, the crosshairs move to that point's XY coordinates, while staying at the current Z height. If you want to move to the XYZ location for a point, use the *XYZ Snap* command from the *Tools* menu.

If *Intersection Snap* is turned off, the rubber-band line does not appear and you cannot get your crosshairs to the exact location of the point.



Perpendicular Snap – If *Perpendicular Snap* is turned on, you can snap to the XY coordinate of existing snap-points in your model. In this mode, when you move your crosshairs past an existing snap point, a rubber band line appears. A rubber-band line is a line that runs from the center of the crosshairs to the nearest intersection snap-point. Click the left mouse button while the rubber-band line is visible, and the cursor moves to the exact XY location of that perpendicular snap-point.

You must stay in the current Z plane when *Perpendicular Snap* is on. If you snap to a point that is at a different height, the crosshairs move to that point's XY coordinates, while staying at the current Z height. If you want to move to the XYZ location for a point, use the *XYZ Snap* command from the *Tools* menu.

If *Perpendicular Snap* is turned off, the rubber-band line does not appear and you cannot get your crosshairs to the exact location of the point.



XYZ Snap – The *XYZ Snap* icon is used to toggle the *XYZ Snap* function on and off. If *XYZ Snap* is turned on, the user can snap to the XYZ coordinates of existing snap-points in a model. In this mode, when the crosshairs of a cursor move past an existing snap-point, a rubber-band line (a line running from the center of the crosshairs to the nearest snap-point) will appear. If you click the left mouse button while the rubber-band line is visible, your cursor will jump to the exact XY location of that snap-point. If *XYZ Snap* is not turned on, the rubber-band line will not appear, regardless of how close you are to the snap-point. You will not be able to get your crosshairs to the exact location of the point.

It is important to note that you will not necessarily stay in the current Z plane when in *XYZ Snap*. If you are snapping to a point that is at a different height, the crosshairs will move to that height. (This can cause problems if you are using *XYZ Snap* while viewing a layout from above, as it is impossible to see changes in the Z direction). If you want to jump to the XY coordinates, and preserve the current Z height, use the *Point Snap* tool instead.

You can tell if *XYZ Snap* is on or off one of two ways. The first is by looking in the *Tools* menu at the *XYZ Snap* command. If the icon to the left of the command name on the *Tools* menu is highlighted, then *XYZ Snap* is on. **The second is to look at the status bar. If a Z is displayed in the lower right, then *XYZ Snap* is on.**

Shortcut: Press the *Control* and *Z* keys on your keyboard at the same time. This will toggle the *XYZ Snap* on and off.

Note: *XYZ Snap* cannot be on at the same time as *Point Snap*. Only one of the two can be active at any given time.



Grid Snap – The *Grid Snap* icon is used to toggle the *Grid Snap* function on and off. If *Grid Snap* is turned on, the user can snap to the points on the grid defined in the *Set Snap Grid* window. In this mode, as your cursor moves around the *Drawing Area*, your X and Y coordinates will jump by the increment defined in the *Set Snap Grid* window. If you click the left mouse button, your cursor will jump to the nearest grid-point. (The XY coordinates that are displayed in the status bar at the time). If *Grid Snap* is not turned on, the cursor will not be restrained by the grid, but will move freely by increments of a sixteenth of an inch.

You can tell if *Grid Snap* is on or off one of two ways. The first is by looking in the *Tools* menu at the *Grid Snap* command. If the icon to the left of the command name on the *Tools* menu is highlighted, then *Grid Snap* is on. **The second is to look at the status bar. If a G is displayed in the lower right, then *Grid Snap* is on.**

Shortcut: Press the *Control* and *G* keys on your keyboard at the same time. This will toggle *Grid Snap* on and off.



Ortho – The *Ortho* icon is used to toggle the *Ortho* function (short for orthogonal) on and off. When *Ortho* mode is active, users are restricted to drawing lines at 90-degree angles. (That is to say, lines can only be drawn straight up, straight down, straight left, or straight right).

You can tell if *Ortho* is on or off one of two ways. The first is by looking at the *Ortho* command on the *Tools* menu. If the icon to the left of the command name on the *Tools* menu is highlighted, then *Ortho* is on. **The second is to look at the status bar. If an O is displayed in the lower right, then *Ortho* is on.**

Shortcut: Press the *Control* and *O* keys on your keyboard at the same time. This will toggle *Ortho* on and off.



Set Snap Point Rules – Clicking on the *Set Snap-Point Rules* icon opens the *Set Snap-Point Rules* dialog box pictured at right. This dialog box contains the following parameters:

- **Enable Framing Member Snap Points** – **When this checkbox is activated, GS Plan...**



BOTTOM SNAP POINTS

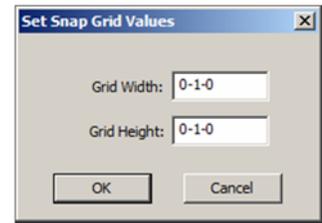
- **Enable for Walls & Posts** – **When this checkbox is activated, GS Plan...**
- **Enable for Beams** – **When this checkbox is activated, GS Plan...**

COMMAND BUTTONS

- **OK** – Click on the *OK* button to save your changes and close the *Set Snap-Point Rules* dialog box.
- **Cancel** – Click on the *Cancel* button to discard your changes and close the *Set Snap-Point Rules* dialog box.
- **Save Defaults** – Click on the *Save Defaults* button to save the current settings as your default settings for snap-point rules.
- **Restore Defaults** – Clicking on the *Restore Defaults* button cancels any changes made to the three checkboxes in this dialog box, and restores the checkboxes to their default state.

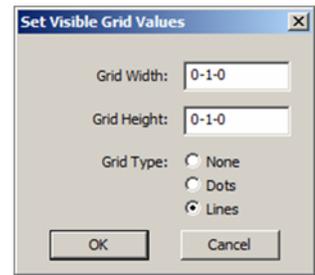
Note that, once a user clicks on the *Save Defaults* button, the *Restore Defaults* button cannot be used to return these checkboxes to their previous default state.

Set Snap Grid – The *Set Snap Grid* command allows you to define the grid-points used by the *Grid Snap* option. When you select *Set Grid Snap* from the *Tools* menu, the *Set Snap Grid Values* window appears. This window is used to set the size of the snap grid used by *Grid Snap*.



There are two values to be entered into the *Set Snap Grid Values* window; the *Grid Width* and the *Grid Height*. The *Grid Width* increment will establish grid lines running vertically across the *Drawing Area*. *Grid Height* establishes the horizontal grid lines. Every time two grid lines cross, a grid-point or snap-point is created.

Set Visual Grid – The *Set Visual Grid* command allows users to define the size of the grid displayed in the *Drawing Area*. When you select *Set Visual Grid* from the *Tools* menu, the *Set Visual Grid Values* window appears. This window is used to set the size and type of the visual grid being displayed.



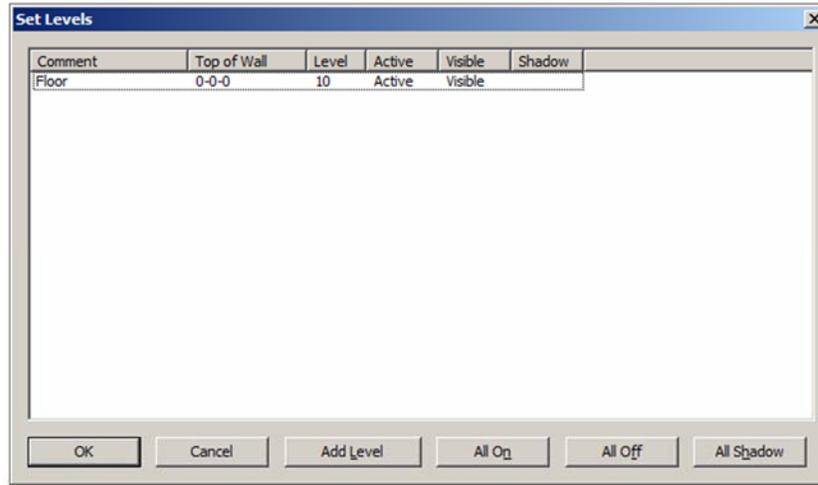
It is important to note that the values for the *Visual Grid* are completely separate from the *Snap Grid*, established in the *Set Snap Grid Values* window. Your visual grid and your snap grid do not need to be the same.

There are three values that can be entered into the *Set Visual Grid Values* window:

- **Grid Width** – Establishes grid lines running vertically across the *Drawing Area*.
- **Grid Height** – Establishes the horizontal grid lines.
- **Grid Type** – There are three different grid types that can be used:
 - Dots* – When *Dots* is selected as the *Grid Type*, a small dot will be displayed at each grid line intersection.
 - Lines* – When *Lines* is selected as the *Grid Type*, each of the grid lines will be displayed.
 - None* – When *None* is selected as the *Grid Type*, the grid will not be displayed.

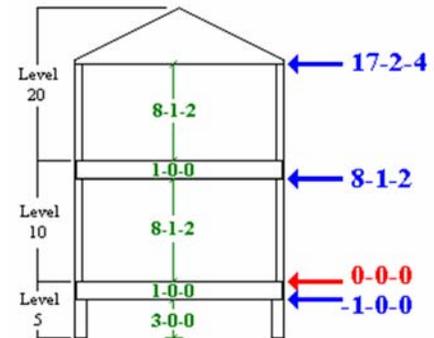


Set Levels – The *Set Levels* command allows users to create new levels, and to view and edit information about all the levels in the current model. See below for more information on each field.



Comment – The *Comment* field is used to assign a name to each level. To edit the name in the *Comment* field, click once on the name and then type in the new level name.

Top of Wall – This is the Z height of the top of the bearings in the level. The picture above shows a section of a two-story house that is divided into three levels. Z dimensions are shown in green. The *Top of Wall* for each level is shown in blue. These *Top of Wall* heights are based on a 0-0-0 point. (In this case, the top of the first-floor joists in red.)



Level – The *Level* field displays the identifying number assigned to each level in a structure. Before a house can be input, you must divide it into levels. Generally, each level should consist of framing members (roof or floor), and the bearings that support them. After you have determined how your structure will be divided into levels, you need to assign a number to each level. In the example above, the house is broken into three levels. When numbering levels, you must go from top to bottom. (The highest level should have the largest number; the lowest level the smallest number).

Notice that the assigned level numbers go from top to bottom. The upper level has the largest number, since it is on top. The middle level has the next largest, and the lower level has the smallest number. The order that you assign numbers to levels is very important, as Structure™ will transfer the load through the building from the level with the largest number down to the level with the smallest number. In this case, roof loads will be developed (level 20) and then passed through the second story walls to the second-story floor (level 10). The loads will then pass through the first story walls to the floor, and then into the basement (level 5). If level numbers were numbered backwards, then the load would originate in the basement and be passed up through level 10 and 20.

Notice that the levels are not numbered in increments of 1. There are several empty levels between the upper and middle levels, and again between the middle and lower levels. This is

a recommended practice, as it may be necessary at a later time to add a load transfer member between levels.

Active – The level that has *Active* displayed next to it is the level that currently being worked on. You can only input or edit items on the active level. For this reason, only one level can be active at a time. When you input on the *Active* level, you will be inputting at the *Top of Wall* for that level. To turn a level to *Active*, click in the *Active* column to the right of the level number.

To quickly make a level both *Active* and *Visible*, double click in the *Active* column next to that level. This will remove the *Visible* or *Shadow* flags from all other levels, and make only the one you double-clicked on *Active* and *Visible*.

Visible – If a level is *Visible*, it is displayed in full color and can be edited. To turn a level *Visible* (or turn *Visible* off), click in this field to the right of the level number. A level can have *Visible* or *Shadow* on one at a time. If neither is on, the level is invisible and cannot be seen or edited.

Shadow – If a level is set to *Shadow*, it is displayed in the shadow color. (The default *Shadow* color is dark blue). This level cannot be edited, but the snap-points are usable. To shadow a level (or turn shadow off), click in this field to the right of the level number. A level can have *Visible* or *Shadow* on one at a time. If neither is on, the level is invisible and cannot be seen or edited.

COMMAND BUTTONS

OK – Click on the *OK* command button to save any changes made and return to the main GS Plan window.

Cancel – Click the *Cancel* button to ignore any changes made and return to the main GS Plan window.

Add Level – Click the *Add Level* button to create a new level. You will need to enter a level number before you can leave the *Set Levels* window. (Only available level numbers will be displayed in the drop-down list). When you return to the main window, the new level will be completely empty. (Levels cannot be saved with the model until at least one entity is entered.)

All On – Click the *All On* button to make all levels in the current job visible.

All Off – Click the *All Off* button to turn the *Visible* and *Shadow* flag off for all levels.

All Shadow – Click the *All Shadow* button to set all levels in the current job to *Shadow*.

Deselect All – The *Deselect All* command allows you to deselect all selected (highlighted) items in the model. (In other words, no items in a model will be selected after this command is run.)

Select – The *Select* command puts you into Select Mode where you can select items to change, edit, or delete. Choose this option and then click on an entity.

Select All Visible – Choose the *Select All Visible* option from the *Tools* menu to highlight every member on every visible level. (See the *Set Levels* help for more information on visible levels).



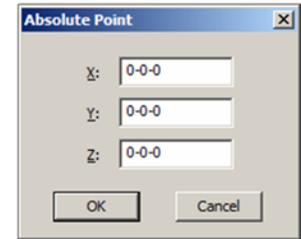
Enter Point – The *Enter Point* command allows users to move the crosshairs to any XYZ location. Selecting the *Enter Point* command from the *Tools* menu will bring up the *Absolute Point* window, which allows you to specify the exact coordinates of a point where you want your cursor to go. This command is most commonly used when inputting entities. If you are in an input mode when this command is run, the cursor will move to and pick the point entered, as if you had picked the point with the left mouse button. Shortcut: Press the *P* key or the *spacebar*.

X – Enter the X coordinate of the point to which you want the cursor to jump. The default value displayed in this field is the coordinate that your cursor was at when the command was run.
Note: No entities should be input in negative space, so do not input a negative number in this box.

Y – Enter the Y coordinate of the point to which you want the cursor to jump. The default value displayed in this field is the coordinate that your cursor was at when the command was run.

Note: No entities should be input in negative space, so do not input a negative number in this box.

Z – Enter the Z coordinate of the point to which you want the cursor to jump. The default value displayed in this field is the coordinate that your cursor was at when the command was run.



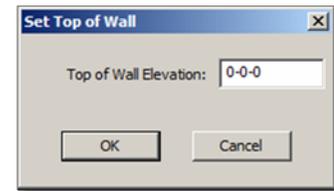
COMMAND BUTTONS

OK – Click on the *OK* command button to move your cursor to the point defined by the coordinates and select it.

Cancel – Click on the *Cancel* button to return to the main GS Plan window without moving your cursor.



Set Top of Wall – The *Set Top of Wall* command allows users to establish a default Z height for input. When you select *Set Top of Wall* from the *Tools* menu, a dialog box like the one pictured at right will open. *Set Top of Wall* dialog box appears. Enter the desired height in the *Top of Wall Elevation* field, and then click on the *OK* command button to apply your new setting.



Shortcut: Press the *Control* and *H* keys on your keyboard at the same time. Below is the *Set Top of Wall* window. See below for more information on each field.



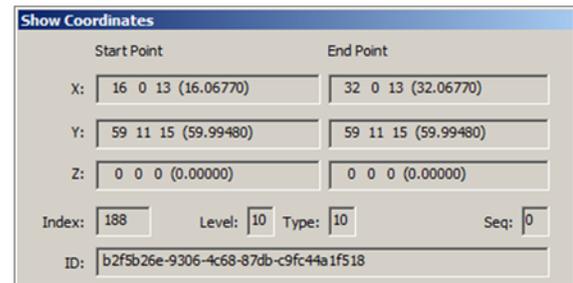
Show Coordinates – The *Show Coordinates* command allows users to view the XYZ coordinates of a selected member. To use this command, select *Show Coordinates* from the *Tools* menu. Your cursor will change into an arrow pointer. Move the tip of the arrow next to the member that you want to see information about, and then click your left mouse button. The *Show Coordinates* window will appear. After reviewing the information in the window, click the *Cancel* button to return to the main window. At this point you are still in the command. Click on any other members for which you need information. Click the right mouse button to exit the command. Below is the *Show Coordinates* window. See below for more information on each field.

START POINT

X – This is the X coordinate of the start point (or beginning point) of the selected line.

Y – This is the Y coordinate of the start point (or beginning point) of the selected line.

Z – This is the Z coordinate of the start point (or beginning point) of the selected line.



END POINT

X – This is the X coordinate of the endpoint of the selected line.

Y – This is the Y coordinate of the endpoint of the selected line.

Z – This is the Z coordinate of the endpoint of the selected line.

Index – As each member is input, it is assigned an index number. This number serves as an ID for the members. Note: Each time a model is altered, or a member deleted, index numbers are reassigned.

Level – This is the level on which the selected member resides.

Type – This is the type of entity that is currently selected. There are 11 potential values:

- 0 – Construction Line
- 1 – Bearing
- 2 – Area
- 3 – Opening
- 4 – Load (Point, Line, or Area)

- 5 – Plane
- 6 – Overhang
- 7 – Gable
- 8 – Hip End Line
- 10 – Framing Member
- 11 – Hip End Member

Seq – If the entity selected is a compound entity (members that are made up of several lines joined together, such as areas, planes, or openings), this field will display which line in the entity was selected. Note: Like the *Index* number, the *Seq* number can change due to alterations to the entity.

ID – The read-only ID field displays...

To close the *Show Coordinates* window, hit the escape key on your keyboard.

Show Messages – If a job encounters problems when it is run through Structure™ (i.e. errors, warnings, or constraints), the *Show Messages* command will become available in the *Tools* menu. Selecting *Show Messages* allows you to view the messages that were generated by Structure. The messages will be displayed in the *Structure Messages* box. There are three buttons at the top of the box:

- *Print* – Click this button to print a copy of the message list.
- *Find* – Click this button to search for a word or phrase in the message list.
- *Close* – Click this button to close the *Show Messages* box.

In the box, a list of all messages for the current job will be listed. There are three different types of messages:

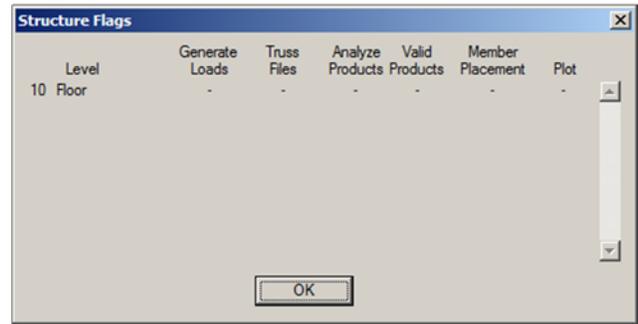
Warnings – A warning is a small problem in the model that you should be made aware of, but that is not a big enough problem to prevent the continued design of your structure.

Errors – An error is a structural problem in the model. This means that there is something wrong with the model that is causing it to not be designed by Structure™. Errors must be fixed before the job will run through Structure.

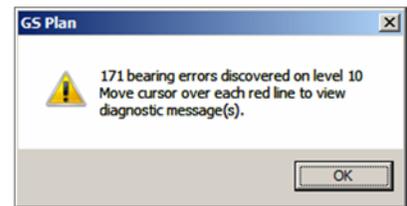
Constraints – A constraint is caused when a model presents Structure™ with a design problem that it cannot solve, or exceeds the limits of Structure’s abilities. Constraints must be fixed before a job will run through Structure.

Show Structure Flags – Clicking on the *Show Structure Flags* listing on the *File* menu opens a *Structure Flags* dialog box like the one pictured at right. This dialog box contains the following attributes:

- **Level** –
- **Generate Loads** –
- **Truss Files** –
- **Analyze Products** –
- **Valid Products** –
- **Member Placement** –
- **Plot** –



Check Bearings – The *Check Bearings* command is used to make sure that each member on the active level has at least two supports. Selecting *Check Bearings* from the *Tools* menu will check the members on the current level. If a member fails the bearing check, it will be highlighted in red and an error message, like the one pictured at right, will be displayed. As long as *Check Bearings* is on, moving the cursor over a red member will display a short message. One of the following messages will be displayed:



- *Fewer than two bearings* – If this message is displayed, it is because the highlighted member has only one, or zero, members supporting it. To fix this problem, add supports to the member, then run *Check Bearings* again.
- *Too many supports* – This message will appear when a member has more than 10 supports. To fix this problem, either remove some of the supports, or break the problematic member into multiple parts.
- *Bearings cross each other* – If bearing or framing members run through each other, you will receive this message. Break one of the members at the point of intersection.
- *Indeterminate structure* – This message appears if members bear improperly upon each other.
- *Conflicting hipends* – If you have two or more sets of hip-end framing interfering with each other, you will receive this message. You may need to move a hip-end line or area to fix this problem.
- *Conflicting bearings* – This message appears if you have two or more of the same bearing type occupying the same space. At least one bearing will have to be moved or changed to fix this problem.

If a member has more than one of these errors, the error names will be displayed in the pop-up separated by a semi-colon (e.g. *Conflicting bearings; Indeterminate structure*). Once a member is highlighted it will remain that way until *Check Bearings* is turned off (selecting *Check Bearings* a second time will turn it off). *Check Bearings* will not re-check the job unless it is run a second time (turned off then on again).



Check Geometry – The *Check Geometry* command checks your model for points that nearly, but not exactly, coincide. It identifies and corrects points that are not multiples of 1/16" (or other specified tolerance value); however, it does not directly identify non-orthogonal lines, or lines that fail to connect with each other precisely.

To use this command, make sure the level you wish to test is active and visible. You should also disable the visibility of any entities you wish to exclude from testing. *Check Geometry* only works on visible lines within the level that is currently active. It will automatically exclude construction lines, framing members, and dimension lines, whether visible or not.

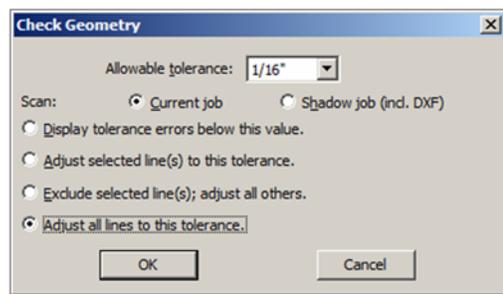
Note that, as soon as you click the *OK* button on the *Check Geometry* screen, the dialog box disappears, allowing you to view the entire *Drawing Area*. However, the command remains checked in the *Tools* sub-menu until all errors have been corrected, or until you re-select *Check Geometry* and click the *Cancel* button.

During the modeling process, there are times when it makes sense to use *Check Geometry*, and other times when it is not advisable. One very good time (for which, in fact, the feature was originally designed) is immediately after you have imported a DXF file as a shadow job. If slight geometric errors in a DXF file are allowed to remain when GS Plan entities are traced, those errors will be propagated throughout the entire model. Therefore, immediately after loading a DXF file, you should use *Check Geometry* to scan the file (as a shadow job) and correct slight errors that are too small to be detected with the naked eye.

Another good time to use *Check Geometry* is immediately after you have drawn all walls (and optionally headers), but before you have added any framing areas or roof planes. In general, the XY coordinates of ridge and valley lines will seldom coincide with even multiples of any specified tolerance. You do not want *Check Geometry* to adjust plane coordinates so that the resultant vertices are no longer co-planar.

Finally, you should consider using *Check Geometry* whenever you get the message "Cannot form exterior boundary" (typically when using the *Create Shear Lines* command or *Ceiling Planes* macro). This message can arise from one or more breaks between exterior walls and/or headers in your model which are too small for the eye to detect, but which *Check Geometry* can readily identify.

After selecting the *Check Geometry* option, the *Check Geometry* window (shown at right) will appear. See below for more information on each field.



Allowable Tolerance – This tolerance should normally be set to 1/16" or 1 mm (since these are the usual increments used in architectural drawings). However, you may also check tolerances to the nearest 1/8", 1/4", or whatever value makes sense within the context of your drawing.

SCAN

Current Job – When this radio button is selected, GS Plan checks the geometry of all visible entities on the level that is currently active.

Shadow Job (incl. DXF) – When this radio button is selected, GS Plan checks the geometry of a file that has been imported via *File*→ *Load Shadow Job*, including imported DXF jobs. Note that shadow jobs are not the same as shadowed levels, which always belong to the current job.

[ACTIONS]

Display tolerance errors below this value – When this option is selected, GS Plan finds all error lines (lines whose endpoints are not even multiples of the specified tolerance) without attempting to correct them. It first displays the number of error lines discovered. If no errors are discovered, a message to that effect is displayed.

After you click on the *OK* button, the error message will disappear and any lines with geometric errors will appear in red on your screen. For bearing members (such as walls or headers), only their reference lines will turn red. To view the XY coordinates of any red line, move the cursor close to it, and the coordinates will be displayed near the cursor.

Adjust selected line(s) to this tolerance – This option changes the XY coordinates of (both endpoints of) selected lines to the nearest 1/16", or other specified tolerance. As each error line is corrected, its display reverts to its native entity color. Line selection is terminated by pressing the *Escape* key on your keyboard, or clicking the right mouse button.

Exclude selected line(s); adjust all others – This option allows you to exclude one or more lines from adjustment. This strategy is useful for non-orthogonal lines (such as diagonal walls), whose endpoints you know cannot be even multiples of the designated tolerance. As each error line is excluded, its display reverts to its native entity color. Line exclusion is terminated by pressing the *Escape* button on your keyboard or clicking the right mouse button, at which point all non-excluded error lines will be adjusted.

Adjust all lines to this tolerance – This option changes the XY coordinates of all visible lines, on the level that is currently active, to the nearest multiple of the specified tolerance, without first displaying them in red. You should be careful not to use this option indiscriminately, or you can potentially corrupt the model beyond easy restoration.

COMMAND BUTTONS

OK – Click on the *OK* button to save your changes and close the *Check Geometry* dialog box.

Cancel – Click on the *Cancel* button to discard your changes and close the *Check Geometry* dialog box.

Options – The *Options* commands allows users to create and edit plan options, and to select the options to be displayed on-screen. Jobs that contain options are saved as .MAY and .M** files (instead of the usual .LAY and .L** files). MAY files cannot be run in Structure, however, so you must use the *Options Extract As* command on the *File* menu if you need to run the job through Structure.

Before drawing any options, you will ordinarily construct a base house, in order to give yourself a frame of reference for the options. Drawing a base house is exactly the same as drawing any other model in GS Plan. Entities (bearings, planes, framing areas, etc.) not specifically associated with an option belong to the base house by default. You may want to save the base house as either a ".LAY" or ".MAY" file, before proceeding with creating the options.

When you select *Options* from the *Tools* menu, a sub-menu opens with five listings:

1. Create/Edit Option
2. Select Options
3. Copy/Xfer Option
4. Copy Selected Entities
5. Option Materials

Create/Edit Option

To create a new option, click on *Tools* → *Options* → *Create/Edit Option*. The screen pictured at right will appear.

Select/Enter New Option – If you are creating a new option, type a name for the option in the *Select/Enter New Option* field. If you select an existing option from the pane immediately below this field, the name of the selected option will be displayed in this field.

Option names may be up to 40 characters in length and may contain spaces. It is recommended that you use names that are reasonably suggestive of the option content, so that you (or someone else) can readily identify them from a list of available options. As soon as you type one or more characters of the option name, the *Clear Selection*, *Add/Modify Option* and *Draw Option* command buttons will be activated.

Option names are stored with a job when it is saved. Therefore, whenever you add a new option name (or modify the spelling of an existing name) and click on *Add/Modify Option*, the job will have been changed – even though you may not have drawn any new entities. If you subsequently exit the job without saving it, you will be prompted to save these changes.

Exclusions – To eliminate the possibility that a user might select or extract conflicting options, the *Create/Edit Option* dialog box allows you to identify "mutually exclusive" options. From the upper list box, select any one of the conflicting options. Then in the lower list box (Exclusions), select those options you wish to exclude. Finally, click on the *Modify/Exclude* button. Thereafter, in the *Select Options* dialog box, it will no longer be possible to select conflicting options.

It is not necessary to exclude options reciprocally. That is, if Option A excludes options B and C, Option B will automatically exclude A and C, and Option C will automatically exclude A and B. To reverse the exclusion process, simply re-select one of the excluded options in the upper list box and de-select the other one(s) in the lower list box; then click again on the *Modify/Exclude Option* command button.

Do not confuse excluded options with "excluded base house entities." (That is, base-house entities that have been deleted from a particular option. The two concepts have nothing to do with one another.)

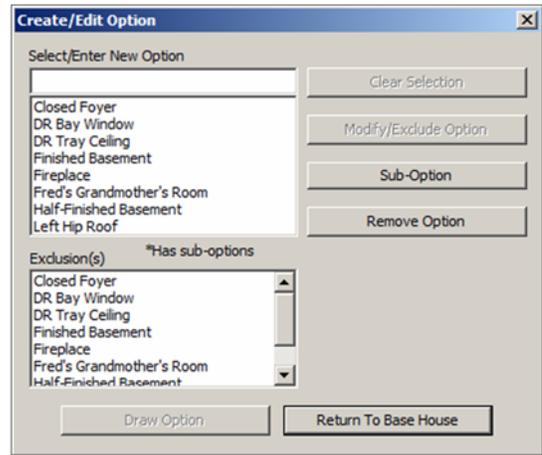


Figure 1: The *Create/Edit Option* window.

COMMAND BUTTONS

Clear Selection – *Clear Selection* erases the name entered into the *Select/Enter New Option* field.

Add/Modify – Copies the name to the two list boxes below.

Sub-Option –

Remove Option – Clicking on the *Remove Option* button will delete the selected option from the list of available options.

Draw Option – The *Draw Option* returns control to the *Drawing Area*, where you may begin drawing entities belonging to that option. It is not necessary to draw each option as you name it. If you are modeling a base house with several options, you may type each name and click *Add/Modify Option* at the outset.

Option names will always be listed in alphabetical order, regardless of the sequence in which they were entered. To select a particular option to draw (or modify), click on the corresponding name in the upper list, and then click on the *Draw Option* button.

Whenever you leave the *Create/Edit Option* dialog box via the *Draw Option* button, you will be in *Draw Option* mode. The specific option being drawn is the one that was displayed at the top of this box. You can return to the base house at any time by clicking the *Exit Options* button that appears at the bottom of the *Classic Sidebar*. (Alternatively, you can return to the *Create/Edit Option* dialog box and click on the *Return To Base House* button.)

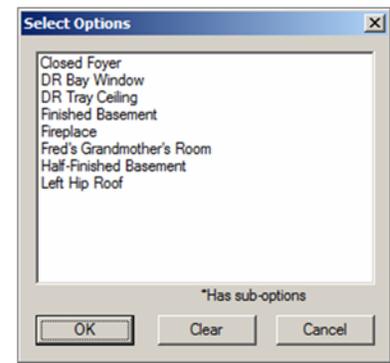
For additional information about options, see the section within this document entitled “*Working with Options*.”

Return To Base House – Returns the user to the base house. (That is the house without any options or sub-options.)

Select Options

To select one or more options simultaneously, click on *Tools* → *Options* → *Select Options*. A *Select Options* dialog box like the one pictured at right will open. Only those options for which entities have actually been drawn will be displayed on this screen. (As compared with the *Create/Edit Option* screen, which also displays options that have been named but not yet drawn.)

Select a particular option by clicking on its name. As each selection is made, the corresponding option will become highlighted. To clear the list of all selections, click on the *Clear* button.



You will sometimes find that selecting an option will cause one or more other options to disappear from this list. This phenomenon arises whenever options have been designated as mutually exclusive. In this case, GS Plan prevents you from choosing two options that would conflict with each other geometrically. (For example, a three-car garage with a gable roof and the same three-car garage with a hip roof.)

To return to the *Drawing Area* showing the base house with all selected options, click on the *OK* button. To ignore any changes, click on *Cancel*. Note that *Cancel* will not necessarily restore the base house if options were selected prior to opening the *Select Options* dialog box. You must first clear the selection(s) and then exit via the *OK* command.

Whenever you leave the *Select Options* screen with one or more options selected, you will be in *Select Options* mode. The same *Exit Options* button will appear on the *Classic Sidebar* as when you were in *Draw Option Mode*. This button provides a convenient way to restore the base house without returning to *Select Options* and deselecting all options.

In *Select Options* mode, you cannot draw any new entities because GS Plan will not know the option to which they belong. You also cannot select and modify existing entities. In fact, most of the *Modify*, *Entities*, *Macros*, and *Lateral* menu commands will be disabled. Only the following commands will remain available:

- *Modify* → *Set Properties* (assuming you have selected a specific entity)
- *Modify* → *Shift/Copy Model*
- *Modify* → *Rotate Model [Temporary]*
- *Lateral* → *Create/Erase Shear Line*.

While using the *Selection Options* function, the *Modify* → *Set Properties* command will allow you to inspect, but not change, the properties associated with the selected option(s). The *Modify* → *Shift/Copy Model* will translate all entities (whether belonging to selected options or not) to a new XYZ-coordinate system. *Modify* → *Rotate [Temporary]* will rotate all entities (whether belonging to selected options or not) around the (0,0) origin. And *Lateral* → *Create/Erase Shear Lines* will create or erase shear lines for all visible entities on the active level, as previously described.

Despite these drawing restrictions, all of the *Tools* and *Display* menu commands are still available in *Select Options* mode. Therefore, you can *Animate*, *Zoom In*, and *Pan* across the entire base house with selected options. You can also check truss profiles, change visibility (by level or by entity type), verify XYZ-coordinate values, and check bearing conditions for any combination of options.

The *File* → *Options Extract As* command operates in conjunction with *Select Options* to generate a set of "L" (.lay and level) files containing the base house plus all selected options. (If no options have been selected, only base house entities will be extracted.)

Other *File* menu commands work with options pretty much as one would expect:

File → *Generate Materials List* creates a file containing non-engineered products and other materials for the base house and selected options only.

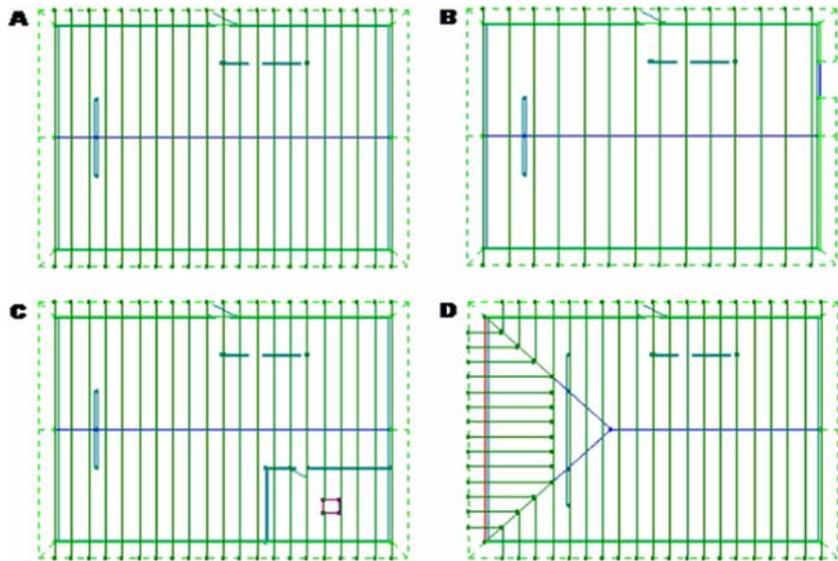
File → *Merge* assumes that all merged entities belong to (the current level of) the base house.

File → *Print Drawing* will print entities belonging to the base house and currently selected options only (i.e. whatever is visible in your drawing area).

Conflicting Options

Any time two or more options cannot co-exist, they are said to conflict. One obvious situation is when options occupy the same geometric space. For example, a three-car garage with a gable roof cannot co-exist with (the same) three-car garage with a hip roof. Another less obvious situation arises when options that apparently do not conflict geometrically require alternate framing configurations – even though those configurations may differ only in some minor respect. Consider the following example:

Figure A shows the base house. Figure B shows the same base house with a fireplace option (overhang cut-away) in the upper right-hand corner. Figure C shows the same base house with an optional room in the lower right-hand corner. Figure D shows the same base house with a hip roof option on the left side.



Although this is just an example, the fact remains that all of these options conflict with each other! The framing area in Figure B requires a different on-center spacing than all the others.

The framing area in Figure C requires a roof opening for a skylight (with a consequent break in one of the trusses), and the framing area in Figure D requires hip-end members from the roof peak down to the left eave. The way these options were modeled was to delete the base house framing area from each of them, and then draw separate framing areas for each one. Note that the roof planes are the same throughout, except for the hip-roof option, where they had to be deleted and replaced with three option-specific planes.

By the way, when in *Draw Option* mode, you may be tempted to delete individual base-house framing members. For example, in drawing the option illustrated in Figure C, you might consider deleting the truss that intersects the skylight opening and drawing two separate truss members on either side. This strategy is not recommended. As soon as you re-generate members for the base house, the deleted truss will re-appear. It is much better to delete and redraw the entire framing area for the specific option.

Copy/Xfer Options

Executing the *Copy/Xfer Option* command will open a *Copy/Transfer Option* dialog box like the one pictured at right. As its name implies, this dialog box is used to make a copy of an existing option, or to transfer an option to a different _____.

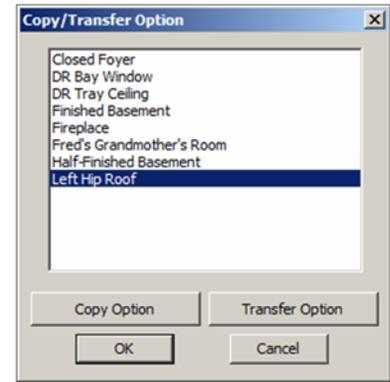
COMMAND BUTTONS

Copy Option – If you attempt to copy a parent-level option, GS Plan will display the error message pictured at right.

Transfer Option – ???

OK – Clicking on the *OK* button...

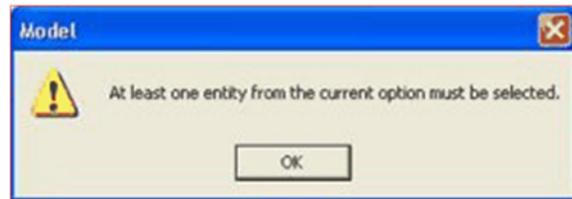
Cancel – Clicking on the *Cancel* button...



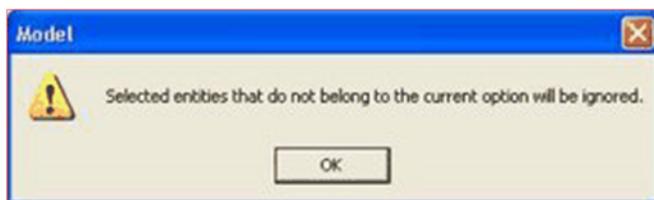
Copy Selected Entities

The *Copy Selected Entities* function adds the ability to make actual physical copies of *Option* entities in addition to the current functionality of being able to make logical copies of *Option* entities. To activate this feature, you must be in *Create/Edit Options* mode with a currently active *Option* displayed on the screen. If this is not the case, the *Copy Selected Entities* menu selection will be disabled, along with the corresponding icon on the *Options* toolbar. Enabled selections will appear as a pull-down selection from the *Tools* menu or as a toolbar icon in the *Options* toolbar.

Copy Selected Entities requires the selection of one or more entities belonging to the current *Option*. If no entities are selected you will receive the following message:



If you have selected entities that do not belong to the current *Option* you will receive this message:

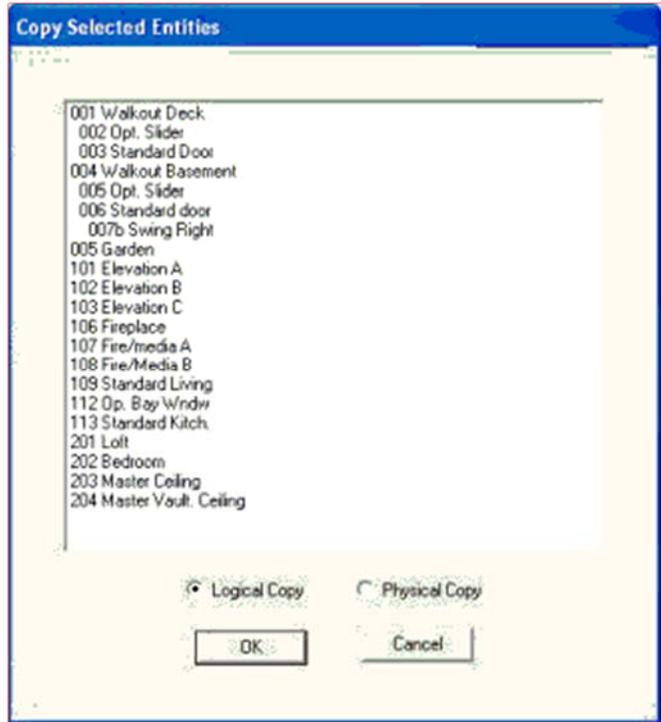


If an entity from the current option has been selected, the *Copy Selected Entities* dialog box will open. The *Copy Selected Entities* dialog box lists the target *Options* available to receive the copies of the selected entities. The current *Option* in this example is:

- 004 Walkout Basement
- 006 Standard Door
- 007a Swing Left

Notice that the current *Option* (007a Swing Left) will never appear in this list. (It does not make sense to copy an option to itself!) Additionally, if the parent of the current *Option* only has one *Sub-Option* (the current option), it will also be removed from the list.

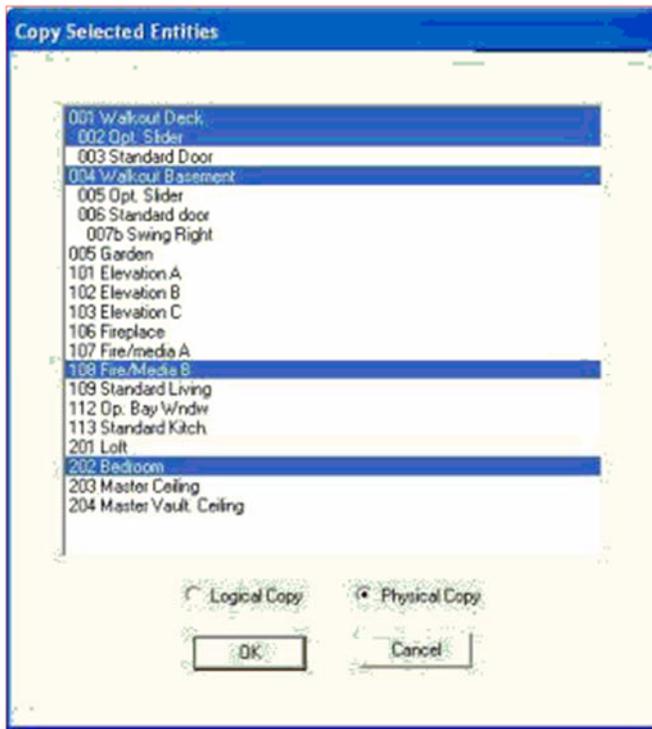
Option entities cannot be copied to direct ancestors of the current *Option*. This logic is applied all the way up the hierarchical chain as appropriate (grandparent *Options*, great grandparent *Options*, and so on). That said, however, if the parent *Option* (or grandparent *Option*, etc.) has more than one *Sub-Option* (a sibling to the current *Option*), the parent *Option* will be displayed along with the sibling *Option(s)*. The ability to copy *Option* entities to sibling *Options* is a necessity. This circumstance creates the potential for copying *Option* entities to direct ancestors which, as we stated previously, is not allowed. Proper handling of this situation will be described below.



You will now select the destination *Options* by highlighting selections in the available *Options* list. Multiple destination *Options* can be selected by holding down the *Control* key down on your keyboard as you click with your mouse.

In addition to selecting destination *Options*, you must also determine whether the new *Option* entities will be *Logical* or *Physical* copies of the current *Option* entities. This is accomplished by clicking on the appropriate radio button. *Logical* is the default selection. A logical copy mimics the current *Copy/Transfer Option* behavior. That is to say, it does not create a new entry in the job files, but rather associates the existing entities with the destination *Options* by setting the appropriate *Option* flag information (a transfer). A *Physical* copy creates a new entry within the job files for each of the selected destination *Options* (a copy). Thus, the newly created entities are autonomous and can be modified without affecting the properties of the original entities.

A typical selection screen might appear as:



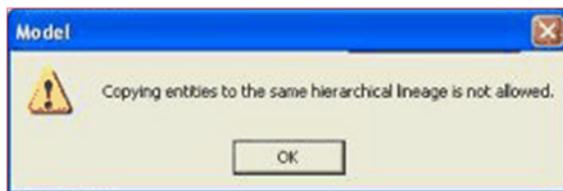
In this situation you have requested to make a physical copy of the currently selected *Option* entities to the following *Options*:

- 001 Walkout Deck
- 002 Opt. Slider
- 004 Walkout Basement
- 108 Fire/MediaB
- 202 Bedroom

Clicking the *Cancel* button will abort the *Copy Selected Entities* process.

Clicking the *OK* button will initiate the process.

There are a few potential issues that need to be addressed. The first is that the options 001 Walkout Deck and 001 Walkout Deck | 002 Opt. Slider are direct ancestors of each other. This will create an unnecessary duplication of the selected entities and as a result will not be allowed. You will receive the following error message and be required to reselect the destination options:



The second is the fact that 004 Walkout Basement is a direct ancestor to the current *Option*.
004 Walkout Basement

006 Standard Door
007a Swing Left

Since this is also not allowed, you will receive this warning message:



You will not be required to reselect destination Options in this case. The offending Option is merely ignored and the remaining valid destination Options are honored.

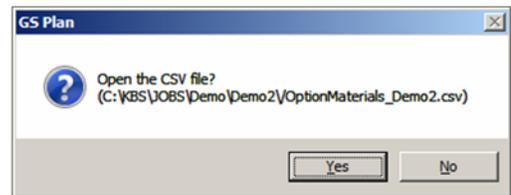
There are no additional error checks aside from those listed above. There is no provision to prevent you from copying a selected entity to a specific destination Option more than once using subsequent Copy Selected Entities requests. Likewise, there is no provision to prevent you from copying a selected entity to an ancestor of an Option to which it has previously been copied. Both of these situations could produce entity duplicates that are unnecessary as well as potentially hazardous to the model processing.

Toolbars – The main Model window can be customized to include one or more toolbars. Each toolbar contains commands either from specific menus (i.e. File, Tools, or Modify) or collections of often used commands (i.e. Classic Sidebar or Standard). Each toolbar option corresponds to a menu option, but they are grouped together for your convenience. Each toolbar is described below. Click on a link to jump to information about that specific toolbar or scroll down to read about each.

Custom Toolbars –

Option Materials

Clicking on the *Option Materials* line-item on the *Tools* → *Options* menu will open a message box like the one pictured at right...



THE MODIFY MENU & TOOLBAR

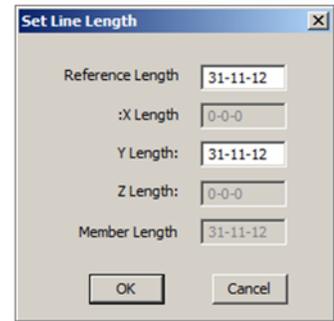
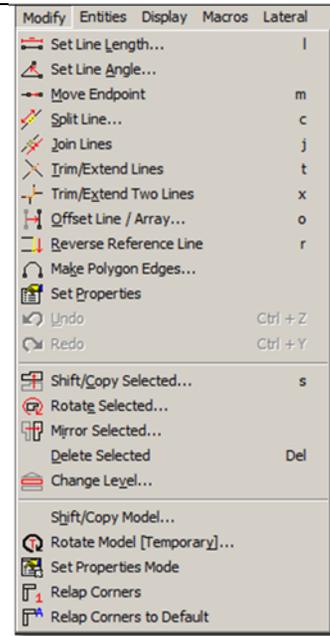
The *Modify* menu and toolbar provide users with commands for editing existing members of your model.



Set Line Length – The *Set Line Length* command is used to view and change the exact length of a selected member. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the line that you want to modify. Keep in mind that the point you select closest to the endpoint is the one that will move.

When you select the *Set Line Length* from the *Modify* menu, a dialog box like the one pictured in Figure X will open.

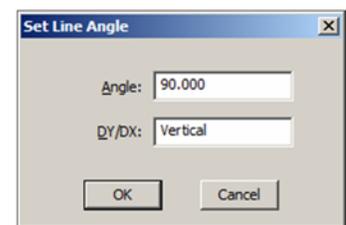
- **Reference Length** – This is the distance, along the line, from the beginning point to the endpoint.
- **X Length** – This is the change, along the X axis, from the beginning point to the endpoint.
- **Y Length** – This is the change, along the Y axis, from the beginning point to the endpoint. In this example, a horizontal line was selected, so the Y Length is grayed out.
- **Z Length** – This is the change, along the Z axis, from the beginning point to the endpoint. In this example, a horizontal line was selected, so the Z Length is grayed out.
- **Member Length** –
- **OK** – Click on the *OK* button to save your changes and close the *Set Line Length* dialog box.
- **Cancel** – Click on the *Cancel* button to discard your changes and close the *Set Line Length* dialog box.



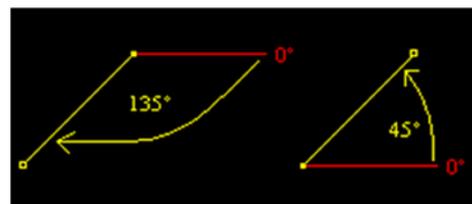
Examples in e-text.



Set Line Angle – The *Set Line Angle* command is used to view and change the exact angle of a selected member. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the line that you want to modify. Keep in mind that the point you select closest to the endpoint is the one that will move. (The line will pivot around the other point.) When you select the *Set Line Length* from the *Modify* menu, a dialog box like the one pictured in Figure X will open.



- **Angle** – This is the angle of the line measured from the beginning point to the endpoint. The angle of a line is based on the XYZ axis and is measured from the beginning point to the endpoint of the line. Pictured at right is the same line selected twice. Notice that the angle



of the line changes because of the location of the endpoint.

- **DY/DX** – This is the change in the Y axis over the change in the X axis.
- **OK** – Click on the *OK* button to apply your changes to the selected line and close the *Set Line Length* dialog box.
- **Cancel** – Click on the *Cancel* button to discard your changes and close the *Set Line Length* dialog box.

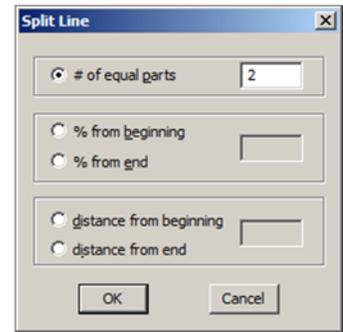
Examples in e-text.



Move Endpoint – The *Move Endpoint* command is used to change the location of a selected member's endpoint. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the line that you want to modify. Keep in mind that the point you select closest to, will be the endpoint (the point that will be moved). Select *Move Endpoint* from the *Modify* menu. Choose the new location for the selected point, using either the mouse or keyboard input commands. (The same way you would choose the endpoint when inputting a single line.)



Split Line – The *Split Line* command is used to break a selected member into multiple pieces. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the line that you want to modify. (Keep in mind that the point you select closest to will be the endpoint.) Next, select the *Split Line* command from the *Modify* menu. When you do so, a dialog box like the one pictured in Figure X will open.



- **# of Equal Parts** – When this radio button is activated, you can enter the number of parts into which you want *Split Line* to break the selected line.
- **% from Beginning** – When this radio button is activated, you can break the selected line by a specified percentage from its beginning point.
- **% from End** – When this radio button is activated, you can break the selected line by a specified percentage from its endpoint.
- **Distance from Beginning** – When this radio button is activated, you can break the selected line by a specified distance from its beginning point.
- **Distance from End** – When this radio button is activated, you can break the selected line by a specified distance from its endpoint.

Examples in e-text.



Join Lines – The *Join Lines* command is used to join two lines of the same type together into one. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the first of the two lines that you want joined together. Select *Join Lines* from the *Modify* menu. Click on a second line that you want joined with the first selected line.



Trim/Extend Lines – The *Trim/Extend Lines* command is used to shorten or lengthen lines so that they meet the selected line. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the line that you want other

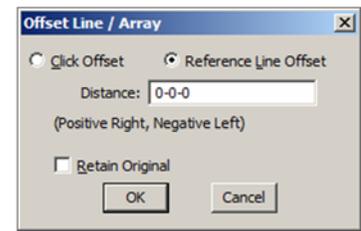
lines to run to (not the lines that will move). Select *Trim/Extend Lines* from the *Modify* menu. Click on the end of the lines that you want to meet the selected line. When finished, click the right mouse button to exit the command.



Trim/Extend Two Lines – The *Trim/Extend Two Lines* command is used to shorten or lengthen two non-parallel lines so that their endpoints meet. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the first of the two lines that you want to meet. Make sure that you click closest to the end at which you want the second line to end. The endpoint is the point that will be joined with the second line. Select *Trim/Extend Two Lines* from the *Modify* menu. Click on the end of the lines that you want to meet the selected lines endpoint. The two selected ends will move so that the lines now are end to end.



Offset Line/Array – The *Offset Line/Array* command is used to move a line a specific distance perpendicular to its original location. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the line that you want to modify. Make sure that you note the end of the line that you click closest to, as the direction of the offset will depend upon the location of the endpoint. Select *Offset Line/Array* from the *Modify* menu. This will open an *Offset Line/Array* dialog box like the one pictured at right.



- **Click Offset** –
- **Reference Line Offset** –
- **Distance** – Enter the distance that the line is to be offset. Enter a positive value to offset the line to the right, and a negative value to offset the line to the left.
- **Retain Original** –
- **OK** – Click on the *OK* button to save your changes and close the *Offset Line/Array* dialog box.
- **Cancel** – Click on the *Cancel* button to discard your changes and close the *Offset Line/Array* dialog box.

Examples in e-text.



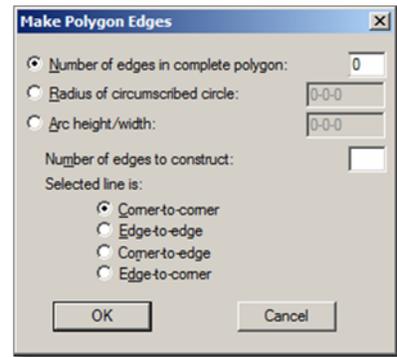
Reverse Reference Line – Bearings are defined by the reference line. The reference line is the part of a wall with a dot on either end. You may need to reverse the reference line due to a bearing or ledger condition.

The *Reverse Reference Line* command is used to change the reference line side of the selected bearing, without moving the bearing's location. The *Reverse Reference Line* command will only become available if you have a bearing member selected. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the bearing that you want to modify. Select *Reverse Reference Line* from the *Modify* menu. The reference line will move to the opposite side of the bearing.

NOTE! A bearing member with the reference line set to center will not be affected by this command.



Make Polygon Edges – The *Make Polygon Edges* command is used to create a polygon (and therefore "rounded" walls) from a current straight line segment. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use your mouse to select the straight line that you want to make rounded. Select *Make Polygon Edges* from the *Modify* menu. The *Make Polygon Edges* dialog box will open.



- **Number of Edges in Complete Polygon** – Enter the number of edges in the complete polygon (as if it were a full circle). This determines the angle of the individual sections as well as the number of segments to appear when the polygon is constructed.
- **Radius of Circumscribed Circle** – Enter the radius of the circle.
- **Arc Height/Width** – Enter the Arc Height/Width.
- **Number of Edges to Construct** – Enter the number of polygon edges to be constructed (with ½ edges being counted as full edges). The number of edges must always be less than the number of edges in the complete polygon, if that option was selected above.
- **Selected Line Is:**
 - A. **Corner to Corner** –
 - B. **Edge to Edge** –
 - C. **Corner to Edge** –
 - D. **Edge to Corner** –



- **OK** –
- **Cancel** –

NOTE! Polygon edges cannot be created from a raked wall.



Set Properties – The *Set Properties* command allows you to view and change the properties of a selected member. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the entity that you wish to modify. Select *Set Properties* from the *Modify* menu. A properties window will open. (Note that the specific type of properties window that opens will depend upon the entity type that you have selected within your model. Review and modify the properties as appropriate, then click *OK*.)



Undo – The *Undo* command reverses the effects of the last single-line *Modify* command.

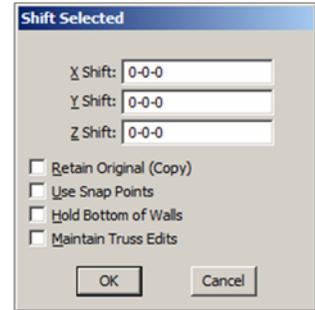


Redo – Click on the Redo button to reapply changes that have been undone using the Undo button.



Shift/Copy Selected – The *Shift/Copy Selected* command allows you to move or copy selected lines in the X, Y, or Z direction. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the member(s) that you want to modify. Select *Shift/Copy Selected* from the *Modify* menu. The *Shift/Copy Selected* dialog box will open.

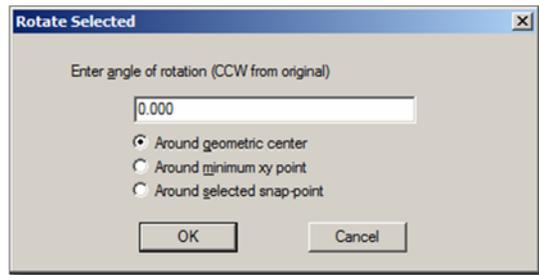
- **X Shift** – This is the amount that you want the selected lines to be moved (or copied) along the X axis. In *Plan View*, enter a positive number to move the line to the right; enter a negative number to move the line to the left.
- **Y Shift** – This is the amount that you want the selected lines to be moved (or copied) along the Y axis. In *Plan View*, enter a positive number to move the line up; enter a negative number to move the line down.
- **Z Shift** – This is the amount that you want the selected lines to be moved (or copied) along the Z axis. In *Plan View*, enter a positive number to move the line toward you; enter a negative number to move the line away from you.
- **Retain Original (Copy)** – If the *Retain Original* checkbox is activated, when the *OK* button is clicked, the original selected line(s) will remain in their original position, and a copy will be created then shifted the distance specified.
- **Use Snap Points** – If the *Use Snap Points* checkbox is activated, when the *OK* button is clicked, you will be prompted to select a "From" and "To" point on your model, rather than defined shift directions and distances.
- **Hold Bottom of Walls** – If the *Hold Bottom of Walls* checkbox is activated, when the *OK* button is clicked, the bottoms of all wall members will remain at their original Z height. The tops of the walls will then move the distance specified in the Z Shift field. This will not affect members shifted on the X or Y axis.
- **Maintain Truss Edits** –
- **OK** – Click the *OK* button to move (or copy) the selected members the distance specified, then return to the main window.
- **Cancel** – Click the *Cancel* button to return to the main window without moving or copying the selected members.



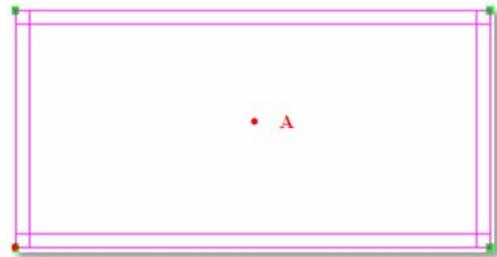
Examples in e-text.



Rotate Selected – The *Rotate Selected* command is used to rotate a selected member (or members), in a counterclockwise direction, around one of three types of points. To use this command, first choose *Select* from the *Classic Sidebar, Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the member(s) that you want to modify. Select *Rotate Selected* from the *Modify* menu. The *Rotate Selected* dialog box will appear.



- **Enter Angle of Rotation (CCW from Original)** – Enter the angle of rotation (positive or negative) that you wish to rotate the model. Remember that the model will be rotated in a counterclockwise direction.
- **Around Geometric Center** – If you choose this option, the model will be rotated around its own center point. (Point A in Figure X.)
- **Around Minimum XY Point** – Choose this option to rotate the model around the point in the model that has the lowest XY coordinate values. (Point B in Figure X.)
- **Around Selected Snap-Point** – Choose this option to rotate the model around any snap-point you choose. If you choose this option and click on the *OK* button, you will be prompted (in the status bar) to click on the snap-point around which you wish to rotate the model.
- **OK** – Click *OK* to rotate the model according to the angle and point selected above.
- **Cancel** – Click the *Cancel* button to return to the main window without rotating your model.



Mirror Selected – The *Mirror Selected* command is used to mirror or invert the selected member (or members) across a specific line. To use this command, first choose *Select* from the *Classic Sidebar, Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the member(s) that you want to modify. Use your mouse to select the axis of reflection (the line that you want the selected member(s) to be mirrored along – usually a construction line). Click on the *Mirror Selected* command on the *Modify* menu. The *Mirror Selected* dialog box will open.

- **Retain Original Portion** – If this box is checked, the selected members will remain where they are and a copy of them will be generated and mirrored along the axis of reflection (the line that you want the selected member(s) to be mirrored along).
- **Mirror Lines on Axis** – If this box is checked, the member selected as the axis of reflection will be mirrored along with the other members. If it is not checked, the axis line will not be mirrored.
- **OK** – Click the *OK* button to return to the main window where you will be prompted to select the axis of reflection. (That is, the line along which you want the selected member(s) to be mirrored.)
- **Cancel** – Click the *Cancel* button to abort the *Mirror Selected* command and return to the main window.



Examples in e-text.

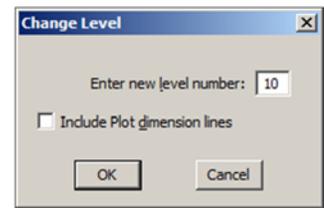
Delete Selected – The *Delete Selected* command is used to permanently erase all selected lines from your model. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the member(s) that you want to delete. Select *Delete Selected* from the *Modify* menu.

If only a single member is deleted with this command, it can be brought back using the *Undo* command. If, however, multiple members are selected when the command is run, the *Delete Multiple Lines* window will open, requiring the user to confirm his or her intention to delete multiple lines. If the user elects to delete multiple members, there is no way to restore the deleted members. (That is to say, there is no *Undo* command available for a multiple-line deletion.)

Shortcut: Press the *Delete* key while one or more members are selected.



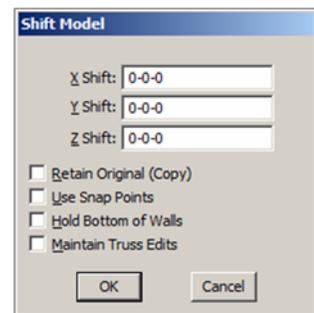
Change Level – The *Change Level* command is used to specify a new level number for the selected members. To use this command, first choose *Select* from the *Classic Sidebar*, *Standard* toolbar, or the *Tools* menu. Use the arrow cursor to select the member(s) that you want to modify. Select *Change Level* from the *Modify* menu. The *Change Level* dialog box will open.



- **Enter New Level Number** – Enter the level to which you want the selected members to be assigned.
- **Include Plot Dimension Lines** – Activate this checkbox if you want to include the dimension lines when the members are moved.
- **OK** – Clicking on the *OK* button will apply your changes and close the *Change Level* dialog box.
- **Cancel** – Clicking on the *Cancel* button will discard your changes and close the *Change Level* dialog box.

When the members are moved to a new level, they will retain their original locations and properties. For example, if a wall with a top plate height of 10-0-0 is moved to a different level, it stays a wall with a top plate height of 10-0-0, regardless of the *Top of Wall* height of the new level. If a member is moved to a level that does not yet exist, that level will automatically be created (with a *Top of Wall* height of 0-0-0) and set to *Visible*.

Shift/Copy Model – The *Shift/Copy Model* command is used to move or copy an entire model in the X, Y, or Z direction. To use this command, select *Shift/Copy Model* from the *Modify* menu. The *Shift/Copy Model* dialog box will open.



- **X Shift** – This field is used to define the distance you want your model moved (or copied) along the X axis. In *Plan View*, enter a positive number to move the line to the right; enter a negative number to move the line to the left.
- **Y Shift** – This field is used to define the distance you want your model moved (or copied)

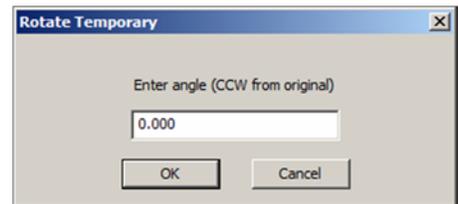
along the Y axis. In *Plan View*, enter a positive number to move the line up; enter a negative number to move the line down.

- **Z Shift** – This field is used to define the distance you want your model moved (or copied) along the Z axis. In *Plan View*, enter a positive number to move the line toward you; enter a negative number to move the line away from you.
- **Retain Original (Copy)** – If the *Retain Original* checkbox is activated, when the *OK* button is clicked, the model will remain in its original position, and a copy will be created then shifted the distance specified.
- **Use Snap Points** – If the *Use Snap Points* checkbox is activated, when the *OK* button is clicked, you will be prompted to select a "From" and "To" point for your model, rather than defined shift directions and distances.
- **Hold Bottom of Walls** – If the *Hold Bottom of Walls* checkbox is activated, when the *OK* button is clicked, the bottoms of all wall members will remain at their original Z height. The tops of the walls will then move the distance specified in the Z Shift field. This will not affect members shifted on the X or Y axis.
- **Maintain Truss Edits** – When the *Maintain Truss Edits* dialog box is activated...
- **OK** – Click the *OK* button to move (or copy) the model the distance specified, then return to the main window.
- **Cancel** – Click the *Cancel* button to return to the main window without moving or copying your model.



Rotate Model (Temporary) – The *Rotate Model (Temporary)* command is used to temporarily rotate the entire model around the Z axis. To use this command, select *Rotate Model (Temporary)* from the *Modify* menu. The *Rotate Temporary* dialog box will open.

- **Enter Angle (CCW from Original)** – Enter the counterclockwise rotation desired, in degrees. To rotate the model back to its original orientation, enter "0."
- **OK** – Click the *OK* button to rotate the job and return to the main window.
- **Cancel** – Click the *Cancel* button to return to the main window without changing the angle of your model.



Examples in e-text.



Set Properties Mode – The *Set Properties Mode* is used to quickly change the properties of any entity. Choose this option from the *Modify* menu or from the right-click menu, and then select any item. The appropriate *Properties* dialog box for that entity will appear. This mode will remain active until you right-click or press the *Escape* key on your keyboard.

Relap Corners –

Relap Corners to Default –

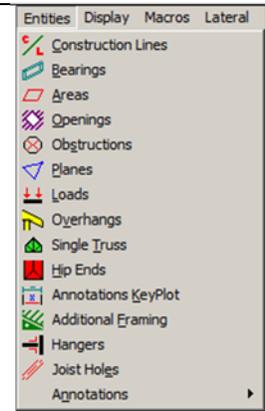
ENTITIES MENU

The *Entities* toolbar provides users with commands to create and modify all of the various parts that make up a structure.



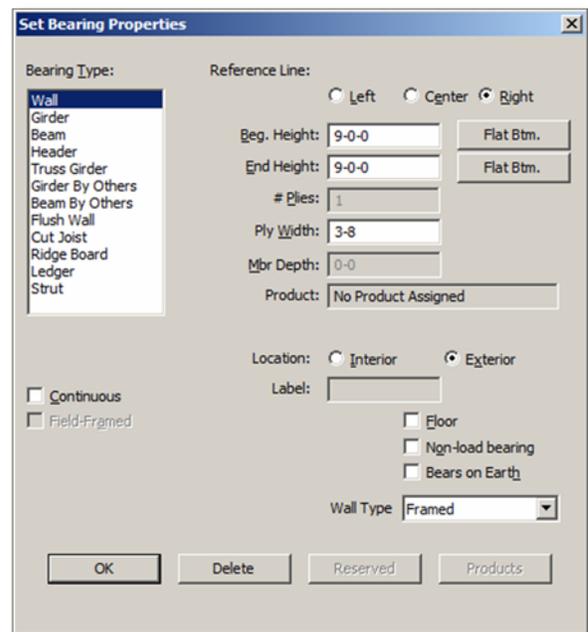
Construction Lines – The *Construction Lines* command allows you to input lines that you can use for reference. Construction lines are lines used for your reference only. A construction line does not affect the loading of a job or final material output. Construction lines are visible only in GS Plan.

To create a construction line, you must select a start point and an endpoint. Since construction lines have no properties to set (as they are just lines), no property box will appear after a construction line is created. After inputting a construction line, your crosshairs will remain connected to the endpoint. The endpoint can then be used as the start point of the next line, or you can click the right mouse button to disconnect.



Bearings – The *Bearings* command is used to input the support members of a model. Bearings are members that directly support other members, or the loads associated with other members. To input a bearing member, you must select a start point and an endpoint. Once the endpoint has been selected, the *Set Bearing Properties* dialog box will open. The specific contents of this dialog box will vary, depending on the *Bearing Type* that is selected. (For more information on the features and functions of the *Set Bearing Properties* dialog box, see the section of this document entitled “Defining Bearing Properties.”)

After drawing a bearing, your crosshairs will remain connected to the endpoint. This endpoint can then be used as the start point for the next bearing, or you can click the right mouse button to disconnect.



The *Bearings* command also allows you to input posts in your model. To input a post, start by selecting a start point. Next, enter a distance of zero (using any of the keyboard input commands). When the *Bearing Properties* window opens, *Post* will be the only choice for *Bearing Type*.



Areas – The *Areas* command is used to add framing areas to a model. An *Area* is a section of the model that is filled with like framing members. Areas are compound entities. That is to say, they are made up of several lines joined together. Areas are used to define floor members, roof rafters, trusses, or concrete slabs.

To draw an *Area*, use the left mouse button to click to all the corners, always moving clockwise around the area to be framed. When you have all but the last side drawn, click your right mouse button to automatically close the *Area* and open the *Area Properties* dialogue box. The *Area Properties* window is pictured at right. See below for more information on each field:

The screenshot shows the 'Set Area Properties' dialog box with the following settings:

- Area Type:** Roof
- Member Type:** Trusses
- Overframing:**
 - Min. Heel Height: 0-0
 - Sheathing: 0-0
- Ledgered Area:**
 - Field-Framed
- Reference Line:**
 - Left
 - Center
 - Right
- Product:** No Product Assigned

Buttons: Define Parallel Side, Define Product List, OK, Delete, Save As Defaults.

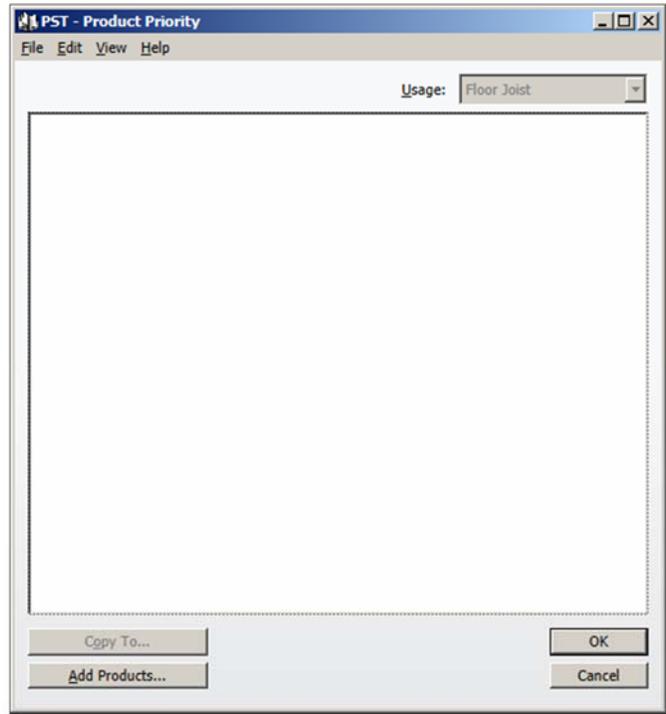
- **Area Type** – The *Area Type* field represents the overall function of the area being created. The options are:
 - ♦ *Roof* – Members that make up the roof of the structure. Often these will be composite members that will make up the roof and ceiling at the same time. Roof areas receive roof loading.
 - ♦ *Ceiling* – Members that make up the ceiling on a roof level.
 - ♦ *Floor* – Members that create the floor of a level. Floor areas receive floor loading.
 - ♦ *Concrete Slab* – This will create a cubic footage of slab for takeoff purposes. No engineering is performed on a slab.
- **Member Type** – This field defines the kind of member that will be used to frame in the area. The choices include:
 - ♦ *Trusses* – *Trusses* are composite members that are generated in Structure, then output to GS Truss software for design. The *Trusses* option is only available if the *Area Type* is set to *Roof* or *Floor*.
 - ♦ *Engineered Joists* – *Engineered Joists* are horizontal members that are loaded and sized in Structure. The *Engineered Joists* option is only available if the *Area Type* is set to *Roof* or *Floor*.
 - ♦ *Non-Engineered Joists* – These are members that generate materials for the final materials list, but have no engineering values. They do not generate, or support, any load. The *Non-Engineered Joists* option is only available if the *Area Type* is set to *Ceiling*, *Roof*, or *Floor*.

- ◆ *Other* – These are members that transfer load and support members above them, however, they do not have materials associated with them. The *Other* option is only available if the *Area Type* is set to *Roof* or *Floor*.
- ◆ *Jacks (Hip End)* –
- ◆ *FTCs (Hip End)* –
- ◆ *Main Girder (Hip End)* –
- **Overframing** – When the Overframing checkbox is activated, the *Roof* members entered will be non-engineered overframe members rather than standard members. Unlike normal roof members (which fill in the area between the roof and ceiling planes), overframe members fill in the area between two overlapping roof planes.
- **Min. Heel Height** – This field defines the minimum value to be used for the vertical dimension measured from the top of a wall to the top of a roof at the exterior face of the wall.
- **Sheathing** –
- **Spacing** – The *Spacing* field defines the distance from the center of one member to the center of the next member. *Spacing* is entered in inches and sixteenths of an inch.
- **Depth** – The *Depth* field is only used when defining floor trusses or a concrete slab. *Depth* is entered in inches and sixteenths of an inch.
- **Offset** – Generally, the first framing member is placed on the edge of the area (parallel side, described below), then members are placed at the standard spacing across the area. If you want the members to start somewhere other than on the edge, a positive offset can be entered. This will move all of the members in the area, maintaining the spacing. Offset is entered in inches and sixteenths of an inch. **Examples available in e-text.**
- **Ledgered Area** – If an area is to bear to the face of a wall or beam, then a ledger needs to be created, and the area needs to be toggled "Ledgered Area." Joists and trusses belonging to a ledgered area will be considered as supported, even though there is no apparent supporting wall or beam.
- **Field-Framed** –
- **Add Bearings** –
- **Add Planes** –
- **Add Overhangs** –
- **Add Sheathing** –
- **Add Rim** –
- **Reference Line** –
- **Product** –

COMMAND BUTTONS

- **Define Parallel Side** – The parallel side is a line that the members of an area run parallel to, and the location from which the layout begins. Click the *Define Parallel Side* button to specify a new line as being the parallel side.

- **Define Product List** – Click on the *Set Products* button to bring up the *PST Product Priority* dialog box pictured at right. This window allows users to specify the materials to be used want to use for the framing members.
- **OK** – Click the *OK* button to apply the information in this window and return to the main GS Plan window.
- **Delete** – Click the *Delete* button to return to the main window without drawing the area. If you have opened the *Set Area Properties* dialog box using the *Set Properties* command, this button will read *Cancel*. Click *Cancel* to return to the main window without applying the changes made to this area’s properties.
- **Save As Defaults** – Click the *Save as Defaults* button to store the information in this window and use it each time a new area is input.



Openings – *Openings* are holes in framing areas that are used to create skylights, stairwells, and other holes. The *Openings* command is used to define openings in framing areas. To draw an opening, use your left mouse button or the keyboard input keys to define all the corners, always moving in a clock-wise around the opening. When you have all but the last side drawn, click your right mouse button to automatically close polygon of the opening. This will open the *Set Opening Properties* dialog box. Click in *Define Bearings* checkbox to activate it. When you click on the *OK* command button, the *Set Bearing Properties* dialog box will open. For more information about the *Set Bearing Properties* dialog box, [see page __](#).





Obstruction – The *Obstructions* command is used to define either circular or rectangular framing obstructions. At this point, obstructions do not automatically adjust framing in GS Plan or in **KeyBuild Structure**. Obstructions *DO* appear in *Adjust Members* in **KeyBuild Structure** and should be used as a visual aid in manually adjusting framing.

Drawing an obstruction is similar to drawing a load, in that you can input a single point or a single line to designate different obstruction types (circular and rectangular vs. point load and line load). To draw a circular obstruction, click on a point with the mouse, and then either click again on the same point with the mouse or use an arrow key and set a "0" length, then right-click or hit the *Escape (Esc)* key. To draw a rectangular obstruction, click on a point with the mouse, and then either click on another point with the mouse or use an arrow key and type in a length, then right-click or hit the *Escape (Esc)* key. You will be prompted with one of the following dialog boxes.



For circular obstructions, input the *Diameter* of the obstruction, with the circle centered around the input point. For rectangular obstructions, input the *Width* of the obstruction, with half its width on either side of the drawn reference line. The *Function* field allows you to enter up to a 20-character description that is saved with the obstruction.



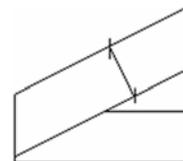
Planes – Planes are 3D surfaces that are used to define roof and ceiling geometry. A plane is defined by a single line (the baseline), and is then extended out to meet other planes and bearings. The *Planes* command is used to define roof or ceiling planes.

A plane is input as a single line along the heel of the framing members. Select the start point, then the ending point, moving clockwise around the building. Once you have drawn the line, click the right mouse button. This will open a *Set Plane Properties* dialog box like the one pictured at right. Define the plane that you want to place and then click on the *OK* button. Once you have entered all of the planes in the roof or ceiling you are defining, use the *Extend* commands to fully develop the geometry. The *Set Plane Properties* dialog box contains the following fields and command buttons:

- **Plane Type** – There are three potential options:
 - ♦ *Roof* – A roof plane will create a surface that the tops of roof framing members will follow.
 - ♦ *Ceiling* – A ceiling plane will create a surface that ceiling members (or the bottom of roof trusses) will follow.
 - ♦ *Stick Roof* – A stick-roof plane functions the same as a roof plane, but the heel height is calculated differently. (The *B/C Upset* turns to *Seat Cut*.)
- **Force Heel Height** – If the *Force Heel Height* checkbox is activated, the user must enter an exact heel height rather than having GS Plan calculate one based on the information in the *Set Plane Properties* dialog box.
- **Match Adjacent** – [Examples in e-text.](#)

Current Plane

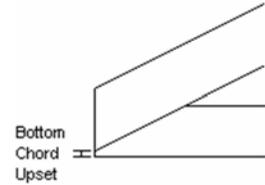
- **Slope** – Enter the slope for this plane. The default value and format of the slope is based on the preset set in the *General Information* window.
- **Seat Cut** – If you are defining a stick-roof plane, this is the size of the *Seat Cut*. This is the amount of the rafter that will sit flat on the bearing. This is used when the *Heel Height* is calculated. If you are defining a ceiling plane, this field is grayed out.
- **T/C Member Depth** – This is the actual width of the member being used as the top chord of a truss or the roof rafter. This value is used when calculating a *Heel Height*. This field is grayed out if you are defining a ceiling plane.
- **Overhang Length** – This field is only used when the *Match Adjacent* checkbox is activated. This value is used by the *Match Slope* calculator. If the *Match Adjacent* checkbox is activated, the *Heel Height* will be generated to align with an adjacent plane. The fields below this checkbox are used to enter information for the adjacent plane, and remained grayed out until this box is checked.



T/C Member Depth

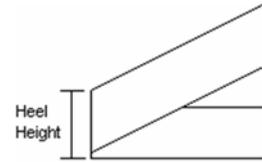
[Match Adjacent] Plane

- **Slope** – This field is only used when the *Match Adjacent* checkbox is activated. This value is used by the *Match Slope* calculator.
- **B/C Upset** – If you are defining a roof plane, this is the bottom chord upset. The bottom chord upset (or butt cut) is the amount of the end of the bottom chord that is left flat when it is beveled. This is used when the *Heel Height* is calculated. This field is only used when the *Match Adjacent* checkbox is activated. This value is used by the *Match Slope* calculator.
- **T/C Member Depth** – This field is only used when the *Match Adjacent* checkbox is activated. This value is used by the *Match Slope* calculator.
- **Overhang Length** – This field is only used when the *Match Adjacent* checkbox is activated. This value is used by the *Match Slope* calculator.



Other

- **Continuous** – If the *Continuous* checkbox is activated, all of the planes drawn after this one (until the plane command is exited, or restarted) will have the same properties. You will not be prompted for property information after inputting planes.
- **Heel Height** – If you are defining a roof or stick-roof plane, this is the *Heel Height* of the plane (or the vertical distance from the wall to the top of the plane). If the *Force Heel Height* checkbox is activated, this value can be entered directly. Otherwise, it is calculated by the *Heel Height* calculator or the *Match Adjacent* calculator.



The *Heel Height* calculator causes the value in the *Heel Height* field to be calculated automatically, based on the information in the *Slope*, *B/C Upset* (or *Seat Cut* for stick-roof planes) and *T/C Board Width*. **As long as the *Force Heel Height* flag is not set, this calculation will be carried out automatically.**

If the *Match Adjacent* option is selected, the value in the *Heel Height* field will be automatically generated. This value is created based on the information for the current plane (*Slope*, *B/C Upset* (or *Seat Cut* for stick-roof planes), *T/C Board Width* and *Overhang*) and the information for the plane to be matched (the fields below the *Match Adjacent* checkbox.)

- **Cantilever Offset** –

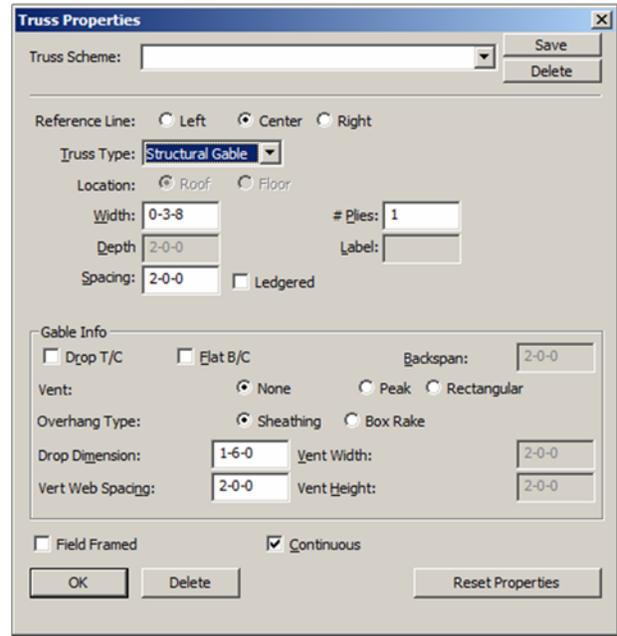
Command Buttons

- **Save As Defaults** – Click this button to store the information in this window to be used each time a new plane is input.
- **OK** – Click the *OK* button to apply the information in this window and return to the main window.
- **Delete** – Click the *Delete* button to return to the main window without inputting the plane. If you are editing a plane using the *Set Properties* command, this button will read

Cancel. Click the *Cancel* button to return to the main window without applying any changes made to the properties.

- **Vertical** – Click the *Vertical* button to cause this plane to angle straight up. A vertical plane allows any plane input to cut along the vertical plane. The only property that is used in a vertical plane is the *Plane Type*.
- **Gable** – Click the *Gable* button to turn the plane line drawn into a *Gable*. This will open the *Truss Properties* dialog box, like the one pictured at right.

- ♦ **Truss Scheme**
- ♦ **Save**
- ♦ **Delete**
- ♦ **Reference Line**
 - *Left*
 - *Center*
 - *Right*
- ♦ **Truss Type**
 - *Truss*
 - *Valley Truss*
 - *Non-Structural Gable*
 - *Structural Gable*
 - *Valley Gable*
- ♦ **Location**
 - *Roof*
 - *Floor*
- ♦ **Width**
- ♦ **Depth**
- ♦ **Spacing**
- ♦ **Ledgered**
- ♦ **# Plies**
- ♦ **Label**
- ♦ **Gable Info**
- ♦ **Drop T/C**
- ♦ **Flat B/C**
- ♦ **Backspan**

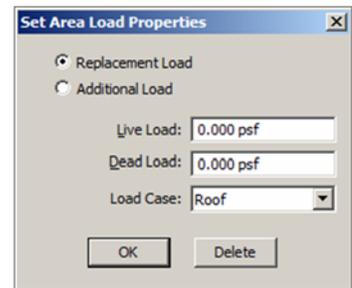


- ◆ **Vent**
 - *None*
 - *Peak*
 - *Rectangular*
- ◆ **Overhang Type**
 - *Sheathing*
 - *Box Rake*
- ◆ **Drop Dimension**
- ◆ **Vert Web Spacing**
- ◆ **Vent Height**
- ◆ **Vent Width**
- ◆ **Field Framed**
- ◆ **Continuous**
- ◆ **OK**
- ◆ **Delete**
- ◆ **Reset Properties**



Loads – The *Loads* command allows you to define non-standard point loads, line loads, and area loads.

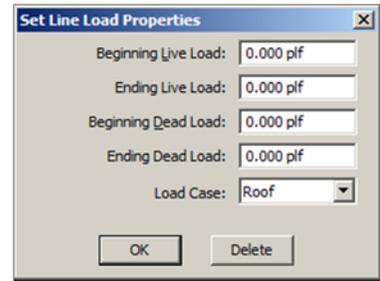
- **Area** – To define an *Area* load, move clockwise around the section that you want loaded. When the last corner has been chosen, click the right mouse button. This will open the *Set Area Load Properties* dialog box pictured at right:



- ◆ **Replacement Load** – When this radio button is selected, GS Plan replaces the standard loads on members with the loads defined in this dialog box.
- ◆ **Additional Load** – When this radio button is selected, GS Plan adds the loads defined in this dialog box to the standard loads on members.
- ◆ **Live Load** – This is the *Live Load* to be use to load any section of any members that fall inside this area.
- ◆ **Dead Load** – This is the dead load to be use to load any section of any members that fall inside this area.
- ◆ **Load Case** –
- ◆ **OK** – Click the *OK* button to apply any changes made to the information in this dialog box and return to the main window.
- ◆ **Delete** – Click the *Delete* button to return to the main window without drawing the load. If you are editing a load using the *Set Properties* command, this button

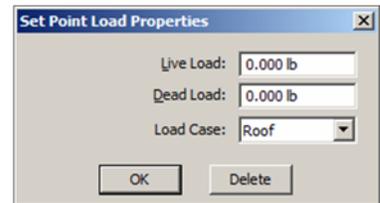
will read *Cancel*. Click the *Cancel* button to return to the main window without applying any changes.

- **Line** – Click the left mouse button at the start of where the load should be, then again at the end. Then click the right mouse button. This will open the *Set Line Load Properties* dialog box pictured at right:



- ♦ **Beginning Live Load** – This is the *Live Load* at the beginning of the line (or the first point you defined).
- ♦ **Ending Live Load** – This is the *Live Load* at the end of the line (or the second point you defined).
- ♦ **Beginning Dead Load** – This is the *Dead Load* at the beginning of the line (or the first point you defined).
- ♦ **Ending Dead Load** – This is the *Dead Load* at the end of the line (or the second point you defined).
- ♦ **Load Case** –
- ♦ **OK** – Click the *OK* button to apply any changes made to the information in this window and return to the main window.
- ♦ **Delete** – Click the *Delete* button to return to the main window without drawing the load. If you are editing a load using the *Set Properties* command, this button will read *Cancel*. Click the *Cancel* button to return to the main window without applying any changes.

- **Point** – To input a point load, select the point that the load should be at two times (either by clicking it twice with the mouse, or selecting it and using the keyboard commands to move a zero length), then click the right mouse button. This will open the *Set Point Load Properties* dialog box pictured at right:



- ♦ **Live Load** – This is the *Live Load* that is to be applied at this point.
- ♦ **Dead Load** – This is the *Dead Load* that is to be applied at this point.
- ♦ **Load Case** –
- ♦ **OK** – Click the *OK* button to apply any changes made to the information in this window and return to the main window.
- ♦ **Delete** – Click the *Delete* button to return to the main window without drawing the load. If you are editing a load using the *Set Properties* command, this button will read *Cancel*. Click the *Cancel* button to return to the main window without applying any changes.



Overhangs – The *Overhangs* command allows you to specify the location and lengths of truss or rafter overhangs.

Overhangs will cause the ends of roof framing members to extend past the edge of the area. An overhang is entered as a single line along the edge of a roof framing area. To input the line, select the start point, then move clockwise to the second point and click again. As soon as the second point is selected, the *Set Overhang Properties* dialog box will open. Specify the overhang length, then click the *OK* button. The ends of the framing members will not extend out until the *Generate Members* command is run.



- **Overhang** – This is the length of the overhang, measured horizontally.
- **Continuous** – If the *Continuous* box is checked, all of the overhangs drawn subsequently will have the same properties. The user will not be prompted for property information after inputting overhangs.
- **Save As Defaults** – Click this button to store the information in this dialog box to be used each time a new overhang is input.
- **OK** – Click the *OK* button to apply the information in this window and return to the main window.
- **Delete** – Click the *Delete* button to return to the main window without inputting the overhang. If you are editing an existing overhang, using the *Set Properties* command, this button will read *Cancel*. Click the *Cancel* button to return to the main window without applying any changes made to the overhang properties.



Single Truss – The *Single Truss* command is used to specify a single truss at a specific location. The *Single Truss* entity allows you to place a single truss member at a specific location. To input a truss member, you need to select a start point and an ending point. Since truss members have no properties to set (like standard trusses, they just follow the planes around them) no property box will appear after it is input.



Hip Ends – The *Hip Ends* command is used to specify the type and location of hip-end framing. A hip-end line is the only way to define hip-end framing members. A hip end is entered as a single line along the edge of a roof framing area where the heels of the jack members will be placed. To input the line, select the start point, then move clockwise to the second point and click again. As soon as the second point is selected, then *Hip End Properties* window will appear. Specify how the hip end should be framed, then click the *OK* button. At this point, you will be prompted to specify the planes and area that define this hip end. Pick the area that the hip end falls in, and then pick the right, face, and left planes of the hip end. The hip-end framing will then appear.

- **Master Girder** –
- **Setback** –
- **Plies** –

Flat Top Commons

- **Placement** – This controls how you want the flat-top commons to be placed. Flat-top commons will follow the IOC spacing of the hip end, regardless of how this flag is set. There are two options:
 - ◆ *Girder to Peak* – This will start the commons at the center line of the girder, and space them up to the peak.
 - ◆ *Peak to Girder* – This will start the commons at the peak, and space them down toward the girder.
- **Spacing** – This is the distance from the center of one hip-end member to the next. All members placed from the peak to the face of the hip are hip-end members.
- **Truss at Peak** –

Left Corner Set

- **Plies** – This is the number of plies in the girder.
- **Holdback from Girder** –
- **Vert Web Holdback** –

Right Corner Set

- **Plies** – This is the number of plies in the girder.

- **Holdback from Girder –**
- **Vert Web Holdback –**

Jacks/Sub-Girders

- **Placement –**
- **Spacing –**
- **Offset End Jack –**
- **Offset Corner Jacks –**
- **Holdback from Girder End Jack –**
- **Holdback from Girder Corner Jacks –**
- **Vert Web Holdback End Jack –**
- **Vert Web Holdback Corner Jacks –**

Command Buttons

- **OK** – Click the *OK* button to apply the information in this window and return to the main window. If you are inputting this hip end and have not already used the *Define Area & Planes* button, you will be prompted to select the area and planes associated with this hip end.
- **Cancel** – Click the *Delete* button to return to the main window without placing the hip end. If you are editing this hip end (using the *Set Properties* window), this button will read *Cancel*. Click the *Cancel* button to abandon the changes made to the properties and return to the main window.
- **Save As Defaults** – Click this button to store the information in this window and use it every time a new hip end is entered.
- **Restore Defaults –**
- **Select Planes –**
- **Select Area –**
- **Clear Planes & Area –**



Annotations Plot – The *Annotations Plot* command opens *KeyPlot*, which is used to place dimensions, labels, and text on your model. You can then use *KeyPlot* to print your model, including the information you have added, or you can export the model as a DXF file. **For more information about KeyPlot, see the section entitled _____ on page __ of this document.**



Additional Framing – ...this will open the *Set Framing Accessory Properties* dialog box pictured at right:

- **Framing Entity** – ...The list of potential values includes:
 - ♦ *Rim Joist* –
 - ♦ *Blocking* –
 - ♦ *Bracing* –
 - ♦ *Cant. Reinf.* –
 - ♦ *Chase Reinf.* –
 - ♦ *Floor Joist* –
 - ♦ *Web Stiffener LGS* –
 - ♦ *Roof Rafter* –
 - ♦ *Roof Rafter (NE)* –
 - ♦ *Ceiling Joist* –
- **Reference Line** –
- **Product** –
- **Ply Width** –
- **Mbr Depth** –
- **# Plies** –
- **Label** –
- **Continuous** –
- **Load Bearing** –
- **Field Framed** –
- **Same as Joist** –

Set Framing Accessory Properties

Framing Entity: **Rim Joist**
Blocking
Bracing
Cant. Reinf.
Chase Reinf.
Floor Joist
Web Stiffener LGS
Roof Rafter
Roof Rafter (NE)
Ceiling Joist

Reference Line: Left
 Center
 Right

Product: No Selection

Ply Width: 1-2

Mbr Depth: 9-8

Plies: 1

Label:

Continuous Field Framed
 Load Bearing Same as Joist

Set Product
Clear Product
Save Defaults

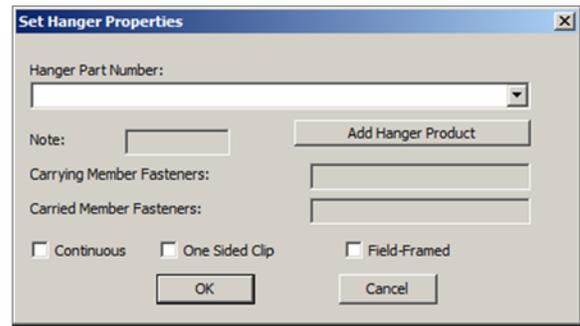
OK Delete

Command Buttons

- **Set Product** – Clicking on this command button will open a Material dialog box that will allow you to select a specific for your Framing Entity. The specific contents of the Material dialog box will depend upon the currently selected type of Framing Entity.
- **Clear Product** –
- **Save Defaults** –
- **OK** –
- **Delete** –



Hangers – The *Hangers* command is used to add non-engineered hangers to a model. When you choose the *Hangers* option from the *Entities* menu, the cursor will change to GS Plan’s cross-hair cursor and you will be prompted to *Set Start Point*. Using standard drawing conventions, draw a line across the ends of the members to which you wish to add hangers. After drawing the line, the *Set Hanger Properties* dialog box will appear.



You can then select a hanger from the pull-down menu, or click on the *Add Hanger Products* button to access the *Add Hanger Products* dialog box. In this dialog box, you must provide a *Hanger Part Number*. Optionally, you can add a *Note*, *Carrying Member Nailing* properties, and *Carried Member Nailing* properties. If you check the *Save to Master Hanger Product List* checkbox, that hanger will be available in future jobs. If you do not check the checkbox, that hanger will only be available in the current job.

Please note that the *Master Hanger Product List* contained in GS Plan is in no way associated with the Simpson Strong-Tie or any alternate hanger manufacturer’s product lists provided in **KeyBuild Structure** and **KeyMan**.

- **Hanger Part Number** –
- **Note** –
- **Add Hanger Product** – Clicking on this command button opens the *Add Hanger Product* dialog box pictured at right.
 - ◆ **Hanger Part Number** –
 - ◆ **Note** –
 - ◆ **Carrying Member Fasteners** –
 - ◆ **Carried Member Fasteners** –
 - ◆ **Save Master Hanger Product List** –
 - ◆ **OK** –
 - ◆ **Cancel** –
- **Carrying Member Fasteners** –
- **Carried Member Fasteners** –
- **Continuous** –
- **One-Sided Clip** –
- **Field Framed** –
- **OK** –
- **Cancel** –





Joist Holes – A new *Joist Hole* entity is used to define hole locations in floor joists and then have those hole locations included in the floor joist design in GS Structure. To draw a joist hole line, select the *Joist Holes* command from the *Entities* menu or toolbar. Once the command is selected, the cursor will change to the cross-hairs input cursor. Select the start and end points for the hole line. This will open the *Set Joist Hole Properties* dialog box pictured at right:

- **Type** – There are two potential values:
 - ◆ *Circular* – Choose this option to create a joist hole that is circular in shape.
 - ◆ *Rectangular* – Choose this option to create a joist hole that is rectangular in shape.
- **Diameter** – When a *Circular* hole type is selected, fill in the *Diameter* field with the joist hole *Diameter*. When a rectangular hole type is selected, fill in the *Width* and *Height* fields with the joist hole dimensions.
- **Description** – Optionally, you can enter a *Description* for the joist-hole line. This joist hole *Description* will be included on the *Framer's Plot* generated from GS Structure.
- **Vertical Offset** – The *Vertical Offset* field defines the offset above the *Set Top of Wall* elevation at which the joist hole line is input. Please note that joist holes designed in GS Structure will assume that the joist hole is centered in the depth of the joist. The *Vertical Offset* of the joist-hole line will be displayed as input when viewed in 3D, and is included in any saw files generated by GS Structure.
- **Slope** – The *Slope* field defines the slope of the joist-hole line drawn in 3D. Please note that joist holes designed in GS Structure assume that the joist hole is centered in the depth of the joist. The *Slope* of the joist-hole line will be displayed as input when viewed in 3D, and is included in any saw files generated by GS Structure.
- **Continuous** – When the *Continuous* checkbox is activated, GS Plan will allow you to continue to input joist-hole lines using the input properties, without prompting you for the properties for each new joist-hole line. Right-click to exit the command.
- **OK** –
- **Delete** –

Annotations – Clicking on the *Annotations* command on the *Entities* menu opens a sub-menu with the following seven options:



Model Entity Labels –



Draw Dimension Lines – A *Dimension Line* is an annotation that is used to display the distance between two points. *Dimension Lines* are saved with the job and included in *Layout Plots* generated from GS Plan. Please note that they will not be included in plots from GS Plan Structure. To draw a *Dimension Line*, select the *Draw Dimension Lines* command from the *Entities | Annotations* pull-down menu or from the new *Annotations* toolbar.



Draw Arbitrary Lines –



Draw or Edit Text Box –



Edit Annotation Attributes –

▪ **Options –**



Point-to-Point Dimension Lines – A Point-to-Point Dimension Line displays the actual distance between two selected end-points and is drawn parallel to a line connecting the selected end-points.



Horizontal Dimension Lines – A Horizontal Dimension Line displays the Horizontal distance between two selected end-points



Vertical Dimension Lines – A Vertical Dimension Line displays the Vertical distance between two selected end-points.



Auto Dimensions Filter

DISPLAY MENU & TOOLBAR

The *Display* toolbar provides quick access to...



Zoom Window – The *Zoom Window* command is used to change the view of a model in the main GS Plan window in one of three ways. You can use the keyboard to pan, zoom in and out, or use the mouse to create a zoom window.

After selecting the *Zoom Window* command, you can use the arrow keys to move the view of the model. This allows you to pan around the model quickly. For example, the *Up* arrow key will move the view of the model up. When you are finished panning, click the right mouse button.

Clicking the *Page Up* key will zoom out on the model, while the *Page Down* key will zoom in. This allows you to change the level of zoom without having to use a window. When finished, click the right mouse button.

If you wish to enlarge a specific section of the model, using the zoom window is best. Use your mouse to choose the opposite corners of a new viewport (that is, the section of the model that you want enlarged).



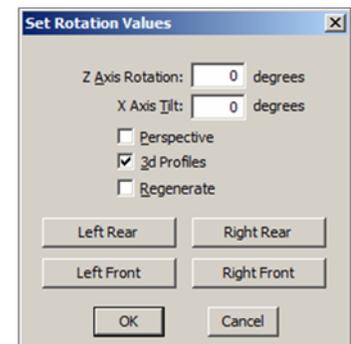
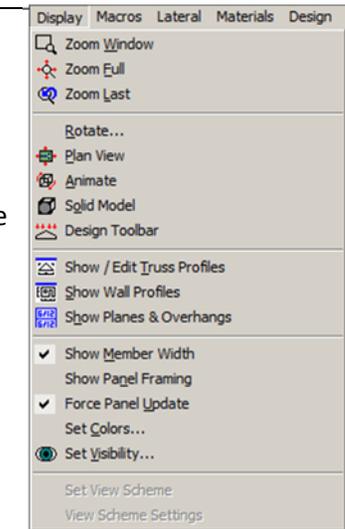
Zoom Full – Selecting the *Zoom Full* command will increase or decrease the current view, so that every section of the model can be seen.



Zoom Last – The *Zoom Last* command returns the user to the previous view of the model.

Rotate – The *Rotate* command is used to set the exact angle of rotation of the model, as well as establish animation defaults. Clicking the *Rotate* command will display the *Set Rotation Values* dialog box pictured at right:

- **Z Axis Rotation** – This is the value, in degrees, that the model should be rotated around the Z axis.
- **X Axis Tilt** – This is the value, in degrees, that the model should tilt forward on the X axis.
- **Perspective** – If this box is checked, GS Plan will draw the animated model from a slightly altered angle, to enhance the perspective.
- **3D Profiles** – If this box is checked, GS Plan will draw roof truss members as three-dimensional members rather than single lines.
- **Regenerate** – If this box is checked, framing members will be regenerated every time you enter animate mode, regardless of changes to the model.



Command Buttons

- **Left Rear** – Click the *Left Rear* button to change the *Z Axis Rotation* to a pre-defined value of 225 degrees.
- **Right Rear** – Click the *Right Rear* button to change the *Z Axis Rotation* to a pre-defined value of 135 degrees.

- **Left Front** – Click the *Left Rear* button to change the *Z Axis Rotation* to a pre-defined value of 315 degrees.
- **Right Front** – Click the *Left Rear* button to change the *Z Axis Rotation* to a pre-defined value of 45 degrees.
- **OK** – Click on the *OK* button to view your model with your newly defined rotation values.
- **Cancel** – Click on the *Cancel* button to abandon your rotation changes and return to the main window.



Plan View – The *Plan View* command will display your model from above, as large as possible on the screen, so that it fills the entire drawing area.



Animate – The *Animate* command is used to rotate your model in 3D space. After selecting *Animate*, your model will be displayed in a preset 3D view. Use the arrow keys on your keyboard (not the keys on the numeric keypad on your keyboard) or your mouse to move the model. To exit the *Animate* mode and return to your previous view, click the right-mouse button or press the *Escape* key on your keyboard. If you move to a 3D view that you want to keep, either click the left-mouse button or press the *Home* key on your keyboard.



Solid Model – This feature allows you to view jobs with a *Solid Model* view in a modeless window. The *Solid Model* viewing window can be resized and/or repositioned at any time. Size and position information are stored when the viewing window is closed and restored when the window is opened. Pressing the *Escape* (*Esc*) key while in the viewing window will close the window and return you to the standard GS Plan view.

Once in the *Solid View* window, you can rotate the model by using the arrow keys on the keyboard, or holding down the left-mouse button and moving the mouse back and forth. To zoom in and out, use the '+' or '-' keys on your keyboard, or the scroll-wheel on your mouse. To pan, hold down the middle-mouse button and move your mouse. To change the display of the *Solid Model* view, right-click in the *Solid View* window. Selecting an item in the right-click menu will toggle a check mark next to that item, to indicate whether or not the behavior attributed to the item is currently active. Note: The menu item *Copy Current View* is not a toggled item.



Design Toolbar –



Show/Edit Truss Profiles – The *Show Truss Profiles* command is used to view the two-dimensional outlines of truss members in your layout. To use *Show Truss Profiles*, select the command from the *Display* menu, then select the truss that you wish to view with your mouse. A profile window will appear. This window will show an outline of the truss, as well as basic information about the truss geometry (slope, heel height, truss length, etc.). Click another truss to display it in the profile window. To close the profile window and exit the *Show Truss Profile* mode, right click your mouse or press the *Escape* key on your keyboard.



Show Wall Profiles – The *Show Wall Profiles* command is used to view the two-dimensional outlines of walls in a layout. It is especially useful for verifying header heights. To use *Show Wall Profiles*, select the command from the *Display* menu, and then select the wall that you wish to view with your mouse. A profile window will appear. This window will show an outline of the wall, as well as basic information about the wall's geometry (length, height, etc.).

If you click on the *Wall Profile* box (to give it focus), and then press the *Home* key on your keyboard, the profile will change to include all the wall segments within the same "span." (That is, walls that are aligned but are not the same wall). Also, if you press the *End* key on your keyboard,

the horizontal dimensions to each door and window opening will toggle from center to right to left (similar to the *Quick Header* macro).

To view another wall profile, simply click on another wall to display that wall in the profile window. (There is no need to close the box and start over; the box is "modeless" and, therefore, you have the ability to click on additional walls in the plan). To close the profile window and exit *Show Wall Profile* mode, right-click your mouse or press the *Escape* key on your keyboard.



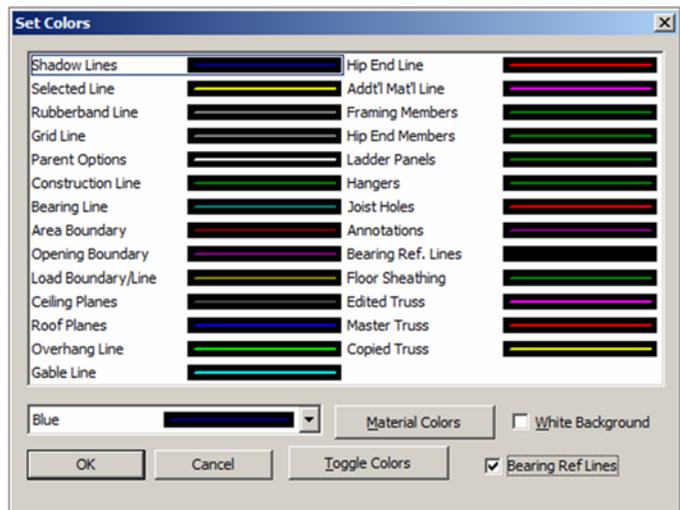
Show Planes & Overhangs –

Show Member Width – Choose this option to display both sides of each framing member, rather than just their centerline locations. This view can be used in conjunction with floor joists, rim joists, blocking panels, web stiffeners, cantilever reinforcements, roof rafters, trusses, jacks, and all other hip members. Display rules apply to both 2D (*Plan View*) and 3D (*Animate*) screen displays. In 3D views, the depth of each member is also displayed. Note: To save the setting of this checkbox for future runs of GS Plan, make sure to choose the *Save Defaults* option from the *File* menu.

Show Panel Framing –

Force Panel Update –

Set Colors – The *Set Colors* command is used to define the colors to be used in GS Plan. Select *Set Colors* to display the *Set Colors* window pictured at right. Use this window to change the display colors for one or more entity types:



- **[Entity List]** –
- **[Color List]** –
- **White Background** – To change the background color to white, click the *White Background* check box. If this box is not checked, the background will use the default color (black). Note: Since the background can be changed from black to white, both black and white lines show up as light gray.
- **Bearing Ref Lines** –

Command Buttons

- **Material Colors** –
- **OK** –
- **Cancel** –
- **Toggle Colors** –

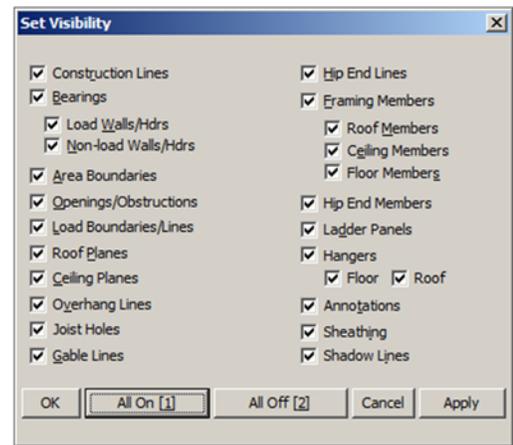
To change the color used to display an entity, click on the entity name in the list. Next, click the drop-down color field at the lower left of the window. A list of available colors will appear. Click

on the color that you want to assign to the selected entity. Once the colors are set, click the *OK* button. The colors will not be saved as the new default color unless the *Save Defaults* command is run. The list of default colors assigned to the various entity types is as follows:

- Shadow Lines (**Blue**)
- Selected Line (**Bright Yellow**)
- Rubber Band Line (**White**)
- Grid Line (**White**)
- Construction Line (**Green**)
- Bearing Line (**Cyan**)
- Area Boundary (**Red**)
- Opening Boundary (**Magenta**)
- Load Boundary (**Brown**)
- Plane Boundary (**Bright Blue**)
- Overhang Line (**Bright Green**)
- Gable Line (**Bright Cyan**)
- Hip End Line (**Bright Red**)
- Framing Members (**Green**)
- Hip End Members (**Green**)
- Ladder Panels (**Green**)
- Background (**White**)



Set Visibility – The *Set Visibility* command is used to pick which entities should be displayed in the *Drawing Area*, and which should be hidden. If a member type name has a check next to it in the *Set Visibility* dialog box, it will be displayed in the *Drawing Area*. If the member is not checked, it will not be shown. To check (or uncheck) a member type, click in the box to the left of the name.



Command Buttons

- **OK** – Click on the *OK* button to save your changes and close the *Set Visibility* dialog box.
- **All On [1]** – Click on the *All On* button to activate the checkbox for all entity types. Alternatively, you can press the “1” key on your keyboard to select all items.
- **All Off [2]** – Click on the *All Off* button to deactivate the checkbox for all entity types. Alternatively, you can press the “2” key on your keyboard to deselect all items.
- **Cancel** – Click on the *Cancel* button to discard your changes and return to the main window.
- **Apply** –

Set View Scheme –

View Scheme Settings –

MACROS MENU

The *Macros* menu contains functions that combine several commonly performed commands into a single step.



Generate Members – Selecting *Generate Members* will recalculate and regenerate any visible framing members in the model.

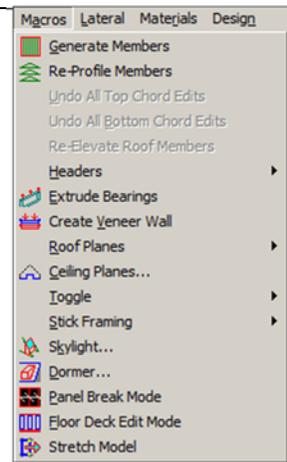


Re-Profile Members –

Undo All Top Chord Edits –

Undo All Bottom Chord Edits –

Re-Elevate Roof Members –



Headers – The *Headers* macro is used to create, move, or delete door and window headers. Selecting the *Headers* option from the *Macros* menu will display the *Headers* sub-menu. This sub-menu contains commands that are used to enter, edit, and delete door and window headers. The sub-menu contains the following commands:

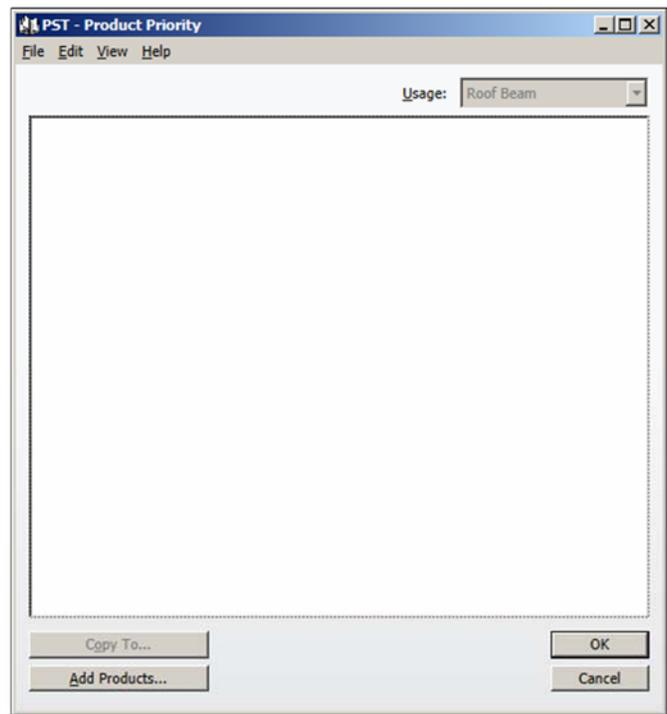
- *Header Products...*
- *Quick Header...*
- *Move Header*
- *Snap Header...*
- *Verify Components*
- *Global Change Components...*
- *Delete Header...*
- *Move Snap Header*

We will now turn to an exploration and explanation of these sub-menu functions.

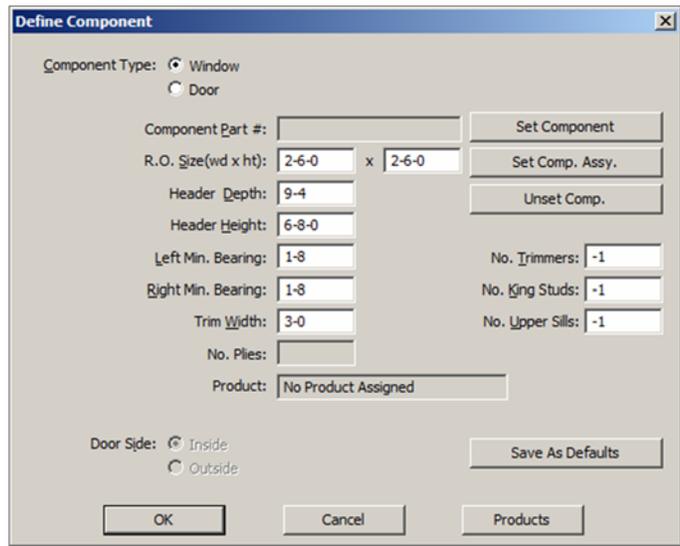
- **Header Products** – The *Header Products* command is used to set the materials used for door and window headers. Selecting *Header Products* from the *Headers* sub-menu will open the *Change Header Products* window pictured at right. To set, or change, the material used for a specific type of header, choose that header type and then click the *OK* button. This will display the *View/Edit Products* dialog box, allowing you to choose which materials you want to be used for the headers.



- ♦ **Loadbearing** –
- ♦ **Non-Loadbearing** –
- ♦ **Both** –
- ♦ **OK** – After you have selected the desired header type, click on the *OK* button. This will open the *View/Edit Products* dialog box pictured at right. Use this dialog box to select the materials you want to be used for your headers.
- ♦ **Cancel** – Click the *Cancel* button to return to the main window without altering the product lists for headers.

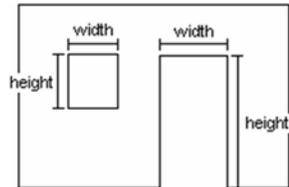


- **Quick Header** – The *Quick Header* command is used to place headers into interior and exterior framed walls. Selecting *Quick Header* from the *Macros* menu will open the *Define Component* dialog box pictured at right. This dialog box allows you to define the opening that you wish to place:

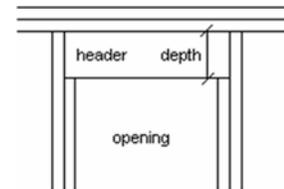


- ◆ **Component Type** – Choose the type of opening component you want to use; *Door* or *Window*. The fields that are available below will change based on the *Component Type* you select.
- ◆ **Component Part #** – This is the name of the component that you are currently viewing. To change (or choose) a component, click the *Define Component* button. Note: it is not necessary to define a component; you may input an opening based on rough opening dimensions only.

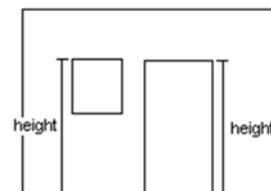
- ◆ **R.O. Size (Wd x Ht)** – This is the exact size of the opening to be placed in the wall. Enter the width of the opening in the first field, and the height in the second. (See the figure at right.)



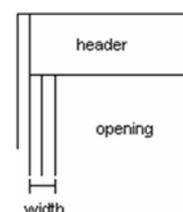
- ◆ **Header Depth** – This is the exact depth of the material that will be used as the header. (See the figure at right.) This value is entered in inches and sixteenths of an inch.



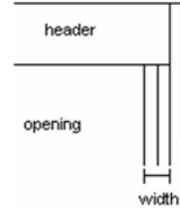
- ◆ **Header Height** – This is the distance from the bottom of the wall to the top edge of the opening. (See the figure at right.)



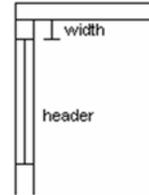
- ◆ **Left Min. Bearing** – This is the total width of any members that will be supporting the left side of the header. (See the figure at right.) This value is entered in inches and sixteenths of an inch.



- ◆ **Right Min. Bearing** – This is the total width of any members that will be supporting the right side of the header. (See the figure at right.) This value is entered in inches and sixteenths of an inch.



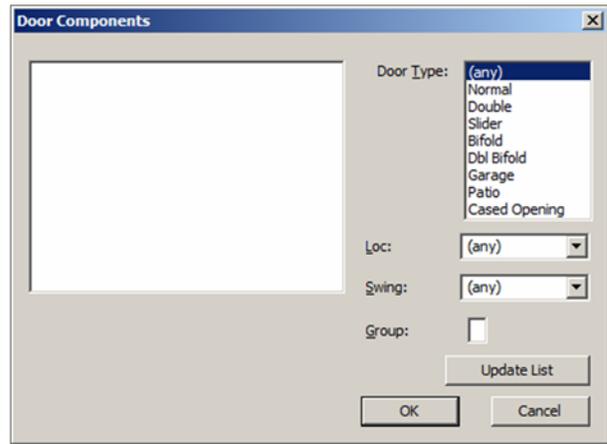
- ◆ **Trim Width** – This is the closest that either end of this opening can get to another opening or an intersection wall. (See the figure at right.) This value is entered in inches and sixteenths of an inch.



- ◆ **No. Plies** –
- ◆ **Product** –
- ◆ **No. Trimmers** –
- ◆ **No. King Studs** –
- ◆ **No. Upper Sills** –
- ◆ **Door Side** – This option is only available if *Door* has been selected as the *Component Type*. The *Door Side* represents the direction that the door will swing. *Inside* and *Outside* refer to the room in which the door is placed.

Command Buttons

- ◆ **Set Component** – Click this button to select a component from the list of doors and windows you have created in KeyMan. The Set Components dialog box is pictured at right. See below for more information. When you first enter this screen, all door or window components are displayed (depending on the component type you chose on the previous screen). To narrow the list, select the component type in the column on the right and/or other options such as Swing or Location. Then click the Update List button to filter the component list based on the selected options.



Highlight the component you wish to place and click the OK button. If no components appear in the list, make sure that the KeyMan® path is set correctly on the File, Set Paths screen.

- ◆ **Set Comp. Assy.** –
- ◆ **Unset Comp.** – When a component is selected, the rough opening size is pre-determined and therefore those fields are grayed out. If you wish to change this door or window from having a component to having a rough opening, click this

button. The Component Part # field will be cleared and the rough opening fields will become available.

- ◆ **Save As Defaults** – Click the Save As Defaults button to store the information in this window and bring it up every time a new header is being defined.
- ◆ **OK** – Click the OK button to apply the information entered in this window and return to the Main Model™ window.
- ◆ **Cancel** – Click the Cancel button to return to the Main Model™ window without making any changes to the selected header or header being placed.
- ◆ **Products** –

Once you have defined the door or window that you want to place, and clicked *OK*, you will be prompted to select a wall in which to place the header. To specify a wall for placement, click the center of the wall. The header placement lines will then appear. Use the keyboard to define the exact location of the header. When the header is in the proper location, press the *Enter* key on your keyboard. You will be returned to your previous view, again being prompted to select a wall in which to place the header. To choose a new header, press the *Escape* key, or click the right mouse button. This will return you to the *Define Component* window. At this point, you can choose a new header to place, then click the *OK* button to place it, or click the *Cancel* button to return to the main window.

- **Move Header** – The *Move Header* command is used to move an existing header to a new location. When you select *Move Header* from the *Macros* menu, you will be prompted to select a header to move. When you click on a header, the header placement lines will appear. Use the keyboard to define the new location for your header. When the location has been defined, press the *Enter* key on your keyboard. You will then be prompted to select another header to move. To return to the main window, click the right mouse button or press the *Escape* key. Note: Clicking the right mouse button or pressing the *Escape* key while a header is selected (while the placement lines are visible) will cause the header to be deleted.
- **Snap Header** – The *Snap Header* macro allows the use of the ends of *Construction Lines* to create snap-points for placement of headers within walls. The *Snap Header* function works in much the same way as *Quick Header*, with some minor differences. To access the *Snap Header* command click on the *Snap Header* option in the *Headers* sub-menu of the *Macro* menu or the *Snap Header* button in the *Header Macros* toolbar.

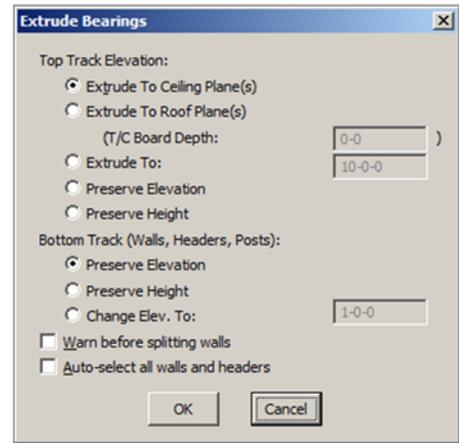
When selected, *Snap Header*, like *Quick Header*, displays the *Define Component* dialog box. When component selection has been completed, again like *Quick Header*, you are asked to select a wall for insertion of the selected component. When a wall has been selected, the typical header placement cursor along with appropriate dimension-line information is displayed. It is at this point that *Snap Header* behavior will differ from *Quick Header*. If *Snap Header* detects either endpoint of a *Construction Line* within the boundaries of the selected wall, the point is added to a list of available snapping points. If any snap points are associated with the selected wall, the header placement cursor will only be allowed to traverse between these snap points as the cursor is dragged using either the mouse or the arrow keys. Except for the Page Up and Page Down keys, the current *Quick Header* keystroke / mouse functionality applies to *Snap Header*, including the End key to change the location along the rough opening where dimensions are drawn to. If *Snap Header* does not detect any available snapping points, the current *Quick Header* drag functionality is employed. Regardless of whether or not snap points are

being used to place the header component, an additional feature of Snap Header allows you to input a specific offset dimension to either side of the target header component. Upon entering Snap Header one of the dimensions on either side of the placement cursor will be highlighted in red. The Page Up and Page Down keys, as well as the middle-mouse scroll wheel, can be used to toggle the highlighted dimension from one side of the placement cursor to the other. Either, clicking the middle mouse button, or pressing the Space Bar key will bring up the Input Header Offset dialog: Inputting a new header offset value and selecting OK will immediately place the header accordingly. Selecting Cancel returns you to the previous placement mode.

- **Verify Components** – The Verify Components command allows you to view and change door and window components used. Select Verify Components from the Headers sub-menu to label each of the component headers. The labels will correspond to the list on the right side of the window. Headers that were entered as rough openings (not as components) will also have a label next to them. Click on a header in the plan view of the model to view or change its properties. To return to the Main Model™ window, click the right mouse button. Enhancements have been made to the Verify Components macro to make it more useful in quality control checking. In Verify Components mode of previous versions of Model, components were labeled together as long as they had the same Component Part Number. Now you can choose additional criteria to group components by including Assigned Product, Header Height, Number of Plies, Rough Opening Size, and Left / Right Minimum Bearing. For Products, if multiple products are assigned to a header, the first product in the product list will be used. To set the criteria, click the new Set Criteria button in Verify Components to access the Set Selection Criteria dialog box shown below. Check the boxes of the properties you would like included in the labeling of components. When in Verify Components mode, you can click on a header to access its properties. If you change any properties, you will now be asked whether you want to "apply updates to all like headers?" If you click yes, those changes will be applied to all like headers with the same label.
- **Global Change Components** – The Global Change Components command allows you to change a door and window component and have that change affect all components of that type in the Model™ (including in any options). Select Global Change Components from the Headers sub-menu and then select one of the components you wish to change. The *Define Component* dialog box (described above) will appear. If the door or window was entered as a component, the Set Component and Unset Comp will be available. Click the Set Component button to change the component for this window or door. Click the Unset Comp. button to delete the component and instead enter rough opening dimensions. When you click OK, the change will affect the selected component as well as all the components that were the same as that one.
- **Delete Header** – The Delete Header command allows you to remove existing headers from the model. When you select Delete Header from the Macros Menu, you will be prompted to select a header to delete. Move your cursor over a header, and click the left mouse button. The header will be removed from the wall. You will then be prompted to select another header to remove. Click the right mouse button or press the Escape key to return to the Main Model™ window.
- **Move Snap Header** –



Extrude Bearings – The Extrude Bearings macro allows you to move the top or bottom of the walls of your model. Selecting the Extrude Bearings command will display the Extrude Bearings window. This window allows you to move the tops or bottoms of your walls to specific heights. Below is the Extrude Bearings window. See below for more information on each field.



TOP TRACK ELEVATION

Extrude to Ceiling Plane(s) – If this option is active (if the button to the left is filled in), you will be prompted to select the ceiling planes that you want the tops of the walls to move to.

Extrude to Roof Plane(s) – If this option is active (if the button to the left is filled in), you will first need to enter the actual width of the roof members (into the T/C Board Width field). You will then be prompted to select the roof planes that you want the tops of the walls to move to. The walls will move to the location of the planes, minus the depth of the T/C members.

T/C Board Depth –

Extrude To – If this option is active (if the button to the left is filled in), you will be prompted for a height to extrude the tops of the walls to (in the field to the right).

Preserve Elevation – If this option is active (if the button to the left is filled in), the current location of the top plate will be held, regardless of the movement of the other

Preserve Height – If this option is active (if the button to the left is filled in), the current distance between the two plates (actual wall height) will be held, regardless of the movement of the other plate.

BOTTOM TRACK (WALLS, HEADERS, POSTS)

Preserve Elevation – If this option is active (if the button to the left is filled in), the current location of the bottom plate will be held, regardless of the movement of the other.

Preserve Height – If this option is active (if the button to the left is filled in), the current distance between the two plates (actual wall height) will be held, regardless of the movement of the other plate.

Change Elev. To – If this option is active (if the button to the left is filled in), you will be prompted for a height to extrude the bottom of the walls to (in the field to the right).

[OTHER]

Warn before splitting walls – If this box is checked, and you attempt to extrude a wall at a point that it will need to break into more than one wall in order to be extruded (i.e. extruding a top plate to a pitch break in a plane), you will be warned that the wall will need to be split.

Auto-select all walls and headers – If this option is checked, all walls on the active level will be extruded when the OK button is clicked.

COMMAND BUTTONS

OK – When you click the *OK* button, you will return to the Main GS Plan window and will be prompted to perform the next step. This may be to select a plane or to select the walls to be extruded.

Cancel – Click the *Cancel* button to abort the command and return to the Main GS Plan window.



Create Veneer Wall – The Create Veneer Wall macro allows you to create a veneer wall on either side of a bearing (to create, for example, a brick ledge). The Define Veneer Wall Properties window allows you to specify which side of the bearing the veneer should be placed as well as define the properties of the veneer wall such as width, location, and whether it is framed or load bearing. Click *OK* and then click on the wall(s) you wish to place veneers on. You will stay in this mode until you right click or press the *Escape* key on your keyboard, and then *Cancel* out of the Veneer Wall Properties box.

Pictured at right is the *Define Veneer Wall Properties* window. See below for more information on each field.

PLACE NEW VENEER WALL AT

Choose which side of the existing wall the veneer wall be placed on – either the side opposite the reference line or the same side as the reference line.

Side Opposite Reference Line –

Side with Reference Line –

NEW VENEER WALL PROPERTIES

Match Heights to Selected Wall – If this option is checked, the veneer will be constructed to be the same height as the wall it is selected to be on. If you uncheck this option, the *Beginning Height* and *Ending Height* fields will become available, allowing you to specify the heights for the veneer to be placed.

Beginning Height –

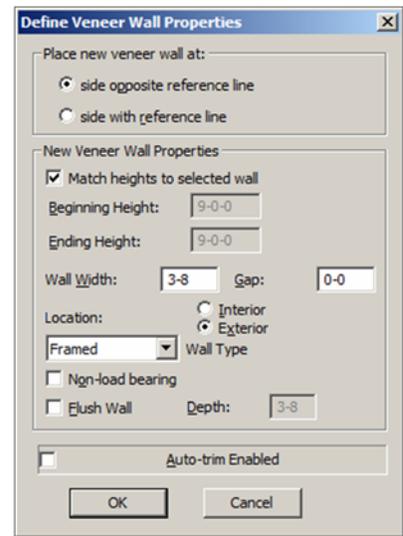
Ending Height –

Wall Width – Enter the width of the veneer wall to be placed, including any air space or sheathing space between the two walls.

Gap –

Location – As in the *Bearing Properties* window, *Interior* or *Exterior* refers to the veneer wall's location in the context of the entire model (if it is an exterior wall or not). It does not in any way affect the side of the wall the veneer is placed (that is controlled by the reference line setting above).

Wall Type –



Non-Load Bearing – If this veneer wall will not support loads transferred from above, the Non-Load Bearing box should be checked.

Flush Wall – When this box is checked, the Depth field will become available so that you can specify the height of the veneer wall above top plate elevation (i.e. the additional height above the beginning and ending height. Generally, it is the depth of the overhead floor joists).

Depth –

Auto-Trim Enabled –

COMMAND BUTTONS

OK – Click the *OK* button when you are finished setting the properties for the veneer wall, and then click on the wall that you want to place the veneer on.

Cancel – Click the *Cancel* button to cancel creating a veneer wall.

Roof Planes – The *Roof Planes* macro contains commands to automatically input and extend roof planes. Selecting the *Roof Planes* option will display the *Roof Planes* sub-menu. This sub-menu contains commands that allow you to enter and extend roof planes. The sub-menu contains the following commands (click on one to jump to that help topic or scroll down to read them all):



Create Planes – When you select *Create Planes*, GS Plan will attempt to place a roof plane on each exterior wall in your model. After selecting the command, you will be prompted to select a boundary for the planes. This means you need to select an enclosed area that planes can be placed around. An example of a closed boundary would be four (or more) walls that form a closed shape. If the shape is open at any point (if there is any section where walls do not connect, leaving a gap) then create planes will not work.

After selecting inside of the boundary, you will be prompted by the *Plane Properties* window to define the planes to be placed. Fill in the information in the window and then click the *OK* button. The plane that you defined will then be placed on every exterior member of the selected boundary.



Extend Manual – The *Extend Manual* command allows you to define the extension path of a selected roof plane. After selecting the command, you will be prompted to pick the plane that you want to extend. When you select it, the plane will extend infinitely in every direction. White lines will be displayed anywhere the plane intersects another plane, or crosses a bearing member. The baseline of the plane (the original line that was input) will turn yellow. Moving in a clockwise fashion, pick each of the white lines that should be used to form an edge of the final plane. Once all of the edges have been selected, click on the baseline, and the plane is finished.



Extend Single – Selecting *Extend Single* will automatically extend one plane. After selecting *Extend Single*, you will be prompted to select the plane that you want extended. As soon as you select it, the plane will be extended based on the locations of other planes, gables, and bearings. It may be necessary to manually extend the plane to get the exact shape that you need in more complex roofs (see above for help on the *Extend Manual* command).



Extend All – The *Extend All* command will automatically extend all of the roof planes in your model. After selecting *Extend All*, all of the planes will be extended based on the locations of other planes, gables, and bearings. It may be necessary to manually extend one or more

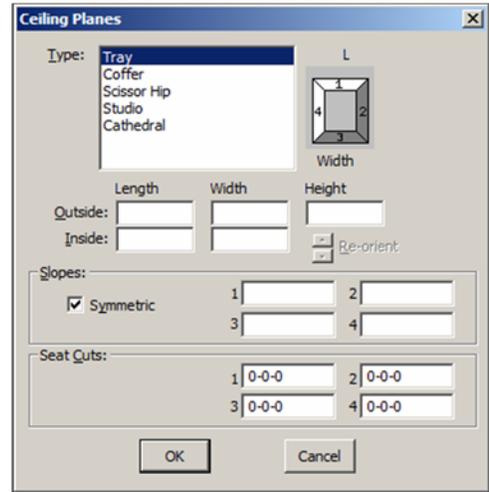
planes to get the exact shape that you need in more complex roofs (see above for help on the *Extend Manual* command).

 **Clip Off Corner –**

 **Five-Sided Bay Window –**

 **Three-Sided Bay Window –**

 **Ceiling Planes –** Select the *Ceiling Planes* option to create and place a ceiling vault in the active level. After choosing the command, you will be prompted to define the geometry of the vault with the ceiling. Below is the *Ceiling Plane Properties* window. See below for more information on each field.



Type –

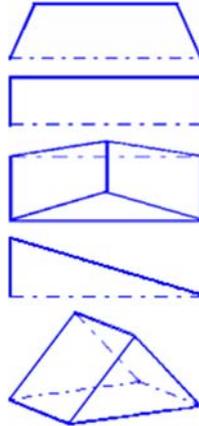
Tray –

Coffered –

Scissor Hip –

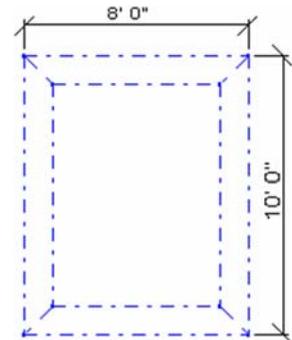
Studio –

Cathedral –



OUTSIDE

Enter the outside dimensions for the ceiling plane. For example, on a tray ceiling the outside dimensions are measured as shown at right. In this case, 10 was entered for the length, 8 for the width. In the Tray example under the Type explanation above, 2 was entered for the height.



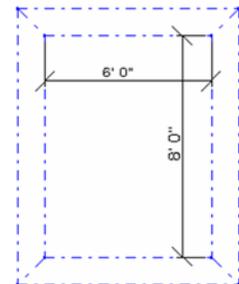
Length –

Height –

Width –

INSIDE

These fields will only be available if you choose either Tray or Scissor Hip from the Type list. For example, on a tray ceiling the inside dimensions are measured as shown at right. In this case, 8 was entered for the length, 6 for the width.



Length –

Width –

SLOPES

The slope will be calculated for you in many instances if you enter the height of the ceiling plane, but if you wish to enter the slopes and have GS Plan calculate the height for you, enter them here.

Symmetric – If this box is checked, GS Plan will ensure that opposite sides of the ceiling plane have the same slopes (so they will be symmetrical).

1 –

2 –

3 –

4 –

SEAT CUTS

This is the amount of the rafter that will sit flat on the bearing when the ceiling plane slopes upward from the wall. In other words, the flat distance from the outside of the wall to the beginning of the slope.

1 –

2 –

3 –

4 –

COMMAND BUTTONS

OK – Click the *OK* button to use the settings from this screen to place a ceiling plane.

Cancel – Click the *Cancel* button to cancel creating a ceiling plane.

Toggle – The *Toggle* macro allows you to view and quickly change certain properties of bearing members. Selecting the *Toggle* option will display the *Toggle* sub-menu. This sub-menu contains commands that allow you to view and change the properties of the bearings on the active level. It contains the following commands:



Toggle Load-Bearing – The *Toggle Load Bearing* function allows you to switch walls on the active level from bearing to non-bearing and back with a single mouse click. Load Bearing walls will be displayed in green. Non-load Bearing walls will be displayed in purple.



Toggle Roof/Floor – The *Toggle Roof/Floor* function allows you to view and change walls on the active level from roof bearing to floor bearing and back by clicking on them. Roof supporting members will be displayed in green. Floor supporting members will be displayed in purple.



Toggle Interior/Exterior – The *Toggle Interior/Exterior* function allows you to switch walls on the active level from interior to exterior and back with a single mouse click. Interior members will be displayed in green. Exterior members will be displayed in purple.



Toggle Wall/Header Widths – The *Toggle Wall/Header Widths* function allows you to view and change the widths of all wall and header members on the current level. Each of the different wall or header width will be displayed in a different color. A key to these colors will be displayed on the right side of the main window.

To change the width of a wall or header, click the width to which you want to change from the sidebar and then click on the wall or header that you want to change. To change a member to a width that is not listed on the sidebar, click the *Other* entry, then enter the new wall width and click the *OK* button. The new width will be listed as an option on the sidebar. To change all members to a specific width, click the *Global Change* button. You will then be prompted to select which types of walls should be changed: *Interior*, *Exterior* or both. Click the *OK* button to change all of the members specified to the currently selected width.



Toggle Overhang Lengths –



Toggle Framed/Non-Framed – The *Framed/Non Framed Toggle* function allows you to switch walls on the active level from framed to non-framed and back with a single mouse click. Framed walls will be displayed in green. Non Framed walls will be displayed in purple.



Toggle Top Plate Elevations – The *Toggle Top Plate Elevation* function allows you to change the heights of walls on the active level by change their top plate elevations. Each existing wall height will be displayed in a different color. To add a new wall height, click the "Other" entry in the list on the right and enter the new top of plate elevation. This number will be added to the list. Select the height you wish to assign from the list on the right and then click on the wall to apply it to. The wall height will change, but the bottom plate will be held in its previous location.



Toggle Wall/Header Heights – The *Toggle Wall/Header Height* function allows you to toggle wall and header heights for the active level. When in *Toggle Wall/Header Height* mode, walls will be visible and colored according to the wall height legend on the right side of the screen. To change wall heights, left-click on the appropriate wall height in the legend, then left-click on each wall to change. To add new wall heights, click on the *Other* button, enter the new wall height in the *Set New Wall Height* dialog box, and click *OK*. To change wall heights globally, changing all wall heights of the active level, click on the appropriate wall height, then click the *Global Change* button, and finally check the appropriate boxes in the *Toggle Wall/Header Values* dialog box and click *OK*.



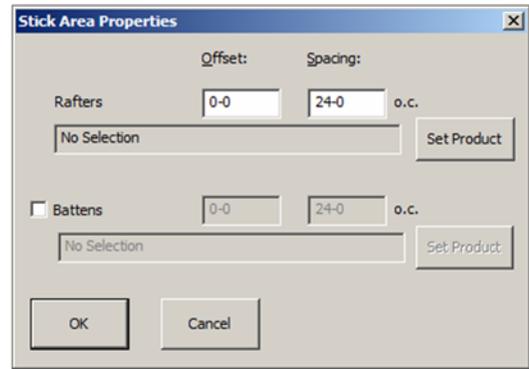
Toggle Bearing Properties – The *Toggle Bearing Properties* function allows you to apply the properties of a selected bearing to other existing bearings. When in *Toggle Bearing Properties* mode, the visibility will change so only bearings are visible. You will be prompted to *Select Template Bearing*, where you will click on an existing bearing to be the template bearing. The *Set Bearings Properties* dialog box will come up for you to verify or change the bearing template properties. These are the properties that will be applied to other bearings. Once the template bearing properties are set, click *OK* to begin applying these properties to other bearings or *Cancel* to not apply the template bearings properties. You will be prompted to *Select Target Bearing*. The properties of each bearing you click on will change to match the templates properties. You can continue to toggle bearing properties until you right-click with the mouse or press the *Escape* key.



Toggle Bears on Earth –

Stick Framing – The Stick Framing macro allows you to quickly input non-structural roof and ceiling framing areas. Selecting the Stick Framing option will display the Stick Framing sub-menu. This sub-menu contains commands that allow you to quickly input non-engineered stick framing members (rafters and ridge members). The sub-menu contains the following commands (click on one to jump to that help topic or scroll down to read them all):

Stick Areas (Roof) – The *Stick Areas (Roof)* macro allows you to select a roof plane in the active level to be framed with roof rafters. When you select *Stick Areas (Roof)* the *Stick Area Properties* window will appear, allowing you to specify the type of material to be used in the area. Once the information for the framing area has been defined, click the *OK* button to choose the roof plane that you want framed. Clicking a plane will create an area inside the roof plane. Click the right mouse button to return to the Main GS Plan window. Below is the *Stick Area Properties* window. See below for more information on each field:



[Rafters] Offset – This macro will pick a start point automatically, generally at a corner or peak point. Entering an *Offset* value will move the members from their standard location.

Spacing O.C. – Enter the on-center spacing to be used in the area.

Set Product – Click *Set Product* to specify a product to be used in the new area.

Battens –

[Battens Offset] –

[Battens Spacing O.C.] –

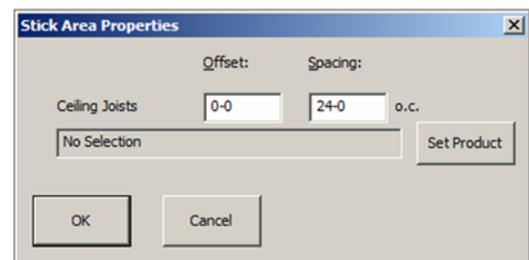
OK – Click the *OK* button to return to the Main GS Plan window to choose which planes will be framed with the area just defined.

Cancel – Click the *Cancel* button to exit the macro and return to the Main GS Plan window.

Stick Areas (Ceiling) – The *Stick Areas (Ceiling)* macro allows you to select a ceiling plane in the active level to be framed with ceiling rafters. When you select *Stick Areas (Ceiling)* the *Stick Area Properties* window will appear, allowing you to specify the type of material to be used in the area. Once the information for the framing area has been defined, click the *OK* button to choose the ceiling plane that you want framed. Clicking a plane will create an area inside the ceiling plane. Click the right mouse button to return to the Main GS Plan window. The *Stick Area Properties* window is pictured at right. See below for more information on each field:

CEILING JOISTS

Offset – This macro will pick a start point automatically, generally at a corner or peak point. Entering an *Offset* value will move the members from their standard location.



Spacing O.C. – Enter the on-center spacing to be used in the area.

Set Product – Click *Set Product* to specify a product to be used in the new area.

OK – Click the *OK* button to return to the Main GS Plan window to choose which planes will be framed with the area just defined.

Cancel – Click the *Cancel* button to exit the macro and return you to the Main GS Plan window.

Ridge Boards (Roof) – The *Ridge Board (Roof)* macro allows you place ridge beam bearing members at any intersection of two roof planes on the active level. When you select *Ridge Boards (Roof)* the *Ridge Board Properties* window will appear, allowing you to specify the type of material to be used for the board. Once the information for the ridge board has been defined, click the *OK* button to choose the intersection that you want framed. Clicking an intersection line will create a board member. Click the right mouse button to return to the Main GS Plan window. The *Ridge Board Properties* window pictured at right. See below for more information on each field:



Ply Width – Enter the width of a single ridge board member.

Plies – Enter the number of plies that you want to use for the ridge board.

Depth – Enter the actual (not nominal) depth of the material to be used as the ridge board.

COMMAND BUTTONS

Set Product – Click the *Set Product* button to choose a specific material to be used as the ridge board.

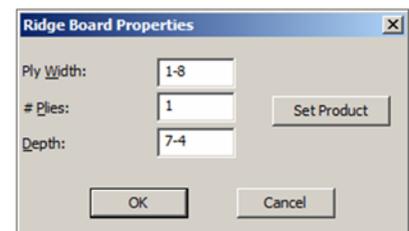
OK – Click the *OK* button to return to the Main GS Plan window and choose a ridge at which you want to place a ridge board.

Cancel – Click the *Cancel* button to exit the macro and return you to the main GS Plan™ window.

Ridge Boards (Ceiling) – The *Ridge Board (Ceiling)* macro allows you place ridge beam bearing members at any intersection of two ceiling planes on the active level. When you select *Ridge Boards (Ceiling)* the *Ridge Board Properties* window will appear, allowing you to specify the type of material to be used for the board. Once the information for the ridge board has been defined, click the *OK* button to choose the intersection that you want framed. Clicking an intersection line will create a board member. Click the right mouse button to return to the Main GS Plan window. The *Ridge Board Properties* window is pictured at right. See below for more information on each field.

Ply Width – Enter the width of a single ridge board member.

Plies – Enter the number of plies that you want to use for the ridge board.



Depth – Enter the actual (not nominal) depth of the material to be used as the ridge board.

COMMAND BUTTONS

Set Product – Click the *Set Product* button to choose a specific material to be used as the ridge board.

OK – Click the *OK* button to return to the Main GS Plan window and choose a ridge at which you want to place a ridge board.

Cancel – Click the *Cancel* button to exit the macro and return you to the main GS Plan™ window.



Skylight – The *Skylight* macro allows you to define and then place a skylight opening in a roof framing area on the active level. After clicking *Skylight*, the *Skylight* macro window will be displayed, allowing you to define the size of the skylight, the component being used (if any), and the manner by which the opening will be supported. Once the *OK* button is clicked, you will be prompted to select the approximate location of the center of the skylight. Use your mouse to click on a point near the center of the opening location. Once the dimension lines appear, use the macro placement keys to define the exact location of the skylight. The Skylight Macro window is pictured at right. See below for more information on each field.



Skylight Length – This is the horizontal width of the skylight opening (distance left-to-right on your screen). Remember, the Skylight Macro places the opening along the slope of the plane, so this dimension is measured along the slope of the roof.

Skylight Width – This is the vertical width of the skylight opening (distance top-to-bottom on your screen). Remember, the Skylight Macro places the opening along the slope of the plane, so this dimension is measured along the slope of the roof.

Component Part # – If you have a database of skylight components created in KeyMan™, you can click the Set Component button to choose one of them to place. Once a component is selected, the component part number will be displayed in this field, and the length and width dimensions will be entered automatically.

[FRAMING SUPPORTS]

Truss-Frame (Truss Girders) – Use this option if you are placing a skylight into a truss framed roof and want Truss Girders supporting the opening.

Stick-Frame (Girders) – Use this option if you are placing a skylight into a conventionally framed roof and want Girders supporting the opening.

Opening Only – Use this option if you do not want any support members automatically placed around the opening.

COMMAND BUTTONS

Set Component – If you have a database of skylight components created in KeyMan, you can click the *Set Component* button to choose one of them to place. Once a component is selected, the component part number will be displayed in this field, and the length and width dimensions will be entered automatically.

OK – Click the *OK* button to place the opening defined above into the active level of the model.

Cancel – Click the *Cancel* button to return to the main window without placing a skylight.



Dormer – The *Dormer* macro allows you to define and input dormer framing. Selecting *Dormer* from the *Macros* menu allows you to define dormer framing in the *Dormer Macro Window* and then place it in the active level of the current model. Place the front of the dormer using the *Macro Placement* keys. The four *Dormer* Types are:

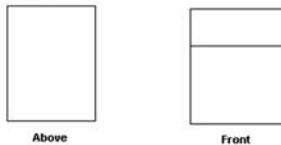
Shed Dormer –

Gable Dormer –

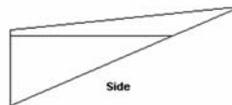
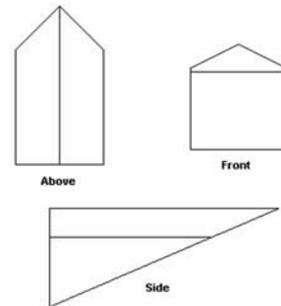
Hip End Dormer –

A Dormer –

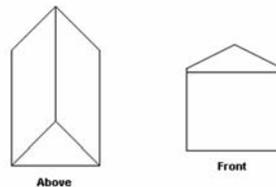
Shed Dormer



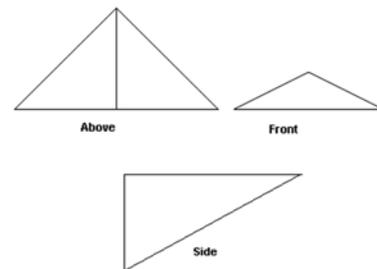
Gable Dormer



Hip End Dormer

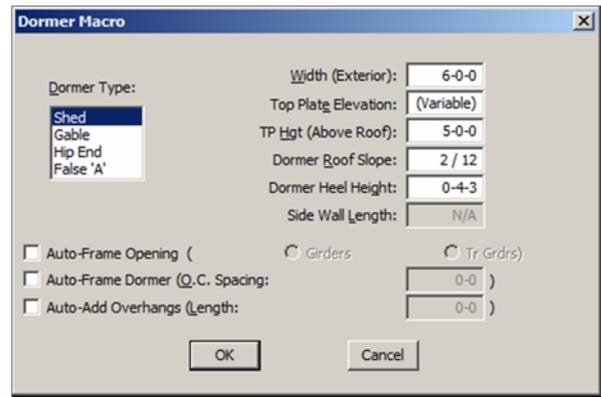


A Dormer

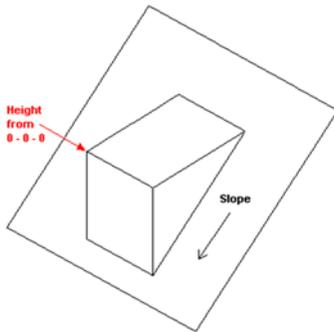


Pictured at right is the *Dormer Macro* window. See below for more information on each field. Note: Based on the *Dormer Type* selected, the information displayed in the *Dormer Macro* window will change.

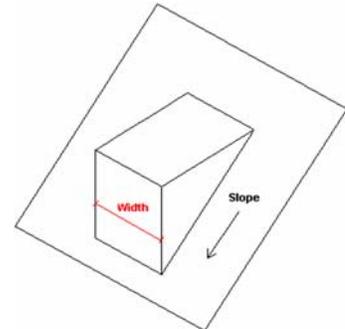
Dormer Type – This list allows you to select the type of dormer you want to define. The rest of the fields in the window will change based upon the *Dormer Type* selected.



Width (Exterior) – This is the dimension across the front (face) of the dormer being defined.



Top Plate Elevation – This is the absolute height of the top of the dormer walls. If this field is left empty or set to (Variable), the final height will be defined by the TP Hgt field, and the placement of the dormer on the existing roof.

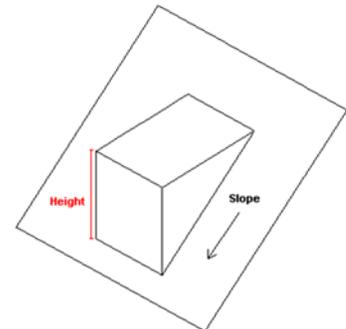


TP Hgt (Above Roof) – This is the vertical dimension of the front wall of the dormer (relative to the top of the existing roof).

Dormer Roof Slope – This is the slope to be used on the dormer roof.

Dormer Heel Height – This is the heel height of the dormer framing members.

Side Wall Length – This is the length of the walls that form the sides of the dormer. If this field is left empty or set to (Variable), the length of the wall will be defined based on the TP Hgt (described above) and the slope of the existing roof.



Auto-Frame Opening – If this box is checked, the opening in the existing roof that is created by the macro will be automatically framed. Choose whether you want GS Plan to frame the opening with:

Girders – For conventionally framed roofs.

Tr Girders – For truss framed roofs.

Auto-Frame Dormer (O.C. Spacing) – If this box is checked, GS Plan will automatically add truss members to the dormer being created. *On Center Spacing* must be defined in the field to the right.

Auto-Add Overhangs (Length) – If this box is checked, overhangs (with their length defined to the right) will be added to the three exposed sides of the dormer.

COMMAND BUTTONS

OK – Click the *OK* button to place the dormer into an existing section of roof.

Cancel – Click the *Cancel* button to clear the information in the window and return to the Main GS Plan window.

How to Place Entities

Macro placement functions are used when placing one of two different entity types. The first is entities that are placed into walls (headers, strong-walls); the second is members that are placed into roof surfaces (skylights, ceiling planes, and dormers). The keyboard keys used for either macro type are the same, but the actions that are performed may vary slightly. The keys used in macro placement are:

Arrow Keys – Press the arrow keys (not the keys on the numeric keypad on your keyboard) to move the member along the wall or surface in which it is being placed. If you are placing a skylight, ceiling, or dormer, the arrow keys allow you to move the member around inside the area or planes selected. Use the four arrow keys to move members up, down, left, and right. To change the value that the member is moving by, use the *Page Up* and *Page Down* keys on your keyboard.

If you are placing a door or window into a wall, the function of the arrow keys changes slightly. The left and right arrow keys continue to move members to the left and right, however, the up and down arrow keys change which side of the wall the header is being placed from. When the dimension lines are drawn for header placement, they start and end at either the outside corner or nearest intersecting wall. In this case, if the placement side of the wall is changed (by pressing the up or down arrow keys) the dimension lines will change. Rather than dimensioning to the outside corners, they will dimension to the inside corners of the intersecting walls.

In this picture, the function of the arrow keys is reversed. The left and right arrow keys will change the side of the wall that the header is being placed, and the up and down arrow keys will move the header along the wall. As above, pressing the *Page Up* and *Page Down* keys on your keyboard will increase or decrease the value that the arrow keys move the header by.

THE PAGE UP KEY

Pressing the *Page Up* key while in macro placement will increase the value of the arrow key movement. When placing a macro member, the default imperial unit that the arrow keys will move by is one inch (1"). Pressing the *Page Up* key will change the increment so that the member is moved by one foot (1') each time an arrow key is pressed. If the member is moving at a sixteenth of an inch at a time, clicking the *Page Up* key will change the increment to one inch; pressing it again will change to one foot. To decrease the value of movement, press the *Page Down* key.

THE PAGE DOWN KEY

Pressing the *Page Down* key while in macro placement will decrease the value of the arrow key movement. When placing a macro member, the default imperial unit that the arrow keys will move by is one inch (1"). Pressing the *Page Down* key will change the increment so that the member is moved by a sixteenth of an inch each time an arrow key is pressed. If the member is moving one foot (1') at a

time, clicking the *Page Down* key will change the increment to one inch; pressing it again will change to a sixteenth. To increase the value of movement, press the *Page Up* key.

THE HOME KEY

Pressing the *Home* key while placing a member will center the member. If you are placing a member into an area or plane, this will move the member so that it is centered (in the area or plane). If you are placing a door or window into a wall, pressing the *Home* key will center the header in the wall.

THE END KEY

Pressing the *End* key while placing a header will change the placement point of the header from the center of the header to one of the ends. Pressing *End* again will change the placement point to the other end. Pressing the *End* key a third time will return the placement point to the center of the header.

Macros Menu (Continued)



Panel Break Mode – The *Panel Break Mode* tools are the first part of a larger effort to offer full floor panelization in GS Plan. These *Panel Break Mode* tools allow you to easily break-up large existing floor framing areas into floor panel sized framing areas, based on your preset floor panel size.

When you are first drawing a framing area for floor panels, you are designing the outer boundary of each area (where each area is the entire region where the framing members run in the same direction) without regard to where panel breaks will occur. Then enter "Panel Break Mode", by clicking on the Panel Break Mode option in the Macros pull-down menu. GS Plan will display the Framing Area boundaries, Openings, and Obstructions, shadows the Construction Lines and Bearings (except non-load bearing walls and headers), and turns off the other member types.

You can right-click in the middle of a framing area to display a pop-up menu with the following commands: *Auto Split*, *Parallel Split*, *Bearing Split*, *Rejoin Panels*, *Split Edge*, *Set Panel ID*, and *Set Nail Edge*.

- Select the Auto Split command to automatically split the Area parallel to the framing, starting at the Parallel Side and creating panels of the preset width across the area.
- Select the Parallel Split command to split the area along a line parallel to the framing at the cursor location, and enter Adjust Split mode (as explained below), allowing the user to fine-tune the split location, if necessary.
- Select the Bearing Split command to split the area along the (non-parallel) bearing that is closest to the cursor location.
- Select the Rejoin command to automatically rejoin the panel your mouse cursor is in with the adjacent panel.
- Select the Split Edge command to split the line between 2 like areas, then drag either of the new edges to the desired location.

You can click the left mouse button on a common panel boundary (whether parallel to the framing or not) and GS Plan will enter Adjust Split mode. In this mode, the panel widths on either side of the selected panel boundary are displayed as "live" dimensions (similar to the ones used in the Quick Header command), and the boundary selected can be dragged from side to side (perpendicular or parallel to the framing) using the same controls as are available in Quick Header,

including using Page Up/Page Down to change the granularity of the arrow keys (feet, inches, and sixteenths, or m, dm, cm, and mm). As the panel boundary is adjusted, the other sides of the two areas are adjusted accordingly, so that the extents of the framing don't change. These adjustments will not allow either area to become less than 0' wide.



Floor Deck Edit Mode –



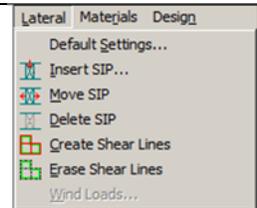
Stretch Model –



Multiple Entity Input –

LATERAL MENU

The *Lateral* toolbar provides users with commands to define information used in the lateral design of the structure.



Default Settings –



Insert SIP – The *Structural Insulated Panel* Insert Strong Wall command allows you to place a Simpson Strong Wall member in your model's walls. After selecting the command, the Select Strong Wall window will appear. Choose the Strong Wall member to be placed and then click OK. Then use the Macro Placement keys to move the Strong Wall to the proper location.

Below is the Select Strong Wall Window. To select a Strong Wall to place in the model, highlight the name in the list and then click the OK button. Click the Cancel button to abort the Insert Strong-Wall command and return to the Main GS Plan window.



Move SIP – This command allows you to move existing *Structural Insulated Panel* Strong Wall members using the Macro Placement commands. After selecting the command, you will be asked to select the Strong Wall that you want to move. Use the mouse to click on the member. Establish the new location, then press the Enter key. Pressing the Escape key at any time will remove the member from the model. To return to the standard GS Plan input mode, press the Escape key or click the right mouse button while no strong walls is selected.



Delete SIP – The Delete *Structural Insulated Panel* Strong Wall command allows you to remove Strong Wall members from your model. After selecting the command, you will be asked to select the Strong Wall that you want to delete. Use the mouse to click on the member. The member will be permanently removed from the model. To return to the standard GS Plan input mode, press the Escape key or click the right mouse button.



Create Shear Lines – The Create Shear Line allows you to define and view the transverse and longitudinal shear lines of your model. To edit the walls used in a shear line, click on the shear line at this point. The walls that are being used as a part of this shear line will highlight. To include a wall in the shear line, move the mouse cursor over a wall and click the left mouse button. Clicking a highlighted wall will exclude it from a shear line. When only the walls needed are highlighted, clicking the right mouse button will return you the shear line selected screen. At this point you can select another shear line to edit, or click the right mouse button a second time to return to the main model window.



Erase Shear Lines – Select Erase Shear Lines to remove all shear lines from the model.

Wind Loads –

MATERIALS MENU

The *Materials* toolbar provides users with commands to assign non-structural materials to specific portions of your model as well as generate material lists to load into KeyMan.



Assign Non-Structural Materials – The *Assign Non-Structural Materials* command allows you to assign expanders that have been defined in KeyMan to specific portions of your model. In other words, you can set up an expander in KeyMan (i.e. an interior wall expander that contains drywall, drywall screws, and paint) and apply those materials to the geometric data created by your model (i.e. apply those materials based on your interior wall square footage). Previously, the only way you could assign non-structural materials (called expanders in KeyMan) to a model was to assign them on the *General Information* screen on the *Other Materials* tab. This method was ineffective because only one expander could be applied to each category for the entire model. The *Assign Non-Structural Materials* option, however, allows you to assign each expander as specifically as necessary, so the interior wall example above could be applied to the living room, but a different expander (i.e. one that contained wallpaper or thicker drywall) could be applied to a different room. The possibilities are endless and allow you complete control over the non- structural materials that will be applied to your model.

Note: this command can only be used if you have created Expanders in KeyMan. If you have not yet done that, please refer to the KeyMan help or contact your salesperson for assistance.

Selecting this command will open the *Geometric Categories* window, pictured at right. This window is used to choose one or more categories to assign materials to your model.

Geometric Categories – Select (highlight) one or more categories to assign non-structural materials.

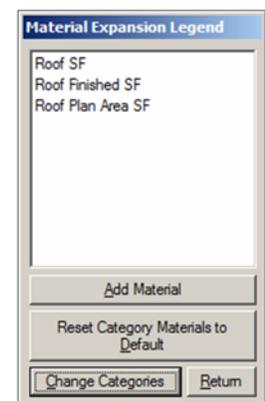
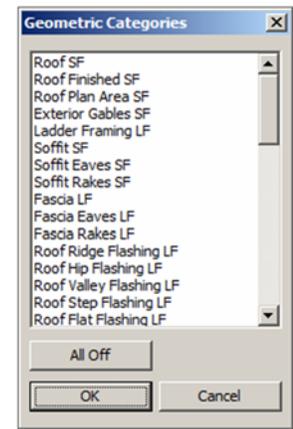
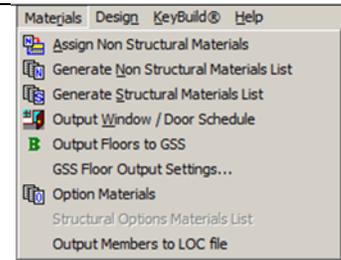
COMMAND BUTTONS

All Off – Click this button to deselect all of the categories in the list above.

OK – Click the *OK* button to move on to the next step and assign non-structural materials to the selected categories (see below for more information).

Cancel – Click the *Cancel* button to exit this screen without assigning non-structural materials to your model.

After you select categories from the *Geometric Categories* screen, click *OK*. The *Material Expansion Legend* window pictured at right will open. When this screen is first opened, the *Geometric Categories* you chose will be displayed. If you selected default non-structural materials on the *Other Materials* tab of the *General Information* screen, those defaults will be listed below the appropriate category. When you add another material (using the *Add Material* button described below), that new material will be displayed in a different color in this box and will correspond to the color displayed on the model.



Once you have the appropriate material showing in this list, select it and then click on the portion of the model to which you wish to assign it. For example, select *Exterior Wall Expander #1* from the list and then click on an exterior wall. That wall will turn green, which is the same color that the expander name is shown in in the list. When you add subsequent materials and apply those to your model (by highlighting them in this list and then clicking on the appropriate portion of the model) their colors will be different, so that you can differentiate materials and where they have been applied.

Command Buttons

Add Material – To add a different material (called an Expander in KeyMan) to the list above so that you can apply it to parts of your model, first highlight the geometric category to which you wish to add the material. Then click this button and select the new material from the drop down that appears on the category name. The new material will be added to the list below the category name and you are now ready to assign it to portions of your model.

Reset Category Materials to Default – If you have mistakenly added materials to the list and assigned them to parts of your model, you may click this button to return the model to its default state (i.e. the default materials you selected on the *Other Materials* tab of the *General Information* screen will be re-applied to the appropriate portions of the model).

Change Categories – Click this button to change which *Geometric Category* (or categories) to which you are applying materials.

Return – Click this button to return to the Main GS Plan window, saving any changes you have made to the materials in your model.

The online document includes an extensive example at this point. “To summarize, here is an example of applying two types of exterior wall expanders to a model...”

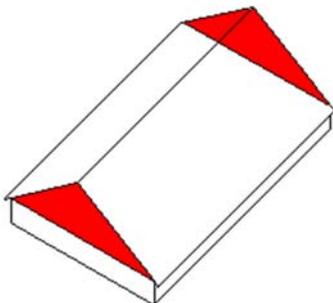
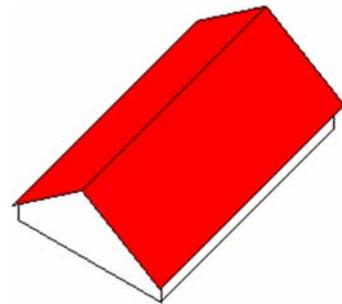
MATERIAL CATEGORY DESCRIPTIONS & MEASURES

Following is a description of the categories including how they are measured and an expander product that might be applied to that geometric data:

Roof SF – This measurement is taken from the surface area (square footage) of roof planes plus the square footage of overhangs in your model. All roof planes including those under overframing areas, are included in this measurement. Openings are not subtracted from the total square footage.

Example Measurement: For a roof that is 30' x 40' with a 6/12 pitch and a 1' overhang, each plane will be about 752 square feet. The total square footage for this roof would be approximately 1,504 square feet.

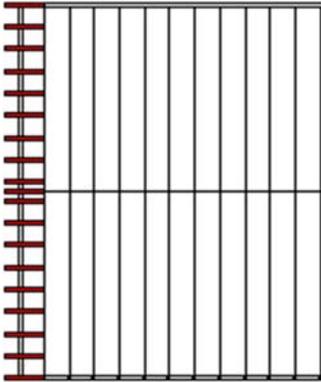
Example Application: Calculating material requirements for ½" roof sheathing.



Exterior Gables SF – This is the exterior square footage of all gables on your job.

Example Measurement: For two gables on a 30' span, 6/12 slope building, the total measurement will be about 225 SF.

Example Application: Calculating material requirements for 7/16" OSB sheathing.



Ladder Framing LF – This measurement is based on the linear footage of ladder panel members generated by gable overhangs (based on the number of pieces and the length of the framing).

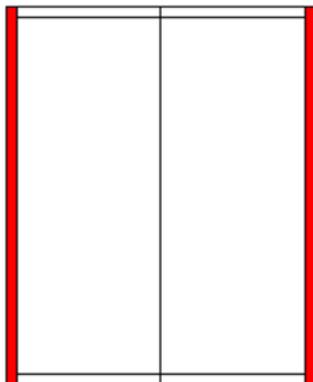
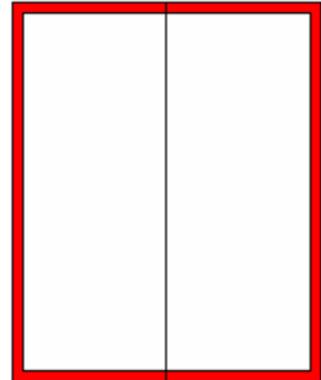
Example Measurement: On a 30-foot gable, with a 1' overhang, each ladder panel will be 3' long (assuming a 24" backspan). If the ladder panels are placed at 24" on-center, and a panel is forced at the peak, 19 panels will be generated. This generates a total linear footage of 57 linear feet of ladder framing.

Example Application: Calculating material requirements for 2 x 8 overhang framing for a gable overhang.

Soffit SF – This measurement is taken from the square footage generated by overhangs. This measurement includes both the Eave and Rake measurements (see below).

Example Measurement: On a 30' x 40' gable roof, with 1' overhangs and a 6/12 pitch, the eaves would generate 42 square feet each (40' length with 1' overhang on each end). The rakes would generate 34 square feet each (since the corner overhang was accounted for by the eave). The total square footage ends up being 152.

Example Application: Calculating material requirements for vinyl soffits.



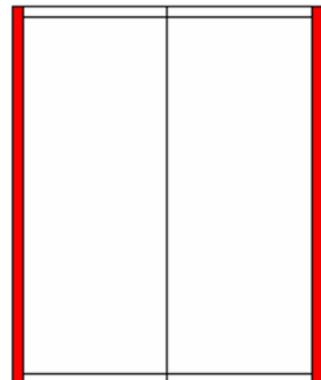
Soffit Eaves SF – This is the square footage of overhang at the tail ends of trusses.

Example Measurement: On a 30' x 40' gable roof with 1' overhangs, the eaves would generate 42 square feet each (40' length with 1' overhang on each end). The total square footage ends up being 84.

Example Application: Calculating material requirements for vinyl soffits.

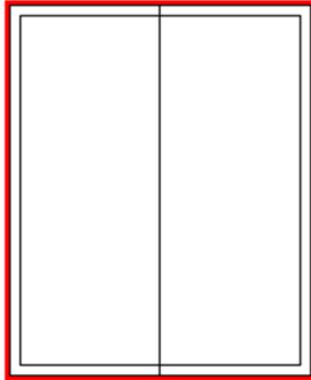
Soffit Rakes SF – This is the square footage of overhang on the ends of gables, taking slope into account.

Example Measurement: On a 30' x 40' gable roof, with 1' overhangs and a 6/12 pitch, the rakes would generate 34 square



feet each (since the corner overhang was accounted for by the eave). The total square footage ends up being 68.

Example Application: Calculating material requirements for vinyl soffits.



Fascia LF – This measurement is taken from the linear measurement of all overhang lines. This measurement includes both the Eave and Rake measurements. Lines are not extended for exterior corners, but overlap for interior corners, so the measurement should average out. This measurement does not account for angled cuts or fascia width.

Example Measurement: If a 30' by 40' house has 6/12 pitch roof with an overhang of 1', the total measurement would be 156 LF (which also takes into account the pitch of the roof).

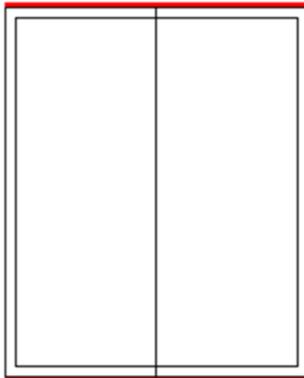
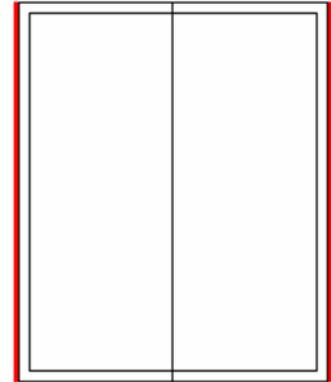
Example Application: Calculating material requirements for face

board trim.

Fascia Eaves LF – This is the linear feet of fascia measured along the eaves (linear footage of overhangs at the tail ends of trusses). This measurement does not account for angled cuts or fascia width.

Example Measurement: On a 20' by 30' gable roof with a 2' overhang, 68 linear feet of fascia would be generated.

Example Application: Calculating material requirements for face board trim.



Fascia Rakes LF – This is the linear feet of fascia measured along the rakes (linear footage of overhangs at gable ends). This measurement does take into account the slope of the roof, but does not account for angled cuts or fascia width.

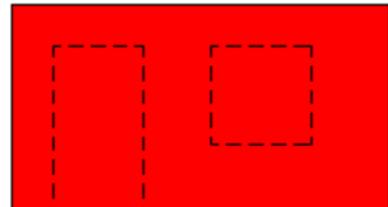
Example Measurement: On a house with a 30' span, 1' overhang, and 6/12 slope, the actual measurement on the end would be about 36'.

Example Application: Calculating material requirements for face board trim.

Exterior Wall Full SF – This measurement is the square footage of the exterior sides of all the exterior walls in your model. This measurement does not subtract the square footage of door and window openings from the total measurement.

Example Measurement: An 8' wall 10' long will come out to 80 SF, regardless of openings.

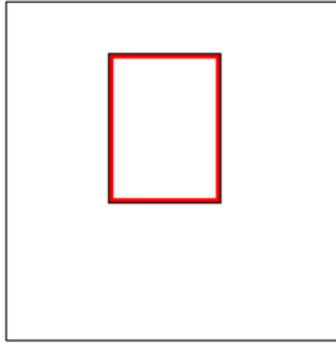
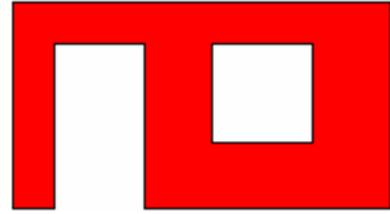
Example Application: Calculating material requirements for 7/16" OSB sheathing.



Exterior Wall Part SF – This measurement is the square footage of the exterior sides of all the exterior walls in your model minus the square footage of door and window openings in these walls.

Example Measurement: An 8' wall 10' long will come out to 80 SF. If the wall has a 3' by 4' window and a 3' by 6' door (equaling 30 SF) your final measurement will be 50 SF.

Example Application: Calculating material requirements for aluminum siding.



Exterior Window Trim LF – This is calculated by adding the measurements for the top, bottom, and sides of all exterior window rough openings. This measurement does not take into account angled cuts or trim width.

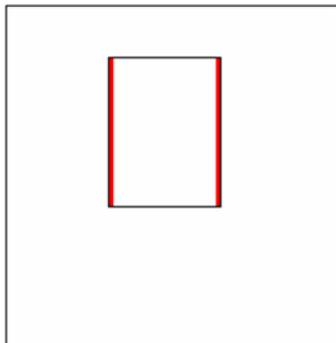
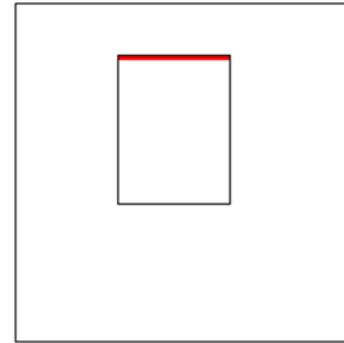
Example Measurement: A window with a rough opening of 3' by 4' would have a final value of 14 LF.

Example Application: Calculating material requirements for 1 x 4 wood trim.

Exterior Window Trim Top LF – This is calculated by adding the widths of all window openings on exterior walls. This measurement does not take into account angled cuts or trim width.

Example Measurement: A structure with 10 three-foot wide window openings would generate 30 linear feet of expander.

Example Application: Calculating material requirements for 1 x 4 wood trim.



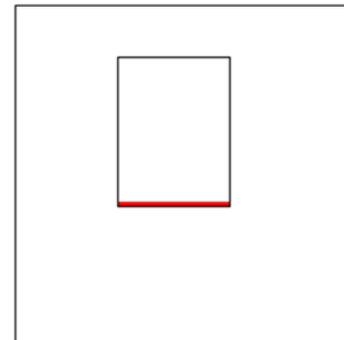
Exterior Window Trim Sides LF – This is calculated by adding the linear feet of the side dimensions of all window openings on exterior walls. This measurement does not take into account angled cuts or trim width.

Example Measurement: A structure with five openings that are four feet tall would have a total of 40 linear feet of expander (each opening has two 4 foot tall sides, so 8 total linear feet per opening times five equals 40).

Example Application: Calculating material requirements for 1 x 4 wood trim.

Exterior Window Trim Bottom LF – This is calculated by adding the linear footage of the bottom of all window openings on exterior walls. This measurement does not take into account angled cuts or trim width.

Example Measurement: A structure with eight openings that are four feet wide would generate a total of 32 linear feet.

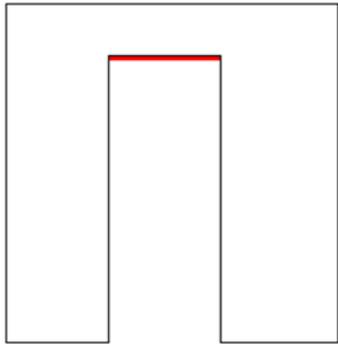
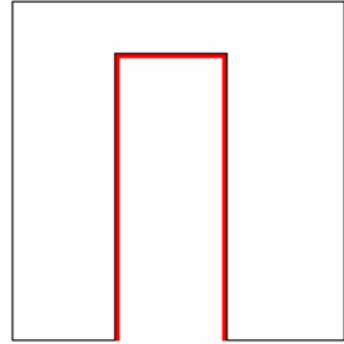


Example Application: Calculating material requirements for 1 x 4 wood trim.

Exterior Door Trim LF – This is calculated by adding the measurements for the top and sides of all exterior door rough openings. This measurement does not take into account angled cuts or trim width.

Example Measurement: A three-foot wide, seven-foot tall door would generate 17 linear feet of expander.

Example Application: Calculating material requirements for 2 x 4 wood bucks.



Exterior Door Trim Top LF – This is calculated by adding the linear footage of the top dimensions of all door openings on exterior walls. This measurement does not take into account angled cuts or trim width.

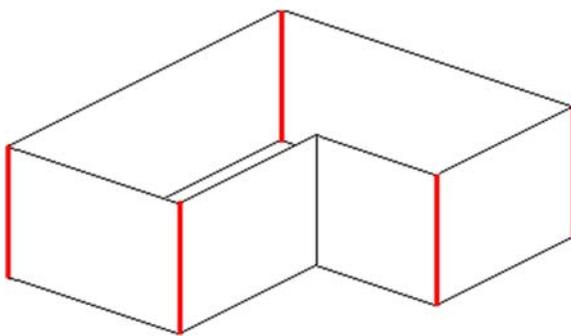
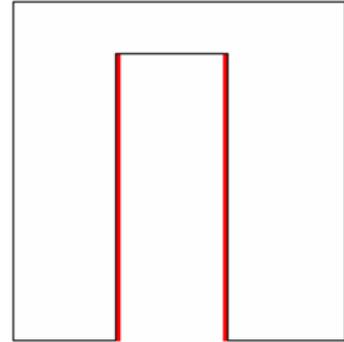
Example Measurement: A structure with 2 three-foot wide exterior doors and 1 six-foot wide door would generate 12 linear feet of expander.

Example Application: Calculating material requirements for 2 x 4 wood bucks.

Exterior Door Trim Sides LF – This is calculated by adding the linear footage of the side dimensions of all door openings on exterior walls. This measurement does not take into account angled cuts or trim width.

Example Measurement: One 7' tall door would generate 14 linear feet of expander.

Example Application: Calculating material requirements for 2 x 4 wood bucks.

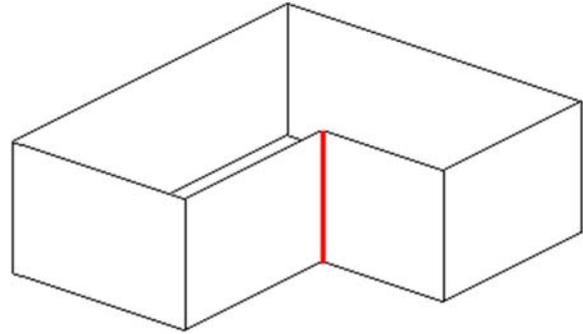


Exterior Outside Corners LF – This is the total number of protruding corners on the outside of exterior walls multiplied by the height of the wall. This measurement is for the corner only, so if materials are needed for both sides of the corners, this number will need to be doubled.

Example Measurement: On an L shaped house there are five outside corners. If the wall height is 8', then the total for outside corners would be 40 LF.

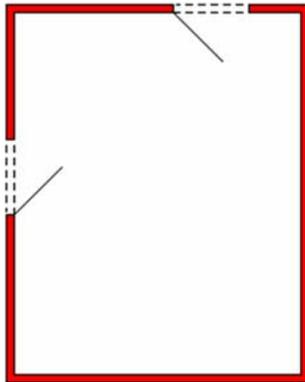
Example Application: Calculating material requirements for corner trim.

Exterior Inside Corners LF – This is the total number of inward turning corners on the outside of exterior walls multiplied by the height of the wall. This measurement is for the corner only, so if materials are needed for both sides of the corners, this number will need to be doubled.



Example Measurement: On an L-shaped house there is one inside corner. If the wall height is 8', then the total for inside corners would be 8 LF.

Example Application: Calculating material requirements for corner trim.



Exterior Wall Part LF – This is the total linear footage of external walls (as measured along the bottom of the wall) minus the linear footage of door openings.

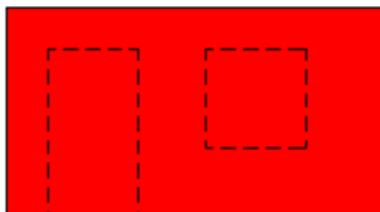
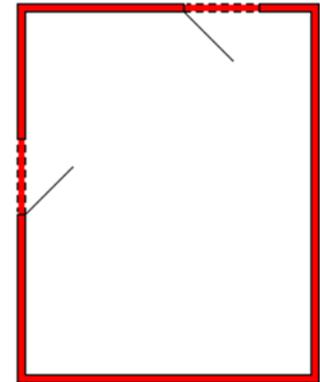
Example Measurement: If a house measures 30' by 40' and has two 3' doors, then the total measurement of exterior wall is 134 LF.

Example Application: Calculating material requirements for starter strip or angle iron.

Exterior Wall Top Plate LF – This is the total linear footage of exterior wall, as measured along the top of the wall to account for raked walls. Door openings are not removed from the total measurement.

Example Measurement: If a house measures 30' by 40', then the total linear footage would be 140, regardless of door openings.

Example Application: Calculating measurements for a second top plate, to be attached to panels in the field.

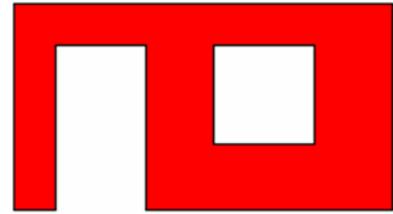


Interior Wall Full SF – This measurement is taken from one side (the inside) of the exterior walls in your model and from both sides of your interior walls. This measurement does not subtract the square footage of door and window opening from the total measurement.

Example Measurement: An 8' tall exterior wall 10' long will come out to 80 SF, regardless of openings. However, an 8' tall interior wall 10' long will come out to 160 SF, regardless of openings. Therefore the total square footage reported for these two walls would be 240 Sq. Ft.

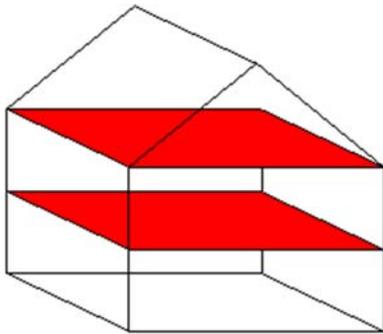
Example Application: Calculating material requirements for interior drywall.

Interior Wall Part SF – This measurement is taken from one side (the inside) of the exterior walls in your model and from both sides of your interior walls. This measurement subtracts the square footage of door and window openings from the total measurement.



Example Measurement: An 8' exterior wall 10' long will come out to 80 SF. If the wall has a 3' by 4' window and a 3' by 6' door (totaling 30 SF of openings) your final measurement will be 50 SF. An 8' interior wall 10' long with the same openings (totaling 30 SF of openings) will generate 100 SF (because both sides of the wall are measured).

Example Application: Calculating material requirements for interior wall paint.

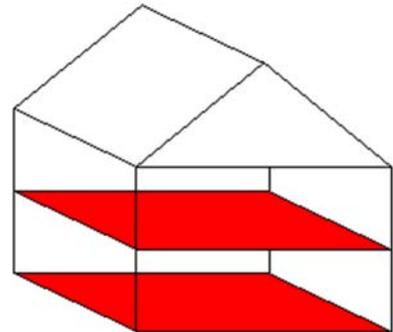


Ceiling SF – This measurement is based on the square footage of ceiling planes in your model, so you must have ceiling planes defined for the area to be calculated. Stair or roof openings are not subtracted from this footage.

Example Measurement: In a two story, 10' x 20' structure, each level would generate 200 square feet of ceiling, thus the entire model would generate 400 square feet total.

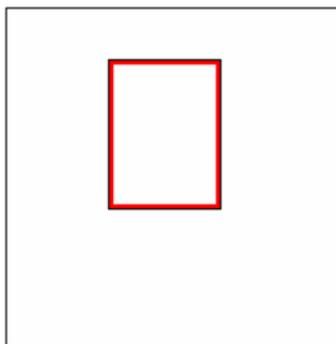
Example Application: Calculating material requirements for drywall.

Floor SF – This measurement is taken from the surface of floor areas in your model. This footage is the same regardless of stair openings.



Example Measurement: In a two story, 10' x 20' structure, each floor generates 200 square feet of floor. The entire structure generates 400 square feet of floor.

Example Application: Calculating material requirements for $\frac{3}{4}$ " OSB sheathing.



Interior Window Trim LF – This is calculated by adding the measurements for the top, bottom, and sides of all interior window rough openings. This measurement will not account for angled cuts or trim width. Dimensions will be added once for exterior windows (for the inside of the wall) and twice for interior walls (both sides of the wall if there are windows in an interior wall).

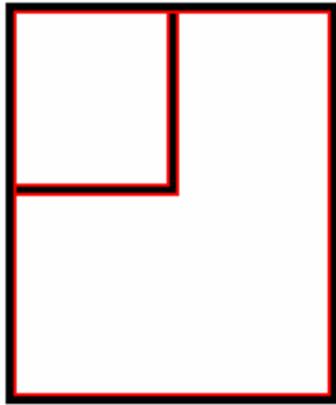
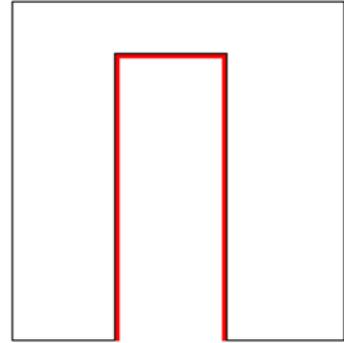
Example Measurement: A 3' by 4' window opening would generate 14 linear feet of expander.

Example Application: Calculating material requirements for 1 x 4 wood trim.

Interior Door Trim LF – This is calculated by adding the measurements for the top and sides of all interior door rough openings. This measurement will not account for angled cuts or trim width. Dimensions will be added once for exterior doors (for the inside of the wall) and twice for interior doors (both sides of the wall).

Example Measurement: A seven-foot tall, three foot wide door opening would generate 17 linear feet of expander.

Example Application: Calculating material requirements for 2 x 4 wood bucks.



Interior Wall Full LF – This measurement is calculated by adding the total linear footage of the interior side of exterior walls and two times the linear footage of interior walls (to account for both sides of the interior walls). The measurement is taken from the bottom of the walls and includes the measurements for all openings.

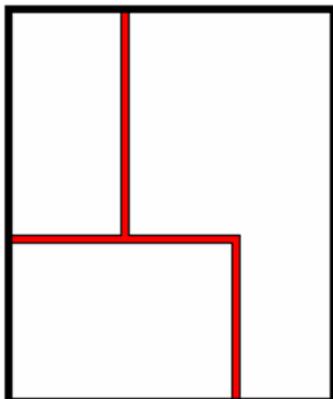
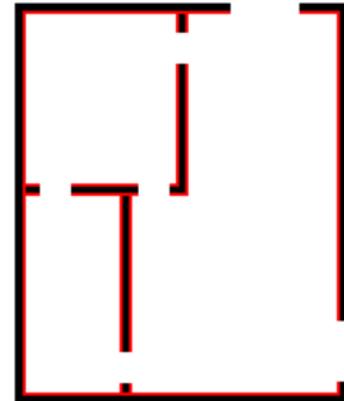
Example Measurement: If the exterior of a structure is 30' by 40', 140 linear feet of expander is generated. If there is 45 linear feet of interior wall, then an additional 90 feet of expander is generated, producing a total of 230 linear feet.

Example Application: Calculating material requirements for crown moulding.

Interior Wall Part LF – This measurement is calculated by adding the total linear footage of the interior side of exterior walls and two times the linear footage of interior walls (to account for both sides of the interior walls). The measurement is taken from the bottom of the walls and does not include the measurements for openings.

Example Measurement: A structure with 30' by 40' of exterior wall, with two 3' doors would generate 134 linear feet. If there is 45 feet of interior wall, with four 2' doors, 74 more linear feet are generated ($45 - 8 = 37$, $37 \times 2 = 74$). The total linear footage generated is 208.

Example Application: Calculating material requirements for base trim.



Interior Wall Top Plate LF – This is the total linear foot measurement of interior walls, which includes the interior side of exterior walls and both sides of interior walls. This measurement is taken along the top of the wall so that it takes into account raked walls, and includes the length along all openings.

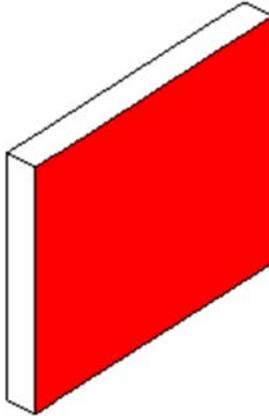
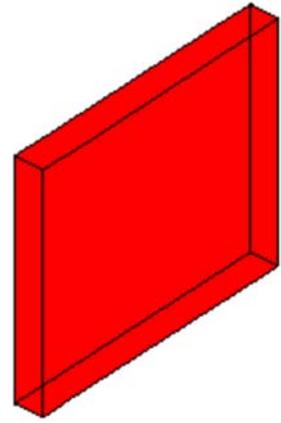
Example Measurement: Three interior walls, one 10', one 17', and one 9' would generate a total of 36 linear feet.

Example Application: Calculating material requirements for crown moulding.

Foundation CF – This footage is calculated by generating the total volume of all non-frame walls. If sixteenth inch walls are entered in Model (usually to pass loads for option walls), these walls are also included in the measurement. This category cannot be assigned using the *Assign Non-Structural Materials* command in GS Plan.

Example Measurement: An 8" wide, 10' tall, 20' long wall would have a final measurement of about 134 CF.

Example Application: Calculating material requirements for concrete for foundation walls.



Foundation SF – This footage is calculated by generating the surface area for one side of all non-frame foundation walls. If sixteenth inch walls are entered in GS Plan (usually to pass loads for option walls), these walls are also included in the measurement.

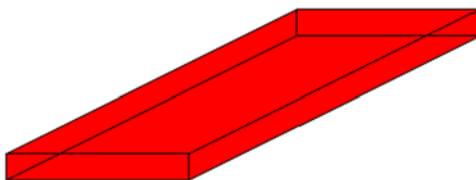
Example Measurement: A 10' tall, 20' long wall would have a final measurement of 200 SF, regardless of wall thickness.

Example Application: Calculating material requirements for foundation sealant.

Foundation LF – This footage is calculated by generating the linear footage for all non-frame foundation walls. If sixteenth inch walls are entered in GS Plan (usually to pass loads for option walls), these walls are also included in the measurement.

Example Measurement: A 20' by 30' house would have 140 LF of foundation wall, regardless of wall height or thickness.

Example Application: Calculating material requirements for sill sealer.



Slab CF – This is the total volume of all Concrete Slab areas used in your model.

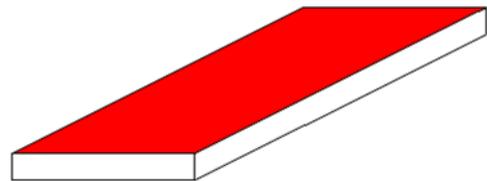
Example Measurement: A 10' by 20' slab that is 4" deep will have a final measurement of about 67 CF.

Example Application: Calculating material requirements for concrete mix.

Slab SF – This is calculated using the surface of all Concrete Slab areas in your model.

Example Measurement: A 10' by 20' slab will have a final measurement of 200 SF, regardless of depth.

Example Application: Calculating material requirements for carpet.





Generate Non-Structural Materials List – Choose this command to generate the list of geometric data and non-structural materials that have been applied to that data in this model. The files (Level_10.mmn, for example) are level specific and can be used in KeyMan® to load the geometric data and expanders from this job for use in your bid.



Generate Structural Materials List – Choose this command to generate a list of structural materials that exist in this model (i.e. beams, girders). The files (Level_10.mms, for example) are level specific and can be used in KeyMan® to load the structural materials from this job for use in your bid. Remember that these members have not been engineered and therefore we recommend running the job through Structure™ before loading the structural materials in KeyMan®. If you run the job through Structure™, these files will be overwritten during the Select Solutions process.



Output Window/Door Schedule – This feature allows you to generate and output a schedule of all the doors and windows in an open job. The schedule is separated into groups of Doors and Windows per level in the job. The schedule includes the Count or Quantity, Component, Product, Plies, Rough opening width, Rough opening height, Sill height, and Header.

If one product is assigned to an opening header, that product will appear in the schedule. If multiple products are assigned to an opening header the first product in the product list will be listed in the schedule. If no product is assigned to an opening header, the product will be listed as 'No Selection' and the Plies will not be right.

To generate the schedule click on the Output Window / Door Schedule option in the Materials pull-down menu or toolbar.



Output Floors to GSS –

GSS Floor Output Settings –



Option Materials Output –

Structural Options Materials List –

Output Members to LOC File –

DESIGN MENU

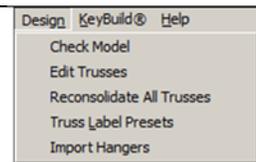
Check Model –

Edit Trusses –

Reconsolidate All Trusses –

Truss Label Presets –

Import Hangers –



GS BUILD MENU

The GS Plan menu allows you to quickly switch from one GS Plan® program to another. Simply click on one of the available programs from the menu, and GS Plan will save the current job and close, then open the current level of the current job in the GS Plan® program selected. Please note that if you choose KeyMan® or KeyBeam®, GS Plan will remain open.



GS Beam – ...opens the GS Beam application...



GS Truss – ...opens the GS Truss application...



GSS Output –

HELP MENU



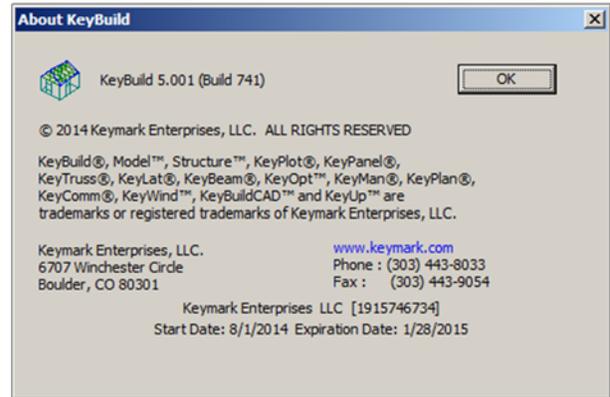
Contents – Displays the Table of Contents for the GS Plan 5.0 Designer's Manual.

What's New – Displays the Release Notes for GS Plan 5.0, including enhancements and

Visit Website – Launches your internet browser and automatically navigates to the Keymark home page.



About GS Plan – Displays the About information box pictured at right.



Plan Toolbars

The 20 Plan toolbars contain a subset of the most commonly used commands and functions available from the Plan menu bar. Toolbars can be added or subtracted from the Main GS Plan Window in two ways:

1. Click on the Tools menu and scroll down to the Toolbars option. On the sub-menu that appears to the right, click on the toolbar that you want to add or remove from your view (those toolbars that are already showing will have a checkmark next to them in the sub-menu).
2. Right click in a gray area of the GS Plan Window (for example, the gray area to the right of the Help menu). On the menu that appears, click on the toolbar you want to add or subtract from your view (those toolbars that are already showing will have a checkmark next to them).

THE ANNOTATIONS TOOLBAR

The *Annotations* toolbar provides quick access to commands for...



The *Annotations* toolbar.

 **Model Entity Labels –**

 **Draw Dimension Lines –** A Dimension Line is an annotation that is used to display the distance between two points. Dimension Lines are saved with the job and included in Layout Plots generated from GS Plan. Please note that they will not be included in plots from GS Plan Structure. To draw a *Dimension Line*, select the Draw Dimension Lines command from the Entities | Annotations pull-down menu or from the new Annotations toolbar.

 **Point-to-Point Dimension Lines –** A Point-to-Point Dimension Line displays the actual distance between two selected end-points and is drawn parallel to a line connecting the selected end-points.

 **Horizontal Dimension Lines –** A Horizontal Dimension Line displays the Horizontal distance between two selected end-points

 **Vertical Dimension Lines –** A Vertical Dimension Line displays the Vertical distance between two selected end-points.

 **Auto Dimensions Filter –**

 **Draw Arbitrary Lines –**

 **Draw or Edit Text Box –**

 **Edit Annotation Attributes –**

THE HEADER MACROS TOOLBAR

The *Header Macros* toolbar provides users with commands to...



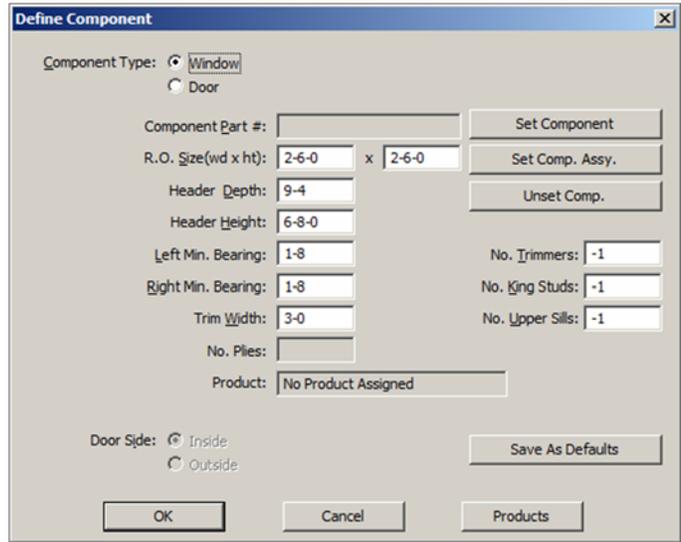
The *Header Macros* toolbar.

 **Quick Header –** The *Quick Header* command allows you to place headers into interior and exterior framed walls. Executing *Quick Header* will first display the *Define Component* window, allowing you to define the opening that you want to place. Once you have defined the door or window that you want to place, and clicked *OK*, you will be prompted to select a wall to place the header in. To specify a wall for placement, click on the center of the wall. The header placement lines will then appear.

Use the keyboard to define the exact location of the header. When the header is in the proper location, press the *Enter* key. You will be returned to your previous view, again being prompted to select a wall to place the header in.

To choose a new header, press the *Escape* key or click the right mouse button. This will return you to the *Define Component* window. At this point, you can choose a new header to place, then click the *OK* button to place it, or click the *Cancel* button to return to the Main window.

Below is the *Define Component* Window. See below for more information on each field.

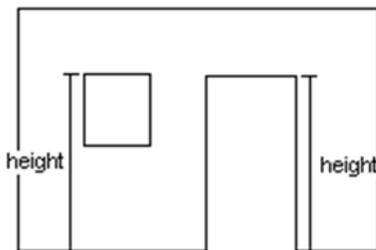
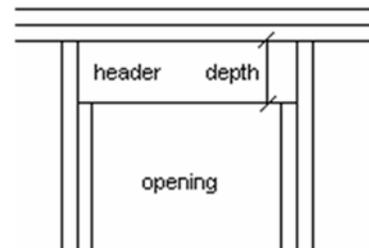


Component Type – Choose the type of opening component you want to use, Door or Window. The fields that are available below will change based on the Component Type you select.

Component Part # – This is the name of the component that you are currently viewing. To change, or choose a component, click the Define Component button. Note: it is not necessary to define a component, you may input an opening based on rough opening dimensions only.

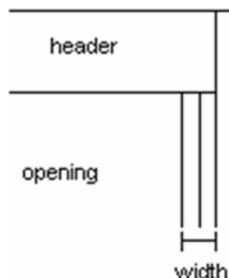
R.O. Size (wd x ht) – This is the exact size of the opening to be placed in the wall. Enter the width of the opening in the first field, and the height in the second.

Header Depth – This is the exact depth of the material that will be as the header. This value is entered in inches and sixteenths of an inch.

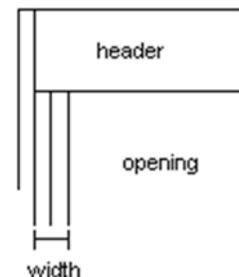


Header Height – This is the distance from the bottom of the wall to the top edge of the opening.

Left Min. Bearing – This is the total width of any members that will be supporting the left side of the header. This value is entered in inches and sixteenths of an inch.



Right Min. Bearing – This is the total width of any members that will be supporting the right side of the header. This value is entered in inches and sixteenths of an inch.



Trim Width – This is the closest that either end of this opening can get to another opening or an intersection wall. This value is entered in inches and sixteenths of an inch.

No. Plies –

Product –

No. Trimmers –

No. King Studs –

No. Upper Sills –

Door Side – This is the direction that the door should swing. *Inside* and *Outside* refer to the room that the door is placed into.

COMMAND BUTTONS

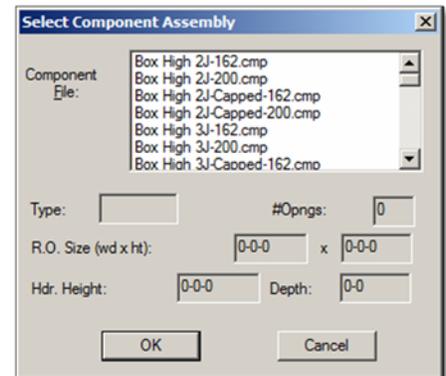
Set Component – Click this button to select a component from the list of doors and windows you have created in KeyMan. Below is the *Set Components* window. See below for more information.



When you first enter this screen, all door or window components are displayed (depending on the component type you chose on the previous screen). To narrow the list, select the component type in the column on the right and/or other options such as Swing or Location.

Then click the Update List button to filter the component list based on the selected options. Highlight the component you wish to place and click the OK button. If no components appear in the list, make sure that the KeyMan® path is set correctly on the File, Set Paths screen.

Set Comp. Assy. – Clicking on the *Set Component Assembly* command button opens the *Set Component Assembly* dialog box pictured at right.



Component File –

Type –

Openings –

R.O. Size (wd x ht) –

Hdr Height –

Hdr Depth –

Unset Comp. – When a component is selected, the rough opening size is pre-determined and therefore those fields are grayed out. If you wish to change this door or window from having a component to having a rough opening, click this button. The Component Part # field will be cleared and the rough opening fields will become available.

Save As Defaults – Click the Save As Defaults button to store the information in this window and bring it up every time a new header is being defined.

OK – Click the OK button to apply the information entered in this window and return to the Main window.

Cancel – Click the Cancel button to return to the Main window without making any changes to the selected header or header being placed.

Products –



Move Header – The Move Header command allows you to move existing headers. When you select Move Header from the Macros menu, you will be prompted to select a header to move. When you click on a header, the header placement lines will appear. Use the keyboard to define the new location for the header. When the location has been defined, press the Enter key. You will then be prompted to select another header to move. To return to the main GS Plan™ window, click the right mouse button or press the Escape key.

Note: Clicking the right mouse button or pressing the Escape key while a header is selected (while the placement lines are visible) will cause the header to be deleted.



Delete Header – The *Delete Header* command is used to remove existing headers from a model. When you select *Delete Header* from the *Macros* menu, you will be prompted to select a header to delete. Move your cursor over a header, and click the left mouse button. The header will be removed from the wall. You will then be prompted to select another header to remove. Click the right mouse button or press the *Escape* key to return to the Main window.



Snap Header – The *Snap Header* macro allows the use of the ends of Construction Lines to create snapping points for placement of headers within walls. *Snap Header* works in much the same way as *Quick Header* with some minor differences. To access the Snap Header command click on the Snap Header option in the Headers sub-menu of the Macro pull-down menu or the Snap Header button in the Header Macros toolbar.

When selected, *Snap Header*, like *Quick Header*, displays the *Define Component* dialog. When component selection has been completed, again like *Quick Header*, you are asked to select a wall for insertion of the selected component. When a wall has been selected, the typical header placement cursor along with appropriate dimension line information is displayed. It is at this point that *Snap Header* behavior will differ from *Quick Header*.

If *Snap Header* detects either endpoint of a *Construction Line* within the boundaries of the selected wall, the point is added to a list of available snapping points. If any snap points are associated with the selected wall, the header placement cursor will only be allowed to traverse between these snap points as the cursor is dragged using either the mouse or the arrow keys.

Except for the *Page Up* and *Page Down* keys, the current *Quick Header* keystroke/mouse functionality applies to *Snap Header*, including the *End* key to change the location along the rough opening where dimensions are drawn.

If *Snap Header* does not detect any available snapping points, the current *Quick Header* drag functionality is employed.

Regardless of whether or not snap points are being used to place the header component, an additional feature of *Snap Header* allows you to input a specific offset dimension to either side of

the target header component. Upon entering *Snap Header*, one of the dimensions on either side of the placement cursor will be highlighted in red. The Page Up and Page Down keys, as well as the middle-mouse scroll wheel, can be used to toggle the highlighted dimension from one side of the placement cursor to the other. Either, clicking the middle mouse button, or pressing the Space Bar key will bring up the **Input Header Offset dialog**:



Header Products –



Verify Components – The *Verify Components* command allows you to view and change door and window components used. Select *Verify Components* from the *Headers* sub-menu to label each of the component headers. The labels will correspond to the list on the right side of the window. Headers that were entered as rough openings (not as components) will also have a label next to them. Click on a header in the *Plan View* of the model to view or change its properties. To return to the Main window, click the right mouse button.

Enhancements have been made to the *Verify Components* macro to make it more useful in quality control checking. In *Verify Components* mode of previous versions of GS Plan, components were labeled together as long as they had the same *Component Part Number*. Now you can choose additional criteria to group components by including *Assigned Product*, *Header Height*, *Number of Plies*, *Rough Opening Size*, and *Left/Right Minimum Bearing*. For *Products*, if multiple products are assigned to a header, the first product in the product list will be used. To set the criteria, click the new *Set Criteria* button in *Verify Components* to access the *Set Selection Criteria* dialog box shown below. Check the boxes of the properties you would like included in the labeling of components.



When in *Verify Components* mode, you can click on a header to access its properties. If you change any properties, you will now be asked whether you want to "apply updates to all like headers?" If you click yes, those changes will be applied to all like headers with the same label.



Change Components – The *Global Change Components* command allows you to change a door and window component and have that change affect all components of that type in your model (including in any options). Select *Global Change Components* from the *Headers* sub-menu and then select one of the components you wish to change. The *Define Component* window (described above) will appear. If the door or window was entered as a component, the *Set Component* and *Unset Comp* will be available. Click the *Set Component* button to change the component for this window or door. Click the *Unset Comp* button to delete the component and instead enter rough opening dimensions. When you click *OK*, the change will affect the selected component as well as all the components that were the same as that one.



Move Snap Header –

THE GS BUILD TOOLBAR

The *Keymark* toolbar provides users with commands to...



GS Beam



GS Truss



The *GS Build* toolbar.



GSS Output

THE OPTIONS TOOLBAR

The *Options* toolbar provides users with commands to...



Create/Edit Option –



Select Options –



Copy/Xfer Option –



Copy Selected Entities Option –



Option Materials Output –



The *Options* toolbar.

THE PANEL TOOLBAR

The *Panel* toolbar provides users with commands to...



1 –



THE ROOF PLANES TOOLBAR

The *Roof Planes* toolbar provides users with commands to...



Create Planes –



Extend Plane –



Auto Extend Single –



Auto Extend All –



Clip Off Corner –



Five Sided Bay Window –



Three Sided Bay Window –



The *Roof Planes* toolbar.

THE SELECTION & SNAP TOOLBAR

The *Selection & Snap* toolbar provides users with commands to...



Set Snap Grid –



Set Visual Grid –



Deselect All –



Select Item(s) –



Select All Visible –



The *Selection & Snap* toolbar.

THE STANDARD TOOLBAR



The *Standard* toolbar.

The *Standard* toolbar provides users with commands to...



Set Levels – The Set Levels command allows you to view and edit information about all the levels in the current model, as well as create new levels. **For additional information, please see the section within this document entitled...**



Set Visibility – The Set Visibility command allows you to pick which entities should be displayed in the drawing area, and which should be hidden. If a member type name has a check next to it in the Set Visibility Window, it will be displayed in the Drawing Area. If the member is not checked, it will not be shown. To check (or un-check) a member type, click in the box to the left of the name. Below is the Set Visibility window. See below for more information on each field.

Member Types – Place a checkmark next to the member types that you want to be visible in the model. Uncheck the member types that you do not want to be visible.

All On (1) – Click the All On button (or press the 1 key on your keyboard) to place a check in every box, and all member types will be displayed.

All Off (2) – Click the All Off button (or press the 2 key on your keyboard) to remove the check from every box, and no member types will be displayed.

OK – Click the OK button to apply the information above and return to the main GS Plan™ window.

Cancel – Click the Cancel button to reset the information in the window (as it was when you entered) and return to the main GS Plan™ window.

Apply – This button is grayed out until you make a change to the checkboxes above. Click this button to apply the changes in this screen immediately and see how the changes affected your model without having to close this window.



Bearings –



Planes –



Extend Plane –



Areas –



Show/Edit Truss Profiles –



Zoom/Pan Window –



Plan View –



Animate –



Solid Model –



Set Properties –



Select Item(s) –

THE STRUCTURE COMMANDS TOOLBAR

The *Structure* toolbar provides users with commands to perform all the functions of the *Structure* program, from generating loads to creating plots. Note: The buttons on the *Structure Commands* toolbar will be unavailable until you save your job.



The *Structure Commands* toolbar.

[Level Drop-Down List] – Click the Levels button on the *Structure Commands* toolbar to select a job level. Selecting a level makes that level both active and visible. If you had other levels active, visible or shadowed, those levels will be turned off.



Set Structure Defaults – Click the *Set Structure Defaults* button to run one or more of the Design and Member Placement steps of designing the structure. These steps will generate loads, analyze the materials and place members.



Vibration Conditions – Click *Vibrations Conditions* on the *Structure Commands* toolbar to open the *General Information* window and view the *Vibration Conditions* tab. This gives you easy access to change your Vibration Condition settings while designing a job, since you will likely change those settings per level. **(THE VIBRATION CONDITIONS TAB IN GENERAL INFO NO LONGER EXISTS.)**



Generate Loads – Generate Loads is the first command that must be run on each level of your job. Click the Generate Loads button to:

1. transfer loads from the level above,
2. generate loads within the current level,
3. verify all loads are supported, and
4. verify all members are supported.

If you are designing a multi-story building, upper levels must be run through Generate Loads before lower levels so that loads can be properly transferred from level to level.

If any problems occur while the steps above are being run, you will receive a message in the Message Log. There are three types of messages: warnings, errors and constraints. If you receive any error or constraint messages, you need to fix those problems in the model and click Generate Loads again in order to continue. If you receive a warning message, that problem in the model can likely be fixed in the field and you can continue designing your job. The table below describes warning, error and constraint messages.



Member Selection – Click Member Selection to choose products for each member in the job. First click a member type (Framing Areas, Joist Areas, Girder, Beams, Headers, Non-Load Bearing Headers or Posts), then select the members. Click Select to choose a product for the selected members. You can also view a calc sheet for any member or open a member in the Beam program for further review.

- **Selected** – Selected shows the numbers of the selected members. In the picture above, all the joist areas are selected so it displays 1-12 in the Selected field. This field can also be used to select members. The numbers for each member can be found next to the member type they represent in the Display Area below. Enter or select the numbers of the members for which you want to select products. There are several ways to select members:
 - Type the number(s) of the member(s) into the Selected field
 - Click the plus (+) and minus (-) buttons to add or remove all members

- Click and drag to draw a box around the members in the Display Area
 - Left click on a single member to add it to the Selected list
 - Right click on a single member to remove it from the list
- *Selected Product* – This field displays the material and plies, lams, or spacing used for the selected members. If the selected members have different solutions, the word "various" will be displayed. If you see "***No Solution***", you will need to choose a product that works for this member.
 - *Select* – Click Select to choose products for the selected members.
 - *Message Box* – This area gives you hints on the next step you need to perform. In the picture above, it says "Left adds/right removes" meaning that you click a member to add it to the Selected list, right-click a member to remove it from the Selected list.
 - *Price* – This field displays the total price for the level you are currently designing. This total is based on the prices entered on the Pricing tab on the General Information window. If you choose a material during the Member Selection process that is not listed on the Pricing tab, you will be prompted to enter a price for that material and it will be entered into the table on the Pricing tab as well.
 - *Display Area* – This area shows you the current level. When you click a member type such as Girders, those members will be highlighted in the Display Area. Click on a member to select it, or click and drag to draw a box around multiple members at once. Hover over a member to see its product solution.
 - *Framing Areas* – Click Framing Areas to display all the Framing Areas in the Display Area. For every area in your job, a Framing Area is displayed here.
 - *Joist Areas* – Click Joist Areas to display all joist areas in the Display Area. A Joist Area is made up of a section of a Framing Area where there is a common design (a part of the Framing Area where the members have the same length and loading).
 - *Girders* – Click Girders to display all girder members in the Display Area. Girders are flush support members (defined as girder, not truss girder or girder by others).
 - *Beams* – Click Beams to display all beam members in the Display Area. Beams are dropped support members (defined as a Beam, not Beam by Others).
 - *Headers* – Click Headers to display all load bearing headers in the Display Area. Headers are members framed above door or window openings (defined as headers in a load bearing wall).
 - *Non-Load Bearing Headers* – Click Non-Load Bearing Headers to display all non-load bearing headers in the Display Area. Non-load bearing headers are members framed above non-bearing door openings (defined as headers in non-load bearing walls).
 - *Posts* – Click Posts to display all posts in the Display Area.
 - *Calc Sheet* – Click Calc Sheet to display the design information for the selected member(s). The calc sheet for the first of the selected members will be displayed. If there are additional calc sheets to view, click Next to move to the next sheet. Click Print at the top of the Calc Sheet window to print these pages.
 - *Open KYB* – Click Open KYB to open the currently selected member in the Beam program.
 - *Exit* – When you have finished selecting solutions for each member, click Exit.



Member Placement – Click Member Placement to place joist accessories as well as generate a list of all the designed materials in the current level along with component, expander, and hanger information. This information will be used to create a material file that can be displayed on plots and imported into KeyMan or TMS.

Joist accessories include rim, blocking, web stiffeners, and cantilever reinforcements. All the accessories required by the design are placed during Member Placement and appear directly in your job. Their placement is affected by the presets on the Joist Accessories, Blocking & Rim, and the Web Stiffeners & Cantilevers tabs of the General Info window.

To adjust any members placed during Member Placement, use the commands on the Modify menu, such as Trim/Extend Lines and Shift/Copy Selected. To add additional framing materials, click Entities and then click Additional Framing.

If any member on the current level does not have a solution, you will receive a message that one or more members does not have a solution. Click Member Selection to check for the failing member and find a material that will work.

NOTE: Generate Loads and Member Selection need to be run on the current level before you can run Member Placement. If these steps have not been done, they will be run for you when you click Member Placement.

NOTE: If you place any additional framing materials (such as rim and blocking) prior to running Member Placement, those members will not be deleted before the Member Placement process adds new ones. You will receive a message in the Message Log that says there are overlapping members. Click on the message to see the location of the warnings in the model and make corrections as needed.



Hanger Selection – Click the Hanger Selection button on the Structure Commands toolbar to view and edit hanger information for the current level.

- *Hanger List* – Click on a plus sign (+) to view more information about that hanger. Click the minus sign (-) to hide that hanger's information
- *Select Hanger* – Click Select Hanger to choose a hanger for the selected connection. The list will be made up of hangers from your KeyMan/TMS Inventory followed by Simpson Strong-Tie hangers that work for the condition. Click a hanger in the list and then click OK. You will receive a message if no solution can be found for a selected hanger, and that hanger will be marked as a Special Order hanger on the output.
- *Select from Inventory* – Click Select from Inventory to choose a hanger from your Keyman/TMS Inventory. The list will only be made up of hangers from your KeyMan/TMS Inventory. Click a hanger in the list and then click OK.
- *Edit Hanger* – Click Edit Hanger to edit the depth and species of the carrying member.
- *OK* – Click OK to exit the Select Hangers window and save your changes.
- *Cancel* – Click Cancel to exit the Select Hangers window without saving any changes.



Additional Framing – This button allows you to easily add additional framing to your job such as rim and blocking. Click the Additional Framing button and then draw a line where you want the members to be placed. Choose the type of members to place (Rim Joist, Blocking, Bracing, etc), set the properties and product and then click OK. The additional framing will be added to the job. To

enter a crush block, click on the location you want it placed and then press the down arrow and 0. You can then use the shift/copy selected command to copy this crush block to multiple locations.

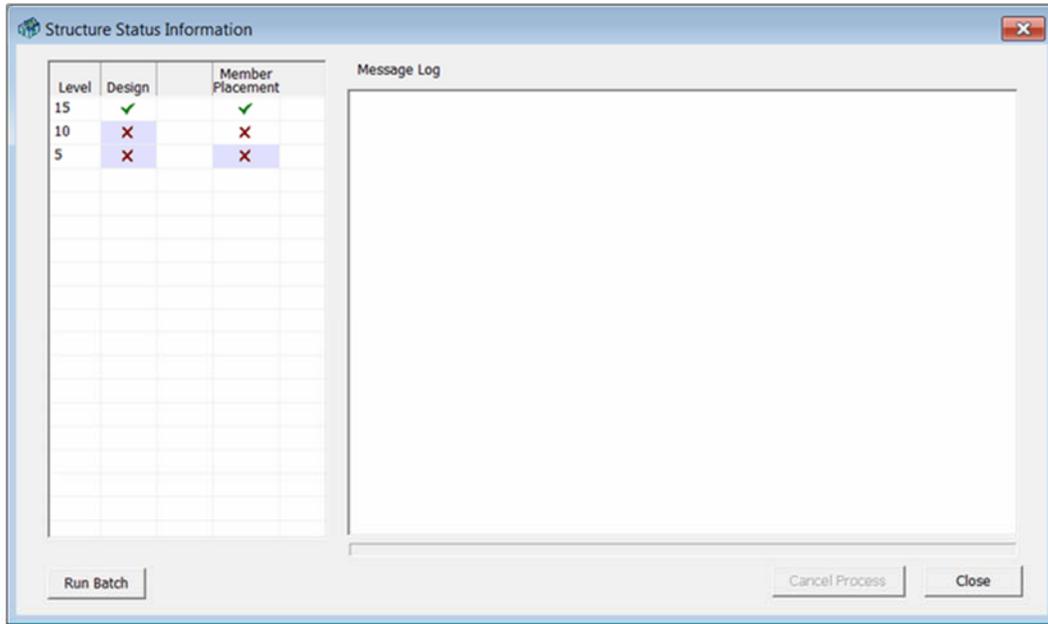


Structure Outputs – Click the Structure Outputs button on the Structure Commands toolbar to generate several different types of output including plots, calc sheets, cut lists and KeyBeam files. The table below contains descriptions of each option.

- *Framer's Plot* – Click Structure Outputs and then click Framers Plot to open KeyPlot and generate a builder's layout. The Framer's Plot contains a layout of the level, a hanger list, and a consolidated materials list. The labels for the members are consolidated as well, with like members (members of the same material and length) bearing the same label.
- *Loads Plot* – Click Structure Outputs and then click Loads Plot to open KeyPlot and generate an engineering layout. The Loads Plot contains a layout of the level and a hanger list. The labels for the members are independent to each member, and correspond to the label on the Calc Sheet.
- *Print Calc Sheets* – Click Structure Outputs and then click Print Calc Sheets to print a calculation sheet for every design member (except By Others or Truss members) on the current level.
- *Print Hanger Calcs* – Click Structure Outputs and then click Print Hanger Calcs to print a list of the calculations used to pick the hangers used on the current level.
- *Print Hanger Table* – Click Structure Outputs and then click Print Hanger Table to print the Hanger Table (the same one that is generated for plots).
- *Print Cut List* – Click Structure Outputs and then click Print Cut List to print the Cut List (the same one that is generated for plots).
- *Print Member Load Report* – Click Structure Outputs and then click Print Member Load Report to print a loads report for any "by others" members (Beam by Others, Framing Member by Others or Girder by Others) in the current level. Since no products are actually sized for these members, this is the only way to see any load information for them.
- *Create KeyBeam Files* – Click Structure Outputs and then click Create KeyBeam Files to generate a .kyb file for each design member (except By Others and Truss members) on the current level. You can open .kyb files in KeyBeam.
- *Structural Materials List (MMS File)* – Click Structure Outputs and then click Structural Materials List (MMS File) to generate a list of structural materials that exist in the model.

NOTE: Member Selection must be run successfully in order to generate a Framer's Plot or Loads Plot, Print Calc Sheets, or Print Hanger Calcs.

The *Structure Status Information* window shows you the status of the *Design* and *Member Placement* steps for each level in your job. In the example below, the Design and Member Placement steps have already been run on Level 15, as shown by the green check marks. The Design steps are set to run on Levels 5 and 10 and Member Placement is set to run on Level 5, as shown by the gray background around the red Xs.



- *Level* – This field lists each level included in the job. For each level, you can see whether or not *Design* and *Member Placement* have been run successfully. You see a red X in those columns if those steps have not been run successfully; a green check mark if they have been run successfully.

Click the boxes for the steps you wish to run on each level. The box background will be gray when that step is set to run. In the image above, *Design* and *Member Placement* will be run for Level 5, while only *Design* is set to run for Level 10. Since *Design* must be run on upper levels before lower levels, if you click the *Design* box on Level 5, any levels above that one will show gray boxes for *Design* as well.

- *Message Log* – Displays a log of the progress analyzing the job. For successful batches, you see a "RESULT: SUCCESS, ALL_DONE" message. For unsuccessful batches, you see a "RESULT: FAILURE, ALL_DONE" message. The error is then highlighted in the message log so you can adjust the model or your settings.
- *Run Batch* – A batch is a set of steps the program will run. Click Run Batch to run the steps selected in the Level List above. While the batch runs, you see a progress bar that indicates the progress of the batch.
- *Cancel Process* – Click this button during a batch to cancel the process. The current process stops and you see "RESULT: Canceled" in the Message Log.
- *Close* – Click to close the window and return to the Main Build Window.

THE TOGGLE MACROS TOOLBAR

The *Toggle Macros* toolbar provides users with commands to...



Toggle Load/Bearing – Toggles between load bearing and non-



The *Toggle Macros* toolbar.

load bearing...The Load Bearing Toggle allows you to switch walls on the active level from bearing to non-bearing and back with a single mouse click. Walls will be displayed as follows: Load Bearing walls will be displayed in green. Non-load Bearing walls will be displayed in purple.



Toggle Roof/Floor – Toggles between roof and floor...The Roof/Floor Toggle allows you to view and change walls on the active level from roof bearing to floor bearing and back by clicking on them. Walls will be displayed as follows: Roof supporting members will be displayed in green. Floor supporting members will be displayed in purple.



Toggle Interior/Exterior – Toggles between interior and exterior...The Interior/Exterior Toggle allows you to switch walls on the active level from interior to exterior and back with a single mouse click. Walls will be displayed as follows: Interior members will be displayed in green. Exterior members will be displayed in purple.



Toggle Wall/Header Widths – The Wall/Header Widths Toggle allows you to view and change the widths of all wall and header members on the current level. Each of the different wall or header width will be displayed in a different color. A key to these colors will be displayed on the right side of the main window.

To change the width of a wall or header, click the width to which you want to change from the sidebar and then click on the wall or header that you want to change. To change a member to a width that is not listed on the sidebar, click the Other entry, then enter the new wall width and click the OK button. The new width will be listed as an option on the sidebar.

To change all members to a specific width, click the Global Change button. You will then be prompted to select which types of walls should be changed: Interior, Exterior or both. Click the OK button to change all of the members specified to the currently selected width.



Toggle Overhang Lengths–



Toggle Framed Foundation/Non-Framed – Toggles between Framed, Foundation, and Non-Framed...The Framed/Non Framed Toggle allows you to switch walls on the active level from framed to non-framed and back with a single mouse click. Walls will be displayed as follows: Framed walls will be displayed in green. Non Framed walls will be displayed in purple.



Toggle Top Track Elevations – The Top Track Elevation Toggle allows you to change the heights of walls on the active level by change their top plate elevations. Each existing wall height will be displayed in a different color. To add a new wall height, click the "Other" entry in the list on the right and enter the new top of plate elevation. This number will be added to the list. Select the height you wish to assign from the list on the right and then click on the wall to apply it to. The wall height will change, but the bottom plate will be held in its previous location.



Toggle Wall/Header Heights – The Wall / Header Height toggle allows you to toggle Wall and Header Heights for the active level. When in Toggle Wall/Header Height mode, walls will be visible and colored according to the wall height legend on the right side of the screen. To change wall heights, left-click on the appropriate wall height in the legend, then left-click on each wall to change. To add new wall heights, click on the Other button, enter the new wall height in the Set New Wall Height dialog box, and click OK. To Globally Change wall heights, changing all wall heights of the active level, click on the appropriate wall height, then click the Global Change button, and finally check the appropriate boxes in the Toggle Wall/Header Values dialog box and click OK.



Toggle Bearing Properties – The Toggle Bearing Properties option allows you to apply the properties of a selected bearing to other existing bearings. When in Toggle Bearing Properties mode, the visibility will change so only bearings are visible. You will be prompted to 'Select Template Bearing,' where you will click on an existing bearing to be the template bearing. The Set Bearings Properties dialog box will come up for you to verify or change the bearing template properties. These are the properties that will be applied to other bearings. Once the template bearing properties are set, click OK to begin applying these properties to other bearings or Cancel to not apply the template bearings properties. You will be prompted to Select Target Bearing. The properties of each bearing you click on will change to match the templates properties. You can continue to toggle bearing properties until you right-click with the mouse or press the Escape key.



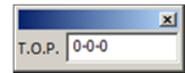
Toggle Bears on Earth – Toggles between Bears on Earth and Bears Not on Earth...

THE TOP TOOLBAR

The *Top* toolbar provides users with commands to...



1 –



THE WALL PANEL TOOLBAR

The *Wall Panel* toolbar provides users with commands to...



The *Wall Panel* toolbar.



Display and Modify Corner Laps –



Relap All Walls to Default –



Frame One Wall –



Frame All Walls –



Unframe One Wall –



Unframe All Walls –



Panel Application –

GENERAL INFORMATION TABS & SETTINGS

Job Info Tab

General Info

Deflection Criteria | Joist Accessories | Blocking & Rim | Material Lists | Cut Lengths | Colors & Line Types | Plot Labels

Job Info | Job Settings | Load Info | Framing Members | Other Materials | Panel Settings | Design Settings

Job Name:

Customer ID:

Customer Address:

Job Comments:

Salesman ID:

Job ID:

Site Address:

Select checkboxes to include fields on calc sheets

OK Cancel

Job Name – Type the name of this job (up to 12 characters).

Salesman ID – Type the salesman's ID number (up to three characters).

Customer ID – Type the customer's ID number (up to six characters).

Job ID – Type the Job ID (up to 32 characters).

Customer Address – Type the customer's name and address.

Site Address – Type the address of the site where this job will be built.

Job Comments – Type any additional information that you want associated with this job.

OK – Click OK to save the changes.

Cancel – Click Cancel to return to the previous window.

Job Settings Tab

Roof Pitch – Enter the default roof pitch that you want to use for this job. Click the up and down arrows to change the pitch format between:

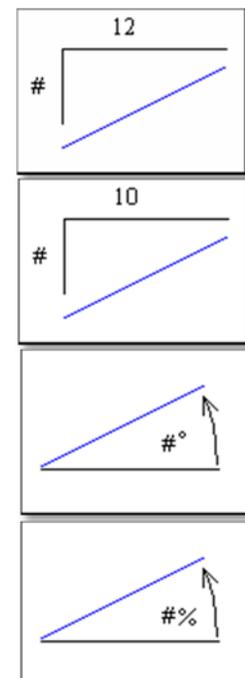
- Slope over 12
- Slope over 10
- Degrees
- Percentage

If you select slope over twelve ($\#/12$), the first number represents the rise for every twelve units of run (as in a 6/12 roof slope).

If you select slope over ten ($\#/10$), the first number represents the rise for every ten units of run (as in a 4/10 roof slope).

If you select degrees ($\#^\circ$), the number entered represents the slope in degrees (as in a 30° slope). This is based on a horizontal line being 0° , so a vertical line would be input as 90° .

If you select percentage ($\#\%$), the number entered represents the percentage of slope. A flat roof is 0% and a 45-degree angle is 100%.



Wall Width – Type the default width of the walls in this model.

Wall Height – Type the default height of walls in this model.

Designer – Type the name of the designer for this job. This information appears on all output.

Company Address – Type your company’s name, address, and phone number. This information appears on all output.

Metric Units – Select this box to use Metric units of measure instead of the default Imperial units.

METRIC

The default value in metric units is millimeters. Possible formats include:

- *m* – meters
- *dm* – decimeters
- *cm* – centimeters
- *mm* – millimeters

If a value is entered with no measurement (e.g., entered as 1000 instead of 1000m or 1000cm) it is assumed to be in mm. To force a value to use a specific unit of measure, simply include the letters above (e.g. 3m means 3 meters)

IMPERIAL

The default format in Imperial units is feet – inches – sixteenths of an inch. Possible formats include:

- *10* = ten feet
- *3-8* = three inches, eight sixteenths (or half) of an inch (a common wall width default value)
- *10-5-8* = ten feet, five inches, eight sixteenths of an inch
- *0-5-8* = five inches, eight sixteenths of an inch

Although the default is feet – inches – sixteenths of an inch, you may enter that value in multiple ways. You could enter two feet as 2+0+0, 2, 24in, 24", 2ft, 2', or 2.00 and all would result in 2-0-0. Certain fields will force you to input only inches and sixteenths of an inch.

Hi Pre Units – Select this box to force the application to use more precise numbers. Numbers display with four decimal places (as in a 3.5000 wall width), allowing measurements to be input to 1/10,000th of an inch for Imperial or to 1/100th of a millimeter for Metric.

FFIISS Units – Select this box to type FFIISS (Foot-Foot-Inch-Inch-Sixteenths-Sixteenths) numbers without additional symbols. This lets you input dimensions more quickly because you are only typing numbers. For a 10-3-8 span, type 100308 instead of 10-3-8 or 10+3+8.

Glued Flooring – Select this box to indicate that you want the structural analysis of the floor design to include the sheathing as glued and nailed to the joists. Clear this box to analyze the floor as nailed only, which is a more conservative design. This setting is reported on joist calc sheets.

Bearing as Floor – Select this box to set your bearings to support floors by default. When you input a bearing such as a wall or girder, the Floor check box is selected.

Unique Rim Labels – This field affects only machinery files output using the commands on the Materials menu. Select this box to ensure that each piece of rim gets a unique label. If this box is cleared, rim that is alike in length and product gets the same label in the machinery file. Since these labels are used by the

machinery to mark the pieces and those labels are used to place the pieces, some users prefer that the pieces are labeled consecutively instead of by like length/product.

Unique Girder Labels – This field affects only machinery files output using the commands on the Materials menu. Select this box to ensure that each girder gets a unique label. If this box is cleared, girders that are alike in length and product gets the same label in the machinery file. Since these labels are used by the machinery to mark the pieces and those labels are used to place the pieces, some users prefer that the pieces are labeled consecutively instead of by like length/product.

EXTEND PLANES

Select one of the following as your default method for extending roof planes:

- **By intersection lines** – The default and the most commonly used option for extending planes. When you select to extend a plane, all possible intersection lines are displayed and you must click on each line in a clockwise order.
- **By adjacent planes** – Lets you extend planes by clicking on an adjacent wall or adjacent plane, instead of clicking on potential intersection lines.

PLOT PROGRAM

Select the program you want to use for plots.

- **Plot** – generates the plot in KeyPlot.
- **Export Plot DXF** – generates a DXF from KeyPlot that can be opened in any other CAD program. This option automates the steps of opening KeyPlot and generating DXFs for you.

COMMAND BUTTONS

Save as Defaults – Click Save as Defaults to store all information on this tab as the default information. Each time you start a new job, these settings are used.

Restore Defaults – Click Restore Defaults to replace all settings on this tab with the default settings that you saved previously using Save as Defaults.

OK – Click *OK* to save the changes.

Cancel – Click *Cancel* to return to the previous window.

Load Info Tab

General Info

Deflection Criteria | Joist Accessories | Blocking & Rim | Material Lists | Cut Lengths | Colors & Line Types | Plot Labels
Job Info | Job Settings | **Load Info** | Framing Members | Other Materials | Panel Settings | Design Settings

Floor Loads

Top Chord / Joist Live 40.000 psf
Top Chord / Joist Dead 15.000 psf
Bottom Chord Live 0.000 psf
Bottom Chord Dead 0.000 psf

Wall Loads

Exterior Wall Weight 10.000 psf
Interior Wall Weight 7.000 psf
Foundation Wall Weight 10.000 pcf
Non-Framed Wall Weight 10.000 pcf

Combine Trusses Load Warning

Roof Loads

Top Chord / Rafter Live 20.000 psf
Load Case Roof
Top Chord / Rafter Dead 10.000 psf
Bottom Chord Live 0.000 psf
Bottom Chord Dead 10.000 psf

Distribute Overframing Loads
 Adjust Dead Load For Pitch

Tributary Area

Max Girder Tributary Width 24-0

Save as Defaults | Restore Defaults

OK | Cancel

FLOOR LOADS

Top Chord/Joist Live – Type the gravity loads for top chord/joist live to be applied to floor members in psf (pounds per square foot) for Imperial users and kN/m² (kilonewtons per square meter) for Metric users.

Top Chord/Joist Dead – Type the gravity loads for top chord/joist dead to be applied to floor members in psf (pounds per square foot) for Imperial users and kN/m² (kilonewtons per square meter) for Metric users.

Bottom Chord Live – Type the gravity loads for bottom chord live to be applied to floor members in psf (pounds per square foot) for Imperial users and kN/m² (kilonewtons per square meter) for Metric users.

Bottom Chord Dead – Type the gravity loads for bottom chord dead to be applied to floor members in psf (pounds per square foot) for Imperial users and kN/m² (kilonewtons per square meter) for Metric users.

WALL LOADS

Exterior Wall Weight – Type the exterior wall weights in psf (pounds per square foot) for Imperial users and kN/m² (kilonewtons per square meter) for Metric users.

Interior Wall Weight – Type the interior wall weights in psf (pounds per square foot) for Imperial users and kN/m² (kilonewtons per square meter) for Metric users.

Foundation Wall Weight – ???

Non-Framed Wall Weight – Type the non-frame wall weight in pcf (pounds per cubic foot) for Imperial users and kN/cu m (kilonewtons per cubic meter) for Metric users.

[OTHER]

Combine Trusses Load Warning – Select this box to see a warning in *Edit Trusses* when you attempt to combine trusses that have different loads.

ROOF LOADS

Top Chord/Rafter Live – Type the gravity loads for top chord/rafter live to be applied to roof members in psf (pounds per square foot) for Imperial users and kN/m² (kilonewtons per square meter) for Metric users.

Load Case – Select the load case. This affects the percentage of increase in allowable stresses, based on the length of time that a load is present on a member or series of members. Select from the following four load cases:

- Live 100%: 10 years - occupancy live load
- Snow 115%: 2 months - snow load
- Roof 125%: 7 Days - construction load
- Wind 133%: 1 day - wind or earthquake loads

Top Chord/Rafter Dead – Type the gravity loads for top chord/rafter dead to be applied to roof members in psf (pounds per square foot) for Imperial users and kN/m² (kilonewtons per square meter) for Metric users.

Bottom Chord Live – Type the gravity loads for bottom chord live to be applied to roof members in psf (pounds per square foot) for Imperial users and kN/m² (kilonewtons per square meter) for Metric users.

Bottom Chord Dead – Type the gravity loads for bottom chord dead to be applied to roof members in psf (pounds per square foot) for Imperial users and kN/m² (kilonewtons per square meter) for Metric users.

Distribute Overframing Loads – When this checkbox is activated, GS Plan distributes the loads created by overframing trusses to the framing below. Typically this causes point loads on the trusses below. When not activated, the application ignores the additional load created by overframing trusses, since the load is assumed to be covered by the dead load applied to the trusses below.

Adjust Dead Load for Pitch – Typically, dead loads are applied in a projected method which is traditional and less conservative. Select this box if you want to apply dead loads along the rake, taking into account the pitch of the roof. This option applies more dead load and is therefore more conservative.

TRIBUTARY AREA

Max Girder Tributary Width –

COMMAND BUTTONS

Save as Defaults – Click Save as Defaults to store all information on this tab as the default information. Each time you start a new job, these settings are used.

Restore Defaults – Click Restore Defaults to replace all settings on this tab with the default settings that you saved previously using Save as Defaults.

OK – Click OK to save the changes.

Cancel – Click Cancel to return to the previous window.

Framing Members Tab

	Full Width	Flange Width
Roof Truss	1-8	0-0
Roof Truss Girder	1-8	0-0
Floor Truss	3-8	0-0
Floor Truss Girder	3-8	0-0

Blocking Panels

Place along reference line
 Place perpendicular to joist

Save as Defaults Restore Defaults Designed Products

OK Cancel

Roof Truss Full Width – Type the width of the standard (non-girder) truss material (in inches and sixteenths of an inch) to be used by default in the roof.

Roof Truss Flange Width –

Roof Truss Girder Full Width – Type the width of the girder truss material (in inches and sixteenths of an inch) to be used by default in the roof.

Roof Truss Girder Flange Width –

Floor Truss Full Width – Type the width of the standard (non-girder) truss material (in inches and sixteenths of an inch) to be used by default in the floor.

Floor Truss Flange Width –

Floor Truss Girder Full Width – Type the width of the girder truss material (in inches and sixteenths of an inch) to be used by default in the floor.

Floor Truss Girder Flange Width –

BLOCKING PANELS

These two radio buttons are used to define how you want blocking panels placed when you draw a line diagonally through a section of joists.

Place along reference line – When this radio button is selected, blocking panels are placed along the reference line (so they line up with the diagonal line you draw).

Place perpendicular to joist – When this radio button is selected, blocking panels are placed perpendicular to joists.

COMMAND BUTTONS

Save as Defaults – Click Save as Defaults to store all information on this tab as the default information. Each time you start a new job, these settings are used.

Restore Defaults – Click Restore Defaults to restore (replace) all settings on this tab with the default settings that you saved previously using the Save as Defaults button.

Designed Products – Click the *Designed Products* button to access the *Product List* window and define the materials you plan to use for the categories listed below. The materials you choose in these categories will be used in the analysis during member selection.

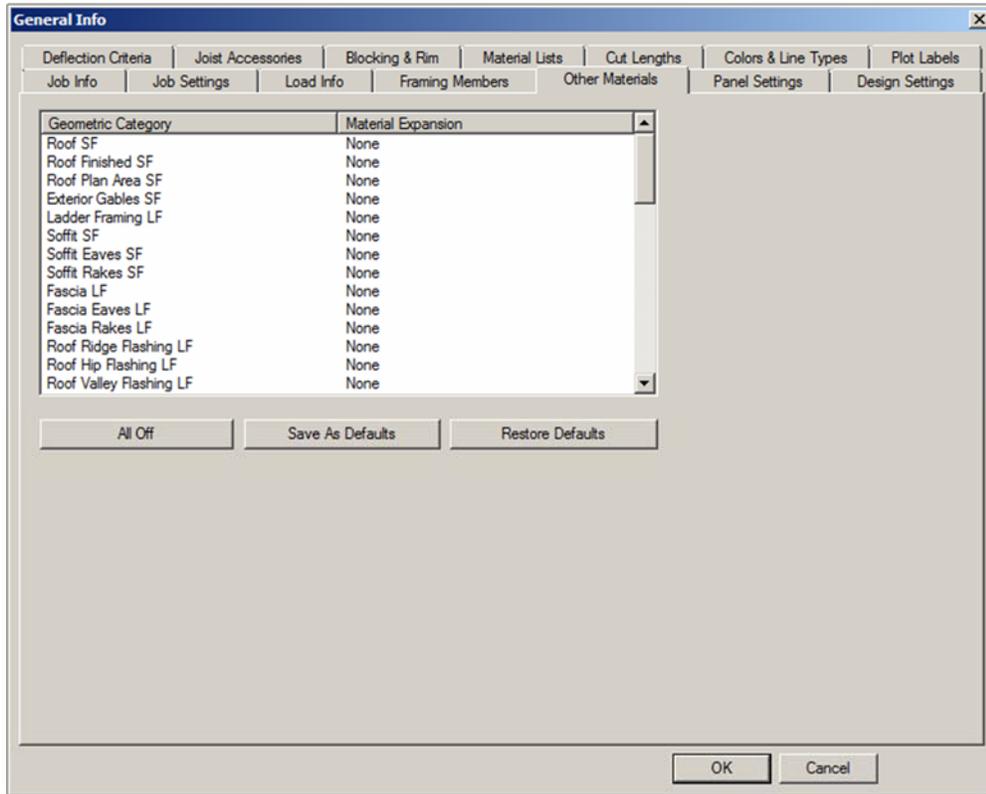
- Floor Beam
- Floor Girder
- Floor Joist
- Roof Beam
- Roof Girder
- Roof Joist

For more information, see *Product List* window and *Field Descriptions*.

OK – Click OK to save the changes.

Cancel – Click Cancel to return to the previous window.

Other Materials Tab



This window allows you to set up expanders to be applied to different geometric categories. Expanders are set up in the management program and are groups of materials that can be applied to a specific geometric category. For example, you might set up a Roof expander that contains roof sheathing, roof felt, and roof shingles. You assign a coverage to each of those materials (for example a 4 x 8 piece of roof sheathing covers 32 square feet). Then, when that roof expander is applied to a job that has 1000 square feet of roof area, the management program does the math to figure out how much of each of those roof materials need to be added to the job. Using the 4 x 8 sheathing example, 32 sheets of roof sheathing material would be added to your materials list since 1000 sq. ft. divided by 32 square feet of coverage equals 31.25 and that number is rounded up to 32.

WARNING! Expanders must be defined in the management program before they can be assigned on this window. If you do not set up expanders in the management program, none will be available to select on this window.

Geometric Category – This list displays all of the geometric categories available. Click on the name of a category to display all of the available expanders for that category. If the expander (group of materials that will be applied based on the geometric data) you want to select is not available, you must exit the application and add the expander in the management program.

Material Expansion – The Material Expansion list will display the expander currently assigned to each of the geometric categories. Click on the name of an expander to list all of the available expanders for that category. More information, including a list of the different geometric categories including examples of how they are calculated, a picture of where the measurement comes from, and an example of an expander product that might be applied to that geometric data will be coming soon!

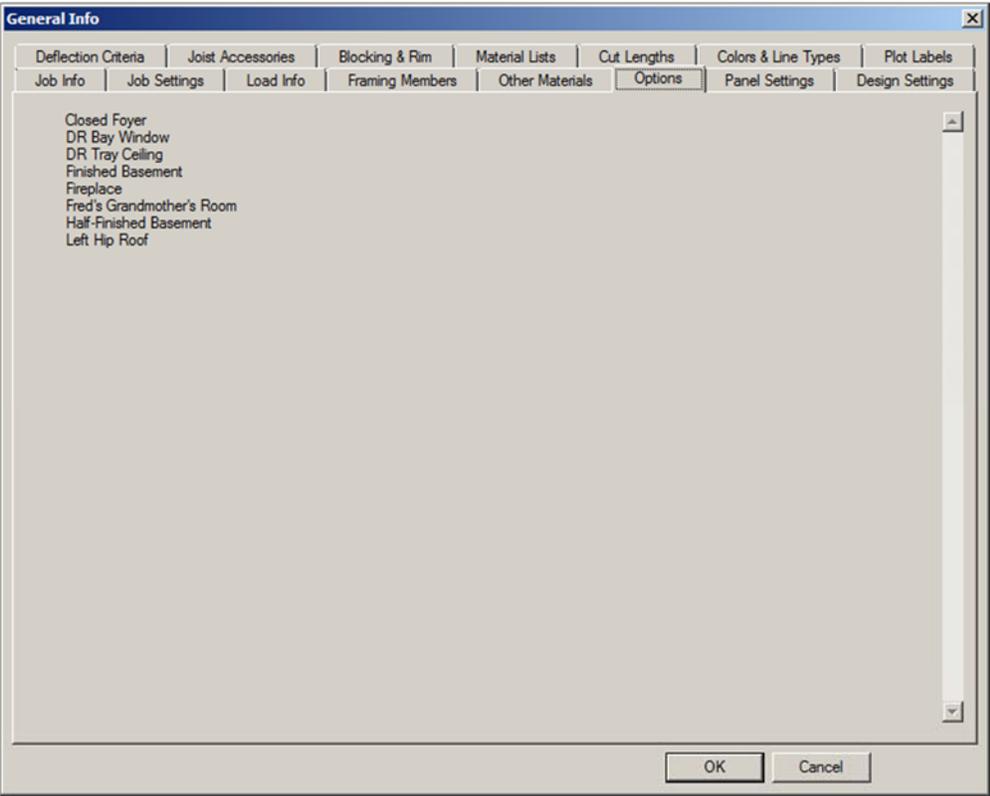
COMMAND BUTTONS

All Off – Click the *All Off* button to change the material expansion for each geometric category to none.

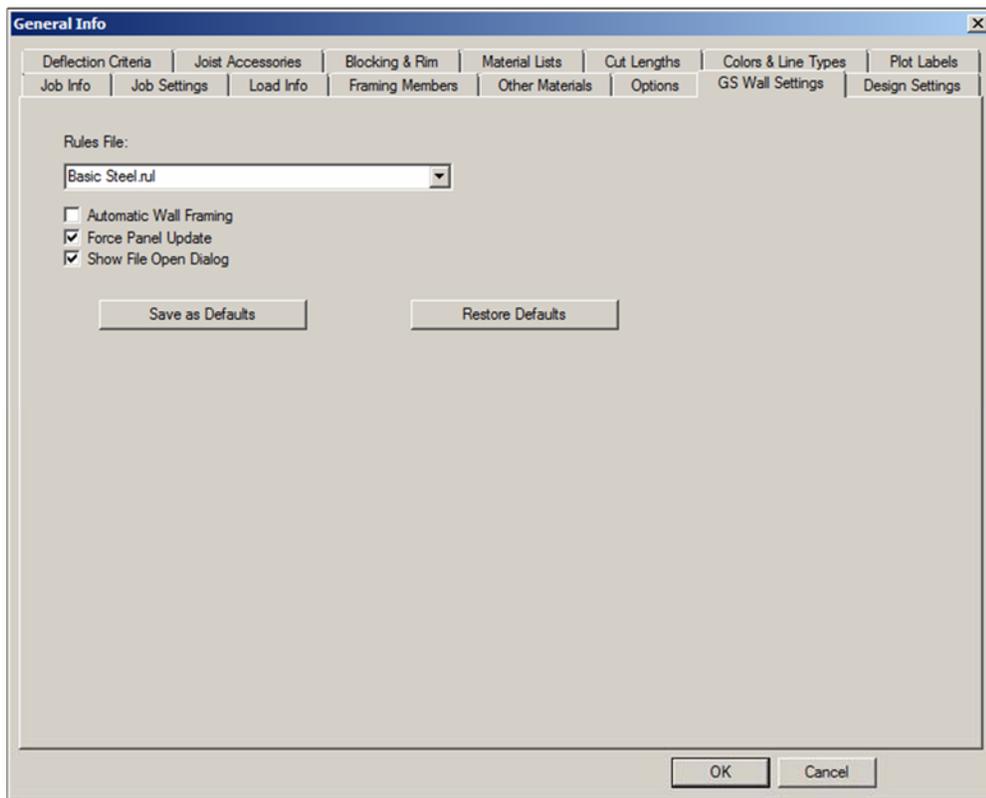
Save As Defaults – Click *Save As Defaults* to store all information on this tab as the default information. Each time you start a new job, these settings will be used.

Restore Defaults – Click *Restore Defaults* to replace all settings on this tab with the default settings that you saved previously using *Save As Defaults*

Options Tab



GS Wall Settings Tab



Rules File – This field provides a drop-down list of...

Automatic Wall Framing – When this checkbox is activated, GS Plan...

Force Panel Update – When this checkbox is activated, GS Plan...

Show File Open Dialog – When this checkbox is activated, GS Plan...

COMMAND BUTTONS

Save as Defaults – Clicking on the *Save as Defaults* command button saves the current settings entered into the *GS Wall Settings* tab as the user's default settings.

Restore Defaults – Clicking on the *Restore Defaults* button...

OK – Clicking on the *OK* button saves the user's changes and closes the *General Info* dialog box.

Cancel – Clicking on the *Cancel* button discards the user's changes and closes the *General Info* dialog box.

Design Settings Tab

General Info

Deflection Criteria | Joist Accessories | Blocking & Rim | Material Lists | Cut Lengths | Colors & Line Types | Plot Labels

Job Info | Job Settings | Load Info | Framing Members | Other Materials | Panel Settings | Design Settings

Maximum Member Length: 60 ft Deflection Criteria: Increased

Maximum Allowable Spacing: 24 (in) Building Code: IBC/IRC

(Standard Loads on Bearings)
(Non-engineered default spacing)

Allowable Spacings (in)

Roof	Floor
<input type="checkbox"/> 8.00	<input type="checkbox"/> 8.00
<input checked="" type="checkbox"/> 12.00	<input checked="" type="checkbox"/> 12.00
<input checked="" type="checkbox"/> 16.00	<input checked="" type="checkbox"/> 16.00
<input checked="" type="checkbox"/> 19.20	<input checked="" type="checkbox"/> 19.20
<input checked="" type="checkbox"/> 24.00	<input checked="" type="checkbox"/> 24.00
<input type="checkbox"/> 32.00	<input type="checkbox"/> 32.00

Bracing: Both Top and Bottom

Material List: FT-IN-16

Dimensional Format: FT-IN-16

Include non-engineered members in calc sheet output

Ignore Warning Messages

Overhang Load Transfer

Save System Defaults

OK Cancel

Maximum Member Length – The maximum length allowable for joist members. If a joist member is longer than the length entered in this field, you will receive a Constraint message in the Message Log during Member Placement.

Maximum Allowable Spacing – The maximum allowable spacing for joist members.

ALLOWABLE SPACINGS

There are two columns that contain allowable spacings: one for *Roof* and one for *Floor*. The program will only attempt to design members at spacings that are selected in this area. Additionally, if a selected spacing is larger than the *Maximum Allowable Spacing* defined above, it will be ignored.

ROOF

Add –

Remove –

FLOOR

Add –

Remove –

Deflection Criteria – Select the deflection criteria you wish to use: Minimum, Increased, or Custom. The deflection criteria for each of these three types are defined on the Deflection Criteria tab.

Building Code – Select the building code you are using to design this structure. This will be displayed on all output. There are currently nine options to choose from:

- NBC – National Building Code
- UBC – Uniform Building Code
- SBC – Standard Building Code
- NBCC/OBC – National Building Code of Canada/Ontario Building Code
- NBCC-2005 – National Building Code of Canada 2005
- NBCC-2010 – National Building Code of Canada 2010
- OBC-2006 – Ontario Building Code 2006
- Other – any other Building Code
- Non Engineered – Loads not applied, no engineering code used
- IBC/IRC - International Building Code / International Residential Code

NOTE: NO LOADS ARE APPLIED to framing members when Non-Engineered is the selected Building Code. No member calculation reports will be generated and all plots will have a small symbol in the lower left corner noting that layout has not been engineered.

Bracing –

Material List Dimensional Format – Choose the dimensional output format you wish to see on your output: feet - inches - 16ths (3-4-8), feet - inches - 8ths (3-4-4), or feet - inches - fractional inches (3' 4 ½").

Include non-engineered members in calc sheet output – There is an option on *Print Calc Sheets* on the *Structure Commands* toolbar which prints all calc sheets for the current level. Check this box to also output a calc sheet for non-bearing headers during that batch, even though they are non-engineered members.

IGNORE WARNING MESSAGES

Overhang Load Transfer – Select this box to ignore warning messages for overhang loads that cannot be transferred to a gable or bearing member during Generate Loads.

COMMAND BUTTONS

Save System Defaults – Click *Save System Defaults* to store all information on this tab as the default information. Each time you start a new job, these settings will be used.

OK –

Cancel –

Deflection Criteria Tab

Category	Type	Minimums	Increased	Custom
Floor Joist	Live	L/360	L/480	L/480
	Total	L/240	L/240	L/240
Floor Bearing	Live	L/360	L/360	L/480
	Total	L/240	L/240	L/360
Flat Roof	Live	L/360	L/360	L/360
	Total	L/240	L/240	L/240
Sloped Roof	Live	L/240	L/240	L/360
	Total	L/180	L/180	L/240

Maximum Live Load Deflection: 9.99 in 1 in

Buttons: Reset System Data, Save as Defaults, OK, Cancel

L/ Values: If necessary, adjust the Live L/ and Total L/ values for each of the three groups of deflection criteria (Minimum, Increased and Custom). Deflection Criteria are usually represented as a ratio – Span over Limit (i.e. L/360). The L stands for length of the span. The larger the limit, the smaller amount of deflection allowed. For example, for a 60 foot member and a limit of 360, the calculation would be $60 \text{ ft}/360 = .1667 \times 12$ (to convert it to inches) = 2 inches of maximum deflection.

FLOOR JOIST

LIVE

Minimums –

Increased –

Custom –

TOTAL

Minimums –

Increased –

Custom –

FLOOR BEARING

LIVE

Minimums –

Increased –

Custom –

TOTAL

Minimums –

Increased –

Custom –

FLAT ROOF

LIVE

Minimums –

Increased –

Custom –

TOTAL

Minimums –

Increased –

Custom –

SLOPED ROOF

LIVE

Minimums –

Increased –

Custom –

TOTAL

Minimums –

Increased –

Custom –

Maximum Live Load Deflection – Maximum live load deflection is the maximum distance a member can move before it fails. Enter the *Maximum Live Load Deflection* value you wish to use, in inches for imperial or millimeters for metric. By default, it is set to a high value of 9.99 inches so that absolute maximum deflection does not control the calculation. You can enter a separate value for the *Maximum Live Load Deflection* to be used when you select the *Custom Deflection Criteria* directly under the *Custom* column.

COMMAND BUTTONS

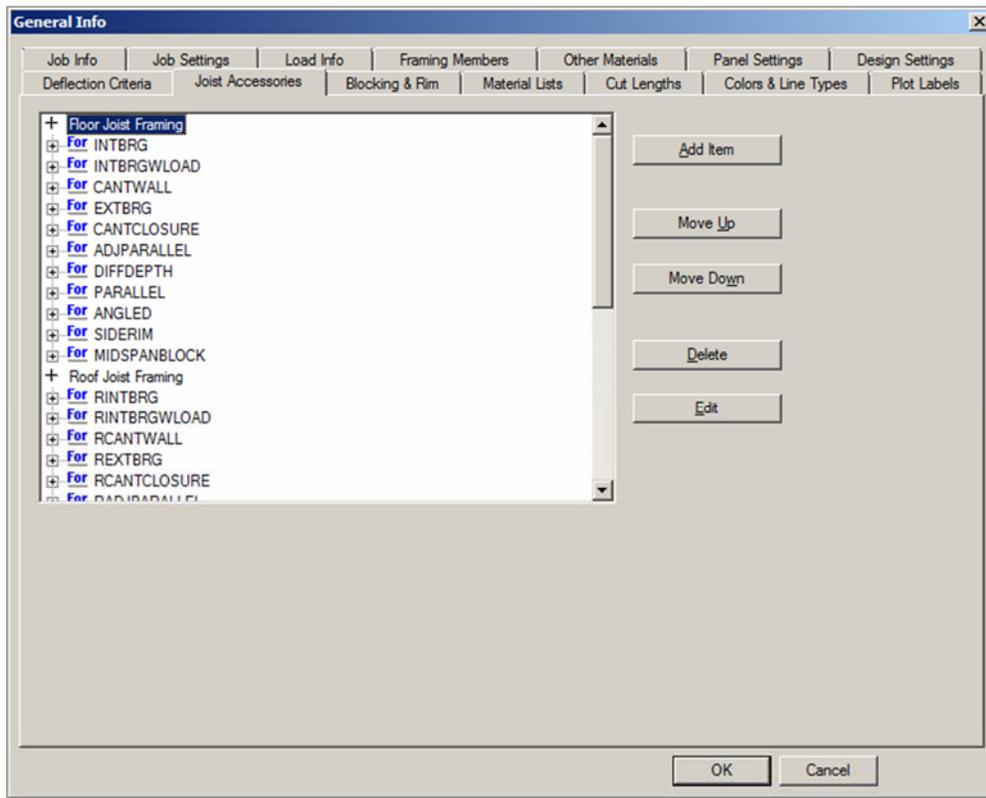
Reset System Data – Click this button to reset the values above to the defaults that were set when you first installed the software.

Save as Defaults – Click Save as Defaults to store all information on this tab as the default information. Each time you start a new job, these settings will be used.

OK –

Cancel –

Joist Accessories Tab



The main categories in the Joist Accessories tree structure include: Floor Joist Framing, Roof Joist Framing, and Ceiling Joist Framing. Below each main category are the possible framing conditions. The table below describes the possible framing conditions that could require joist accessories and the commands found on the Joist Accessories tab.

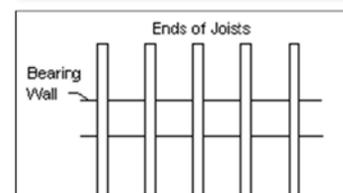
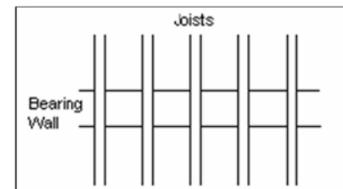
NOTE! The names of the conditions below are the names used in the *Floor Joist Framing* category. When displayed for the *Roof Joist Framing* section, the names will start with the letter “R.” The names in the *Ceiling Joist Framing* category begin with the letter “C.”

FLOOR JOIST FRAMING

INTBRG (Floor: Interior Bearing) – This framing condition occurs when a joist runs across a bearing member and is often referred to as mid-span bearing. This condition is used when there is no load from above being supported at this location.

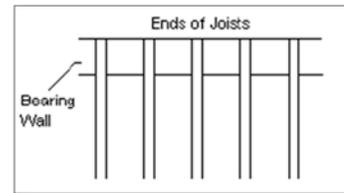
INTBRGWLOAD (Floor: Interior Bearing with Load) – This framing condition is similar to INTBRG, but a load from above is being supported.

CANTWALL (Floor: Cantilever Wall) – This framing condition occurs whenever the ends of the joists run past the end bearing. This category

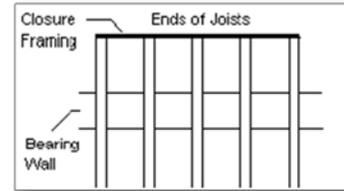


determines how the top of the bearing is framed (NOT the ends of the joists, that is determined by the CANTCLOSURE category).

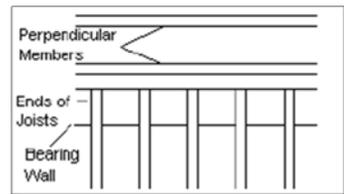
EXTBRG (Floor: Exterior Bearing) – This framing condition occurs when the end of the joists fall on an exterior bearing wall (or beam).



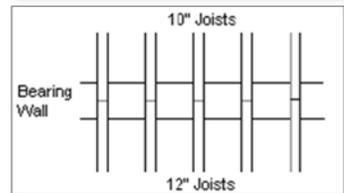
CANTCLOSURE (Floor: Cantilever Closure) – This framing condition occurs whenever the ends of the joists run past the end bearing. This category determines how the ends of the joists are framed (NOT the top of the bearing, that is determined by the CANTWALL category).



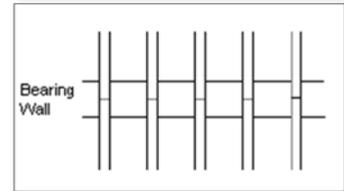
ADJPARALLEL (Floor: Adjacent Parallel) – This framing condition occurs when an area ends at the face of a perpendicular area. This category determines how the ends of the area should be framed (not the side of the perpendicular area).



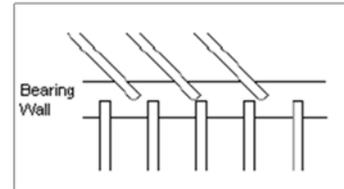
DIFFDEPTH (Floor: Different Depth) – This framing condition occurs when two areas of different depth end on a common bearing.



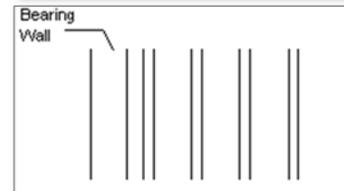
PARALLEL (Floor: Parallel) – This framing condition occurs when two areas of the same depth end on a common bearing.



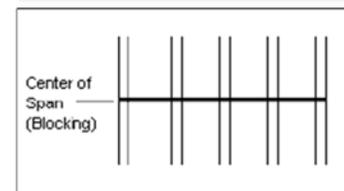
ANGLED (Floor: Angled) – This framing condition occurs when two areas meet on a wall, but the areas are neither parallel nor perpendicular to each other (the members run at some other angle).



SIDERIM (Floor: Side Rim) – This framing condition occurs at the parallel sides of a framing area (as opposed to the ends).



MIDSPANBLOCK (Floor: Mid-Span Blocking) – This framing condition occurs in the middle of the span of joists.



ROOF JOIST FRAMING

RINTBRG (Roof: Interior Bearing) – This framing condition occurs when a joist runs across a bearing member and is often referred to as mid-span bearing. This condition is used when there is no load from above being supported at this location.

RINTBRGWLOAD (Roof: Interior Bearing with Load) – This framing condition is similar to INTBRG, but a load from above is being supported.

RCANTWALL (Roof: Cantilever Wall) – This framing condition occurs whenever the ends of the joists run past the end bearing. This category determines how the top of the bearing is framed (NOT the ends of the joists, that is determined by the CANTCLOSURE category).

REXTBRG (Roof: Exterior Bearing) – This framing condition occurs when the end of the joists fall on an exterior bearing wall (or beam).

RCANTCLOSURE (Roof: Cantilever Closure) – This framing condition occurs whenever the ends of the joists run past the end bearing. This category determines how the ends of the joists are framed (NOT the top of the bearing, that is determined by the CANTWALL category).

RADJPARALLEL (Roof: Adjacent Parallel) – This framing condition occurs when an area ends at the face of a perpendicular area. This category determines how the ends of the area should be framed (not the side of the perpendicular area).

RDIFFDEPTH (Roof: Different Depth) – This framing condition occurs when two areas of different depth end on a common bearing.

RPARALLEL (Roof: Parallel) – This framing condition occurs when two areas of the same depth end on a common bearing.

RANGLED (Roof: Angled) – This framing condition occurs when two areas meet on a wall, but the areas are neither parallel nor perpendicular to each other (the members run at some other angle).

RSIDERIM (Roof: Side Rim) – This framing condition occurs at the parallel sides of a framing area (as opposed to the ends).

RMIDSPANBLOC (Roof: Mid-Span Blocking) – This framing condition occurs in the middle of the span of joists.

CEILING JOIST FRAMING

CINTBRG (Ceiling: Interior Bearing) – This framing condition occurs when a joist runs across a bearing member and is often referred to as mid-span bearing. This condition is used when there is no load from above being supported at this location.

CINTBRGWLOAD (Ceiling: Interior Bearing with Load) – This framing condition is similar to INTBRG, but a load from above is being supported.

CCANTWALL (Ceiling: Cantilever Wall) – This framing condition occurs whenever the ends of the joists run past the end bearing. This category determines how the top of the bearing is framed (NOT the ends of the joists, that is determined by the CANTCLOSURE category).

CEXTBRG (Ceiling: Exterior Bearing) – This framing condition occurs when the end of the joists fall on an exterior bearing wall (or beam).

CCANTCLOSURE (Ceiling: Cantilever Closure) – This framing condition occurs whenever the ends of the joists run past the end bearing. This category determines how the ends of the joists are framed (NOT the top of the bearing, that is determined by the CANTWALL category).

CADJPARALLEL (Ceiling: Adjacent Parallel) – This framing condition occurs when an area ends at the face of a perpendicular area. This category determines how the ends of the area should be framed (not the side of the perpendicular area).

CDIFFDEPTH (Ceiling: Different Depth) – This framing condition occurs when two areas of different depth end on a common bearing.

CPARALLEL (Ceiling: Parallel) – This framing condition occurs when two areas of the same depth end on a common bearing.

CANGLED (Ceiling: Angled) – This framing condition occurs when two areas meet on a wall, but the areas are neither parallel nor perpendicular to each other (the members run at some other angle).

CSIDERIM (Ceiling: Side Rim) – This framing condition occurs at the parallel sides of a framing area (as opposed to the ends).

CMIDSPANBLOC (Ceiling: Mid-Span Blocking) – This framing condition occurs in the middle of the span of joists.

COMMAND BUTTONS

Add Item – Click Add Item to add a solution type to the highlighted condition to the left. Click here for more information about Adding and Editing joist accessory solutions.

Move Up – The program will try the listed solutions based on their order in the list. It tries the first solution type listed, then the second, then the third, and so on until it finds a solution type that works for that condition. Highlight an item in the box to the left and then click the Move Up button to move it up in the list (and therefore have that solution tried earlier in the process).

NOTE: The Nothing type will always work for every condition, so you likely don't want that solution at the top of the list.

Move Down – The program will try the listed solutions based on their order in the list. It tries the first solution type listed, then the second, then the third, and so on until it finds a solution type that works

for that condition. Highlight an item in the box to the left and then click the Move Down button to move it down in the list (and therefore have that solution tried later in the process).

NOTE: The Nothing type will always work for every condition, so you likely don't want that solution at the top of the list.

Delete – Click this button to remove the highlighted solution from the list.

Edit – Click this button to edit the highlighted condition. Select a new type, change the material or edit the properties of the material such as cut length. [Click here for more information about Adding and Editing joist accessory solutions.](#)

OK –

Cancel –

Blocking & Rim Tab

The screenshot shows the 'Blocking & Rim' tab in a software interface. The window is titled 'General Info' and has several tabs: Job Info, Job Settings, Load Info, Framing Members, Other Materials, Panel Settings, Design Settings, Deflection Criteria, Joist Accessories, Blocking & Rim (selected), Material Lists, Cut Lengths, Colors & Line Types, and Plot Labels. The 'Blocking & Rim' tab is active and contains several sections:

- Blocking Settings:**
 - Don't label "typical" blocking
 - Combine blocking shorter than: [0] in
 - Report blocking as total pieces
- Joist Rim Settings:**
 - Combine rim shorter than: [0] ft
 - Report rim as total pieces
 - Ignore Rim Usage
 - Treat side rim as joist
- Mid-span Blocking Settings:**
 - Floor Joists every: [8] ft
 - Roof Joists every: [8] ft
 - Ceiling Joists every: [8] ft
- Rim nesting width:** [1] in
- Ladder Settings:**
 - Minimum ladder length: [0] ft
 - Ladder cut length: [0] ft
 - Place ladder starting from: [dropdown]

At the bottom are 'OK' and 'Cancel' buttons.

BLOCKING SETTINGS

Don't Label "Typical" Blocking – Select this check box if you don't want to label blocking panels that fall between two joists that are a standard distance apart (12", 16", 19.2" or 24"). Clear this check box to label every blocking panel on the plot. When you select this option, the Report blocking as total pieces option will be unavailable.

Combine Blocking Shorter Than – Select this check box to combine all blocking panels shorter than the specified length into one overall piece. Blocking panels longer than the specified length will be maintained individually. The length should be entered in decimal inches.

Report Blocking as Total Pieces – Select this box to report blocking as a total number of pieces to the materials list for plots and output files. The number of pieces will be determined by the total length of required blocking divided by the cut length you entered for this type of blocking on the Joist Accessories tab. When you select this option, the Don't label "typical" blocking and Combine blocking shorter than options will be unavailable.

JOIST RIM SETTINGS

Combine Rim Shorter Than – Select this check box to combine all rim shorter than the length specified into one overall piece. Rim longer than the specified length will be maintained individually. The length should be entered in decimal feet. When you select this option, the Report rim as total pieces option will be unavailable.

Report Rim as Total Pieces – Select this box to report rim as a total number of pieces to the materials list for plots and output files. The number of pieces will be determined by the total length of required rim divided by the cut length you entered for this type of rim on the Joist Accessories tab. When you select this option, the Combine rim shorter than option will be unavailable.

Ignore Rim Usage – Select this option to ignore the usage of the rim when consolidating pieces. This results in better consolidation since end rim and side rim might be the same material, for example.

Treat Side Rim as Joist – Select this box to treat the side rim as a joist. This allows the side rim material to be labeled the same as the joist area on plots.

Rim Nesting Width –

MID-SPAN BLOCKING SETTINGS

Floor Joists Every – Enter a length in feet, for Imperial, or mm, for Metric. Any floor joist area that is greater than the length specified will receive mid-span blocking.

Roof Joists Every – Enter a length in feet, for Imperial, or mm, for Metric. Any roof joist area that is greater than the length specified will receive mid-span blocking.

Ceiling Joists Every – Enter a length in feet, for Imperial, or mm, for Metric. Any ceiling joist area that is greater than the length specified will receive mid-span blocking.

LADDER SETTINGS

Minimum Ladder Length – Enter the minimum distance ladder members should be drawn from the outside edge of the rim. If the defined Minimum Ladder Length is greater than the distance from the outside edge of the rim to the first joist member, ladder members will also be placed to the first and second joists.

Ladder Cut Length – This is the cut length to which the application will consolidate the ladder materials if the Report Blocking as Total Pieces option is checked above.

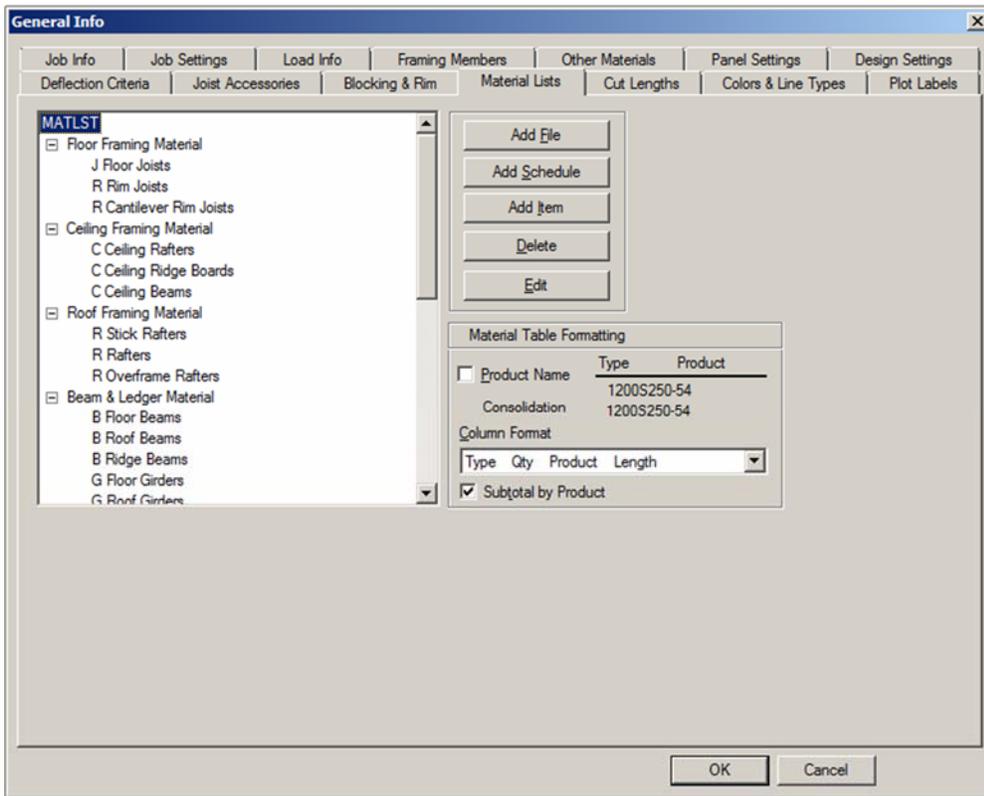
Place Ladder Starting From – Choose either Mid span or From End. If you choose Mid span, ladder members will be placed from the mid-point of the rim and then measured out towards the ends. If you choose From End, ladder members will be placed the specified Ladder Spacing from the end point of the rim and then placed at the specified Ladder Spacing along the rim.

COMMAND BUTTONS

OK –

Cancel –

Material Lists Tab



This window allows you to configure the material lists shown on your plots by placing types of materials into different sections and naming those sections.

MATLST – This box lists all the types of members found in the program and where they will be located in the materials list in the plot program. This box displays a tree of names, the first name being the *File* name, the second are the *Schedules*, and the last are the *Items*, each of which are described below.

- *File* – (Default is MATLST) This name is what you want the materials list layout to be saved as. This name must be six or fewer characters with no spaces. To add a file, click Add File. To edit the file name, select it and click Edit.
- *Schedules* – These are the heading or section names for the material list. These names can be formatted however you want and there is no limit to how many you can have. To edit a schedule name, select it and click Edit. To add a New Schedule, click Add Schedule.
- *Items* – Items are types of materials such as floor joists and roof girders. Items should be placed under the schedule where you want them to appear in the materials list. For example, you may want a schedule named Headers and you might want to add the roof headers and floor headers items to that schedule. You cannot control the item names. However, you do have the ability to control where in the list the item is placed, and what single letter will be used to label that item.

To move an item into another schedule, drag the item name to the appropriate schedule. The item will be added to the bottom of that schedule's list. To edit an item, select the item and click Edit. To remove an item from the materials list, select the item name and click Delete.

COMMAND BUTTONS

Add File – The file name is what you want the materials list layout to be saved as (the default is MATLST). Click Add to add a new one. This name must be six or fewer characters with no spaces. To edit the file name, select it and click Edit.

Add Schedule – Click Add Schedule to add a new heading or section name to your materials list. These names can be formatted however you choose and there is no limit to how many you can have.

Add Item – Click Add Item to add a new item to a schedule. The items should be placed under the Schedule as you want them to appear in the materials list. The item names are not user controlled and are chosen from a pre-defined list. You do, however, have the ability to control where in the list the item is placed, and what single letter will be used to label that item. If you click Add Item and the item you wish to add is not in the list, it has most likely already been added to another schedule.

Delete – Click Delete to delete the selected item.

Edit – Click Edit to edit the selected item.

MATERIAL TABLE FORMATTING

Product Name Consolidation – Select Product Name Consolidation if you do NOT want like items to have their names printed twice. If the product on line 1 matches the product of line 2, line 2 will show "v" symbols in the product name location to show that the product is the same as the one above. Clear this check box if you want do not wish to consolidate the product names. The product names will be listed again even if the product on line 2 is identical to the product on line 1. The display to the right will change automatically to show you which option is currently selected and how it will affect your materials list.

Column Format – Choose the order or the columns on the materials list, either: Type, Quantity, Product, Length (the format specified by NDS) or Type, Quantity, Length, Product.

Subtotal by Product – Select Subtotal by Product to print the subtotaled length under each product type on the materials list.

Cut Lengths Tab

General Info

Job Info | Job Settings | Load Info | Framing Members | Other Materials | Panel Settings | Design Settings

Deflection Criteria | Joist Accessories | Blocking & Rim | Material Lists | Cut Lengths | Colors & Line Types | Plot Labels

Members will be cut to the nearest length indicated; zero is not a valid cut length.

Glulams in (-1 means nearest foot for <8'
nearest even foot for >8')

Steel Beams in

LGS in

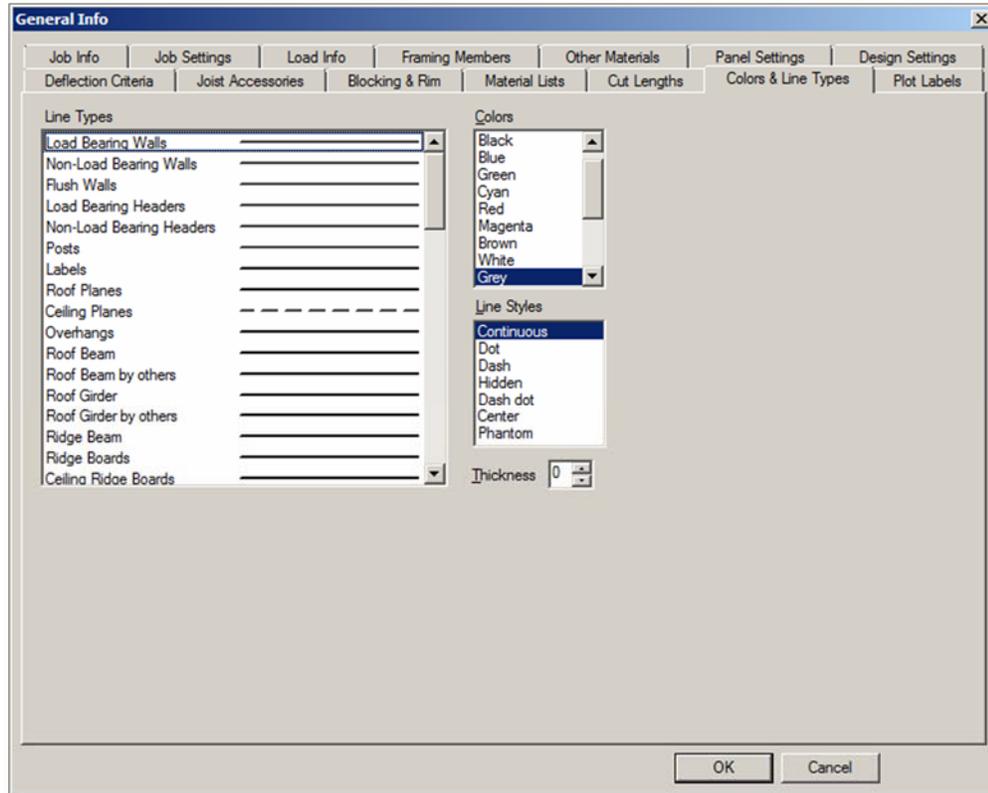
OK Cancel

Glulams – Enter what increment, in decimal inches, the program should use when calculating cut lengths. Type -1 to track materials by the foot for members under eight feet and the nearest two feet for members eight feet or greater. Type 24 to track materials by the nearest even foot, which is the typical way that glulams are stocked and shipped. Type 1 to track material to the nearest foot. Type .0625 (1/16") to report the length to the nearest sixteenth of an inch. This cut length is for any member added using the Glulam material type.

Steel Beams – Enter what increment, in decimal inches, the program should use when calculating cut lengths. Since steel beams are usually manufactured to the exact specifications of the plan, leave this at the default of 0.625. This cut length affects only hot formed steel beams, which are selected from the Steel product type.

LGS –

Color & Line Types Tab



LINE TYPES

This is a list of all the different members that can be displayed on plots. To the right of each member name is a sample of the line being used to draw that member. You can change how these lines look on your plots by changing the line properties using the fields to the right (colors and line styles).

COLORS

This is the color that will be used when the selected member is drawn in the plot program. If <None> is selected, then the selected member will not be drawn at all in the plot program. If you are using a pen plotter (as opposed to laser or ink jet), you will need to set up your pen numbers. The pen number for each color is listed below:

- Black – Pen 0
- Blue – Pen 1
- Green – Pen 2
- Cyan – Pen 3
- Red – Pen 4
- Magenta – Pen 5
- Brown – Pen 6
- White – Pen 7
- Grey – Pen 8
- Bright Blue – Pen 9
- Bright Green – Pen 10
- Bright Cyan – Pen 11
- Bright Red – Pen 12
- Bright Magenta – Pen 13
- Bright Yellow – Pen 14
- Bright White – Pen 15

NOTE! The colors that are used in the plot program may not correspond to the colors of the pens in your plotter. The displayed colors are only used to differentiate where a different pen will be used. Also, Pen #0 (Black) cannot be used by many plotters. You should verify that your plotter will plot with a pen #0 before assigning any members to this pen.

LINE STYLES

The Line Styles list displays all of the line types available. Click a line type to and then select a style for that type from the list. The selected line type will change to reflect the line style you selected.

[OTHER]

Thickness – Select the thickness for the selected line type. As you change the size, the size will change in the Line Types box as well. Thickness is an effective tool for making good quality plots.

Plot Labels Tab

Member Type	Label Format
Joists	Consolidation Label
Beams	Consolidation Label
Rim	Consolidation Label
Blocking	Consolidation Label
Hangers	Consolidation Label
Posts	Consolidation Label

SETTINGS

Label Prefix – Type a one or two character label prefix to be applied to the front of each label on the plots.

Label # Style – For labels with single digits, you can choose whether to display a leading 0 (zero), for example 1 or 01.

Include Spacing w/ Label (Joist) –

Spacing Suffix –

Include Plies w/ Label (Beam and Girder) –

Ply Suffix –

PLOT LABEL FORMATTING

Select the member for which you want to format a label.

JOISTS

- *Consolidation Label – Product (Qty / Cut Length)* –
- *Calc Label – Product (Qty / Cut Length)* –
- *Consolidation Label* – Select this option to generate a consolidation label for each group of like materials (same material and length). Fewer consolidation labels are generated than

calc labels because the calc labels take into account loading. For example, J3, where all joists of the same material and length receive the same label. The information that you enter is displayed in the Preview field.

- *Calc Label* – Select this option to generate a calc label for each group of like materials based on their material, span, and loading. More calc labels are generated than consolidation labels because calc labels take into account loading. For example, A2, where all joists of the same material, length and loading all receive the same label. The information that you enter is displayed in the Preview field.

BEAMS

- *Consolidation Label – Product (Plies X Cut Length)* – For members that have a number of plies, such as beams and girders, choose whether to display the plies with label. You can include the number of plies on the label. The information that you enter is displayed in the Preview field. Select this option to add cut length to the label format. The information that you enter is displayed in the Preview field.
- *Calc Label – Product (Plies X Cut Length)* – For members that have a number of plies, such as beams and girders, choose whether to display the plies with label. You can include the number of plies on the label. The information that you enter is displayed in the Preview field. Select this option to add cut length to the label format. The information that you enter is displayed in the Preview field.
- *Consolidation Label* – Select this option to generate a consolidation label for each group of like materials (same material and length). Fewer consolidation labels are generated than calc labels because the calc labels take into account loading. For example, J3, where all joists of the same material and length receive the same label. The information that you enter is displayed in the Preview field.
- *Calc Label* – Select this option to generate a calc label for each group of like materials based on their material, span, and loading. More calc labels are generated than consolidation labels because calc labels take into account loading. For example, A2, where all joists of the same material, length and loading all receive the same label. The information that you enter is displayed in the Preview field.

RIM

- *Product* – Select this option to add product descriptions to the label format. The information that you enter is displayed in the Preview field.
- *Consolidation Label* – Select this option to generate a consolidation label for each group of like materials (same material and length). Fewer consolidation labels are generated than calc labels because the calc labels take into account loading. For example, J3, where all joists of the same material and length receive the same label. The information that you enter is displayed in the Preview field.

BLOCKING

- *Product* – Select this option to add product descriptions to the label format. The information that you enter is displayed in the Preview field.
- *Consolidation Label* – Select this option to generate a consolidation label for each group of like materials (same material and length). Fewer consolidation labels are generated than calc labels because the calc labels take into account loading. For example, J3, where all joists of the same material and length receive the same label. The information that you enter is displayed in the Preview field.

HANGERS

- *Product* – Select this option to add product descriptions to the label format. The information that you enter is displayed in the Preview field.
- *Consolidation Label* – Select this option to generate a consolidation label for each group of like materials (same material and length). Fewer consolidation labels are generated than calc labels because the calc labels take into account loading. For example, J3, where all joists of the same material and length receive the same label. The information that you enter is displayed in the Preview field.

POSTS

- *Product* – Select this option to add product descriptions to the label format. The information that you enter is displayed in the Preview field.
- *Consolidation Label* – Select this option to generate a consolidation label for each group of like materials (same material and length). Fewer consolidation labels are generated than calc labels because the calc labels take into account loading. For example, J3, where all joists of the same material and length receive the same label. The information that you enter is displayed in the Preview field.

LABEL SIZE

General – Type the label size, in decimal inches, for general labels on the Plot.

Hanger – Type the label size, in decimal inches, for hanger labels on the Plot.

Calc ID – Type the label size, in decimal inches, for Calc ID labels on the Plot.

Framer's Plot: Point Loads by Load Case – Select this option to display point loads (typically posts) on the Framer's Plot by Load Case. The loads are separated into live, dead, snow, and others. They are displayed on the Framer's Plot (for example, 137# live, 40# dead, 400# snow). If you are using a Canadian building code, these loads are un-factored. If you do not select this option, point loads displayed on the Framer's Plot are the total load (for example, 2200#). If you are using a Canadian building code, the total load is the total factored load.

Pass Thru Framing Minimum Point Load –

Custom Beam and Girder by Others Labels –

- *Beam by Others* –
- *Girder by Others* –

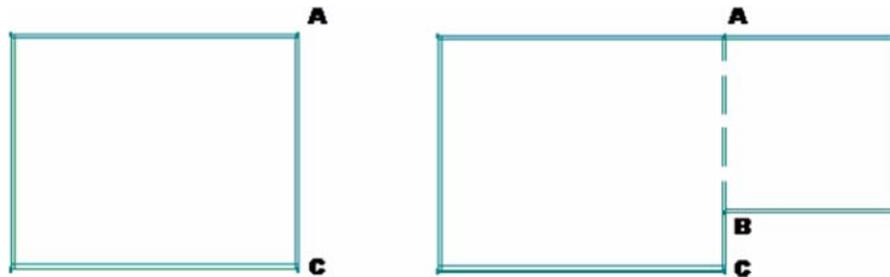
WORKING WITH OPTIONS

Drawing an Option

Drawing option entities is essentially the same procedure as drawing base house entities. You have access to the same entity types, the same *Modify* commands, the same *Display* commands, and the same *Macros* as if you were drawing any GS Plan model. The only difference is that, in *Draw Option* mode, you cannot alter any existing base house entity.

To underscore this restriction, all base house entities will appear to be shadowed (dark blue color). You can select a base house entity to check its properties or to trim/extend an option entity to it. You can also delete it from the option (but not from the base house) by selecting it and pressing *Delete*. All other *Modify* commands will be disabled (grayed out) for the selected base house entity.

Deleting a base house entity from an option is an important concept. Take the example presented below in Figure X, in which we want to model a simple rectangular house with an optional rectangular structure to the right.



To provide physical access to the option, you should (while drawing the option):

1. Delete the base house wall from A to C.
2. Draw a beam from A to B.
3. Draw a wall from B to C.

Other option entities are also possible (e.g. a door header in an option wall from A to C), but they all begin with deletion of the original base house wall.

Other base house entities are handled similarly. For example, suppose you wanted to extend the framing area (in the level below) to include floor joists beneath the option. You would (while drawing the option):

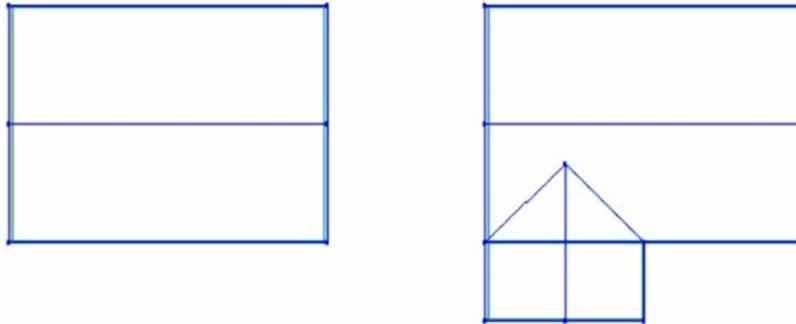
1. Delete the base house framing area. (The associated framing members will also disappear.)
2. Draw a new framing area around the base house and the option. (The associated framing members will then become part of the current option.)

Your model will now have entities on two levels belonging to the same option, a common situation. Note that both framing areas and their respective framing members will co-exist in the .may file when you save the job. Thus, base house entities that have been deleted from an option are not really forgotten, they are just suppressed until you return to the base house.

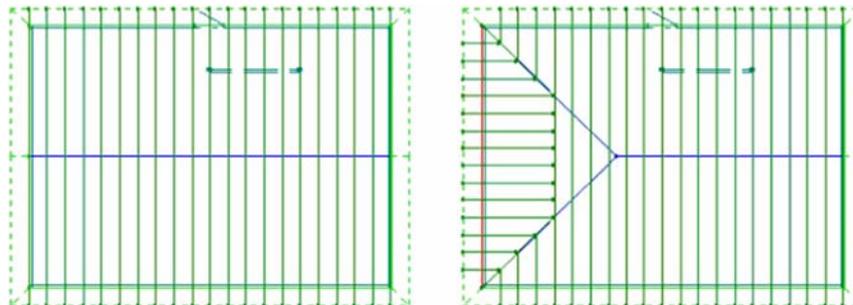
Using the *Modify* → *Undo* command (to reverse the effect of a previous single-entity modification) acquires special behavior when used in conjunction with options. If you modify an option entity – including deletion of a base house entity – and then undo the modification, the status quo will be

restored exactly as if you modified and "undid" a base house entity. However, if you modify the entity and then restore the base house or switch to a different option, the *Undo* command will be disabled; you cannot modify an entity belonging to one option and then recover it in another. To enable the *Undo*, you must first return to the option from which the modification took place. Of course, since *Undo* only recalls the previous *Modify* command, any subsequent single-entity modification will preclude "undoing" the original option modification.

Although you cannot modify a base house entity in *Draw Option* mode, you can still use it for reference purposes. As mentioned above, you can select it for use in trimming/extending an option entity. You can choose a base house entity as the parallel side of an option framing area. You can extend option planes to base-house planes (but not vice-versa), as illustrated by the following optional front wing:



However, you cannot select a base house framing area or plane for use in constructing a hip end. To draw the hip roof option pictured in Figure X, you should first delete the two gable roof planes and framing area belonging to the base house (left). Then draw and extend the three option planes and a new framing area (over the same region as the base house area). Finally, draw the hip end line. The result will be a separate set of planes, area, framing members and hip members belonging just to the hip-end option.



Overhang lines belonging to the base house should likewise be deleted and redrawn (as part of the option), unless they intersect option planes along the same 3D lines. In the previous example, the two left overhangs belonging to the base house were deleted and a new overhang line drawn for the hip face plane. The other four base house overhangs retain their original geometry and do not have to be redrawn.

Dimension lines always belong to the option being drawn. Since dimension lines are not associative, GS Plan cannot tell whether a dimension line really applies to a particular option; therefore, you should be careful not to dimension base house entities in *Draw Option* mode. Base house entities will be visible while you are drawing option dimensions, but base house dimension lines will not. Indeed, it is not possible to "delete" a base house dimension line (from a specific option) the way you can "delete" a base house entity.

If you need to dimension a plan including two or more options simultaneously, you should extract the model to a .lay file and create "hybrid" dimension lines from Structure.

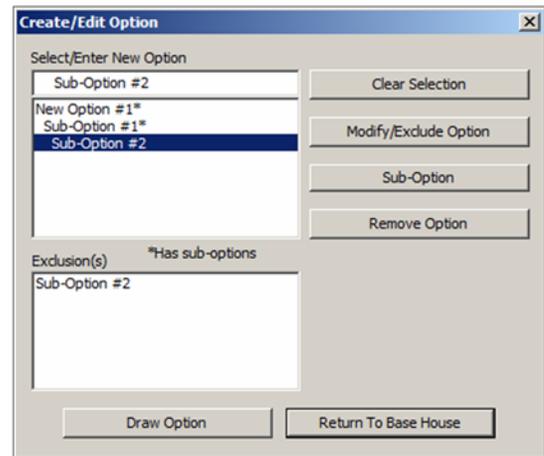
Adding a Sub-Option

To add a sub-option, select its parent (the option to which the sub-option will belong) and click on the *Sub-Option* button. In this example, assume we want to add the sub-option "Three-Car Garage" under "Elevation A." After clicking *Sub-Option*, we have typed "Three-Car Garage" under *Select/Enter New Option*, followed by the *Add/Modify Option* button. The results are depicted in Figure X below.

Note the asterisk now appearing after "New Option #1", denoting that "New Option #1" now has one or more sub-options. Also note that the sub-option name appears under its parent option, slightly.

We could also continue adding more sub-option names by pressing the *Clear Selection* button (to clear the current selection) and type another *Sub-Option* name. The new *Sub-Option* (Sub-Option #2) is added underneath *Sub-Option #1* and also shown as indented.

When you are drawing a *Sub-Option*, entities belonging to all higher levels of the current hierarchy will be displayed in shadow mode. Thus, you can delete entities from the current *Sub-Option*, regardless of which level of the hierarchy they belong to, as long as they are **forebears** of the current option. In the previous example, you could delete "New Option #1" entities from "Sub-Option #1", just as you could delete entities belonging to the base house itself. (In this context we really mean "pseudo-delete", because the deleted entities remain as part of the model when visibility of the corresponding parent option is restored.)



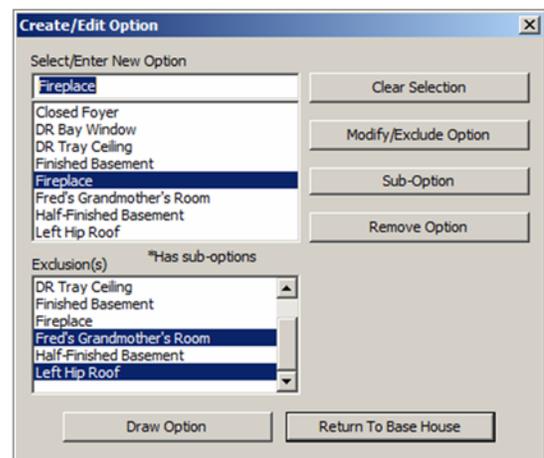
Excluding Options from Each Other

To eliminate the possibility that a novice user might select or extract conflicting options, the Create/Edit Option dialog box allows you to identify "mutually exclusive" options. From the upper list box, select any one of the conflicting options. Then in the lower list box [Exclusion(s)], select those options you wish to exclude. Finally, click on the Modify/Exclude button. Thereafter, in the Select Options dialog box, it will no longer be possible to select conflicting options.

An example of the *Create/Edit Option* dialog box, with mutually excluded options, is shown in Figure X at right.

It is not necessary to exclude options reciprocally. That is to say, if Option A excludes options B and C, Option B will automatically exclude A and C, and Option C will automatically exclude A and B.

To reverse the exclude process, simply re-select one of the excluded options in the upper list box and de-select the other one(s) in the lower list box; then click again on the *Modify/Exclude Option* command button.



Do not confuse excluded *Options* with "excluded base house entities." (i.e. base house entities that have been deleted from a particular option.) The two concepts have nothing to do with each other.

Using Macros with Options

In general you can use *Macro* commands (*Generate Members*, *Quick Header*, etc.) in *Draw Option* mode, just as you can when drawing the base house. (Bearing in mind the usual proviso that base house entities can be referenced but not modified.) The following paragraphs outline specific rules that apply to each macro. For the sake of conciseness, "current option" means whatever option currently appears at the top of the *Create/Edit Option* dialog box; "base house" refers to base house entities not excluded (i.e. deleted) from the current option.

1. *Generate Members* is restricted to framing areas and hip ends belonging to the current option (and, of course, on the active level).
2. *Quick Header* will only insert headers into walls belonging to the current option. (The reason base house walls are not eligible is that the insertion process subdivides a wall into two segments, which would be an implicit modification of a base house entity.)
3. *Move Header* will only reposition a header belonging to the current option.
4. *Verify Components* will display designated window and door components for headers belonging to both the base house and the current option, but will only allow modifications to the latter.
5. *Delete Header* will only remove headers belonging to the current option.
6. *Extrude Bearings* will extrude walls (including header walls) of the current option to planes belonging either to the base house or to the current option.
7. *Create Planes* will create planes around a designated exterior boundary, regardless of whether boundary lines are comprised of base house or option entities. However, the auto-extend macros (*Extend Single* and *Extend All*) will only pay attention to option planes. Thus, you can readily create a completely different roof system as an option for the same base house walls. You can also auto-create a roof system for a separate option structure (such as a detached garage) without affecting the base house. But if you are simply creating roof planes for a pop-out (e.g. a bay window option), you are better off drawing and extending them manually.
8. *Ceiling Planes*, *Skylight* and *Dormer* macros will display "live dimensions" to either base house or option entities, whichever is closer. Of course the planes, openings, walls, gables and framing members thereby created will all belong to the current option.
9. Toggle macros (*Load-Bearing*, *Roof/Floor*, *Interior/Exterior*, *Wall/Header Widths*) will display corresponding properties of, and allow modification to, current option entities only. The base house will be shadowed for reference purposes.
10. Stick-framing macros (*Stick Areas* and *Ridge Boards*) will create rafters or ridge boards for planes belonging either to the base house or to the current option

GLOSSARY OF TERMS

B/C Upset – See “Bottom Chord Upset.”

Beam –

Bottom Chord Upset – The distance of the vertical cut at the outside end of a bottom chord of a truss that is left flat when it is beveled.

Cut Joist –

Dimension Line – A *Dimension Line* is an annotation that is used to display the distance between two points. *Dimension Lines* are saved with the job and included in *Layout Plots* generated from GS Plan. Please note that they will not be included in plots from **KeyBuild Structure**.

Flush Wall –

Gable – Non-structural trusses that are placed at exposed ends of truss layouts.

Girder – An element that provides the primary support for loads.

Girder by Others –

Header –

Heel Height – The vertical distance from the wall to the top of the plane.

Ledger –

Overhang – The part of the roof frame that extends past the support bearings.

Ridge Board – A non-structural member that is used to keep **rafters** aligned.

Seat Cut – The amount of the rafter that sits flat on the bearing when the ceiling plane slopes upward from the wall. In other words, the flat distance from the outside of the wall to the beginning of the slope.

Slope – The roof line's degree of incline based on the ratio of its rise to its run.

Snap Point – A snap point is a specified location on the snap grid or at the end of a member to which the user's cursor can automatically align.

Strut –

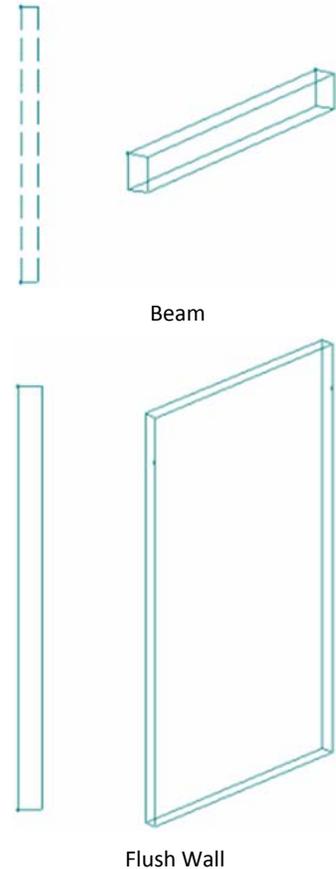
T/C Depth – See “Top Chord Depth.”

Top Chord Depth – The width of the roof framing member used as the top chord in a truss or the width of the top part of a rafter.

Truss Girder –

Valley – The place where two sloping roof planes meet.

Wall –



Set Bearing Properties for a Wall

Bearing Type – Wall

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field defines the height of your new wall at its starting point.

Flat Btm. –

End Height – This field defines the height of your new wall at its end point.

Flat Btm. –

Plies – This field is disabled when *Wall* is selected as the *Bearing Type*.

Ply Width – The *Ply Width* field defines the thickness of a single ply of material used for construction. For walls, this is the width of the plate.

Mbr Depth – This field is disabled when *Wall* is selected as the *Bearing Type*.

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your wall.

Location – The *Location* field designates your wall as existing on the *Interior* or *Exterior* of the structure. This classification effects the framing rules that will be applied to the wall's construction.

Label – This field is disabled when *Wall* is selected as the *Bearing Type*.

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Wall* that is created. The user will not be prompted to define the properties again for additional walls.

Field-Framed – This field is disabled when *Wall* is selected as the *Bearing Type*.

Floor – When the *Floor* checkbox is activated, the wall being defined is designated as supporting floor-framing members.

Non-Load Bearing – When this checkbox is activated, the *Wall* is treated by GS Plan as non-structural.

Bears on Earth – When this checkbox is activated, the *Wall* (whose properties are being defined within this dialog box) serves as the ultimate and final destination for all loads applied to it. No loads are transferred to bearings beneath this wall.

Wall Type – As the name implies, the *Wall Type* field identifies the kind of wall being defined. There are three options: *Framed*, *Foundation*, and *Non-Framed*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Reserved – This command button is disabled when *Wall* is selected as the *Bearing Type*.

Products – This command button is disabled when *Wall* is selected as the *Bearing Type*.

Set Bearing Properties for a Girder

Bearing Type – Girder

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field is disabled when *Girder* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Girder* is selected as the *Bearing Type*.

End Height – This field is disabled when *Girder* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Girder* is selected as the *Bearing Type*.

Plies – This field defines the number of plies of material to be used in the construction of your girder.

Ply Width – The *Ply Width* field defines the thickness of a single ply of the material that will be used to construct your girder.

Mbr Depth – The *Member Depth* field is used to specify the depth dimension of the girder being defined. (See the figure at right.)



Member Depth

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your girder.

Location – The *Location* radio buttons are disabled when *Girder* is selected as the *Bearing Type*.

Label –

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Girder* that is created. The user will not be prompted to define the properties again for additional girders.

The screenshot shows the 'Set Bearing Properties' dialog box. The 'Bearing Type' list has 'Girder' selected. The 'Reference Line' section has radio buttons for 'Left', 'Center', and 'Right', with 'Right' selected. The 'Beg. Height' and 'End Height' fields are both set to '0-0-0' and have 'Flat Btm.' buttons next to them. The '# Plies' field is set to '1', 'Ply Width' is '3-8', and 'Mbr Depth' is '12-0'. The 'Product' field shows 'No Product Assigned'. The 'Location' section has radio buttons for 'Interior' and 'Exterior', with 'Exterior' selected. There is a 'Label' text box. At the bottom, there are checkboxes for 'Continuous' and 'Field-Framed', and a group of checkboxes for 'Ledgered', 'Floor', 'Define Posts', 'Non-load bearing', and 'Bears on Earth'. At the very bottom are buttons for 'OK', 'Delete', 'Reserved', and 'Products'.

Field-Framed – When the *Field-Framed* checkbox is activated, the girder being defined will not appear on individual **panel plots** and will not be included in the materials lists for those plots.

Ledgered – If the *Ledgered* checkbox is activated, GS Plan permits framing members to hang from the side of the girder, rather than having to be supported by the top of the girder.

Define Posts –

Floor – When the *Floor* checkbox is activated, the girder being defined is designated as supporting floor-framing members.

Non-Load Bearing – This field is disabled when *Girder* is selected as the *Bearing Type*.

Bears on Earth – This field is disabled when *Girder* is selected as the *Bearing Type*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Reserved – This command button is disabled when *Girder* is selected as the *Bearing Type*.

Products – Clicking on the *Products* command button will open a *PST Product Priority* dialog box, which can be used to add to, or modify, the list of materials available for use in the construction of your girder. For more information about the features and functions of the *PST Product Priority* dialog box, see the section within this document entitled “*Defining Product Priorities*.”

Set Bearing Properties for a Beam

Bearing Type – Beam

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field is disabled when *Beam* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Beam* is selected as the *Bearing Type*.

End Height – This field is disabled when *Beam* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Beam* is selected as the *Bearing Type*.

Plies – This field defines the number of plies of material to be used in the construction of your

The screenshot shows the 'Set Bearing Properties' dialog box. The 'Bearing Type' list has 'Beam' selected. The 'Reference Line' section has 'Right' selected. The 'Beg. Height' and 'End Height' fields are both set to '0-0-0'. The '# Plies' field is set to '1'. The 'Ply Width' field is set to '3-8'. The 'Mbr Depth' field is set to '12-0'. The 'Product' field is set to 'No Product Assigned'. The 'Location' section has 'Exterior' selected. The 'Label' field is empty. The 'Continuous' and 'Field-Framed' checkboxes are unchecked. The 'Define Posts', 'Floor', 'Non-load bearing', and 'Bears on Earth' checkboxes are also unchecked. The 'OK', 'Delete', 'Beam Type', and 'Products' buttons are visible at the bottom.

beam.

Ply Width – The *Ply Width* field defines the thickness of a single ply of the material that will be used to construct your beam.

Mbr Depth – The *Member Depth* field is used to specify the depth dimension of the beam being defined. (See the figure at right.)



Member Depth

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your beam.

Location – The *Location* radio buttons are disabled when *Girder* is selected as the *Bearing Type*.

Label –

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Beam* that is created. The user will not be prompted to define the properties again for additional beams.

Field-Framed – When the *Field-Framed* checkbox is activated, the beam being defined will not appear on individual **panel plots** and will not be included in the materials lists for those plots.

Define Posts –

Floor – When the *Floor* checkbox is activated, the beam being defined is designated as supporting floor-framing members.

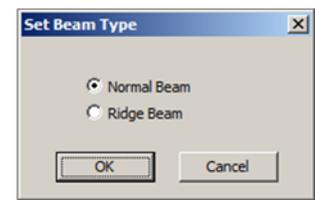
Non-Load Bearing – This field is disabled when *Girder* is selected as the *Bearing Type*.

Bears on Earth – This field is disabled when *Girder* is selected as the *Bearing Type*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Beam Type – Clicking on the *Beam Type* command button will open a *Set Beam Properties* dialog box like the one pictured at right. Use this dialog box to designate the beam as one of the following:



- *Normal Beam* – When *Normal Beam* is selected, the beam will extend down from the **top of plate height** and support framing members from below.
- *Ridge Beam* – When *Ridge Beam* is selected, the beam will extend down from the roof plane that it is placed below, to support framing members from below.

Products – Clicking on the *Products* command button will open a *PST Product Priority* dialog box, which can be used to add to, or modify, the list of materials available for use in the construction of your beam. For more information about the features and functions of the *PST Product Priority* dialog box, see the section within this document entitled “*Defining Product Priorities*.”

Set Bearing Properties for a Header

Bearing Type – Header

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field defines the height of your new header at its starting point.

Flat Btm. –

End Height – This field defines the height of your new header at its end point.

Flat Btm. –

Plies – This field is disabled when *Header* is selected as the *Bearing Type*.

Ply Width – The *Ply Width* field defines the thickness of a single ply of the material that will be used to construct your header.

Mbr Depth – This field is disabled when *Header* is selected as the *Bearing Type*.

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your header.

Location – The *Location* field designates your header as existing on the *Interior* or *Exterior* of the structure. This classification effects the framing rules that will be applied to the header's construction.

Label – This field is disabled when *Header* is selected as the *Bearing Type*.

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Header* that is created. The user will not be prompted to define the properties again for additional headers.

Field-Framed – This field is disabled when *Header* is selected as the *Bearing Type*.

Floor – When the *Floor* checkbox is activated, the header being defined is designated as supporting floor-framing members.

Non-Load Bearing – When this checkbox is activated, the *Header* is treated by GS Plan as non-structural.

Bears on Earth – This field is disabled when *Girder* is selected as the *Bearing Type*.

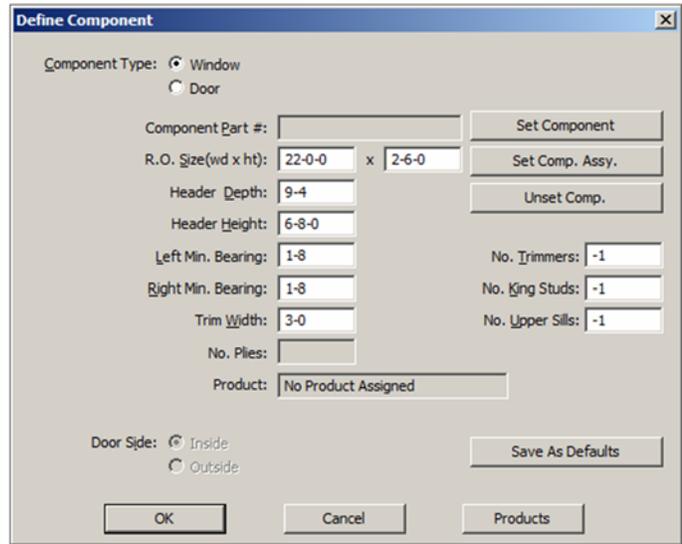
Wall Type – As the name implies, the *Wall Type* field identifies the kind of wall in which the header being defined will be placed. There are three options: *Framed*, *Foundation*, and *Non-Framed*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Header Info – Clicking on the *Header Info* command button will open a *Define Component* dialog box, like the one pictured at right. For more information about the features and functions associated with this dialog box, see “*Defining Component Properties*” on page ___.

Products – Clicking on the *Products* command button will open a *PST Product Priority* dialog box, which can be used to add to, or modify, the list of materials available for use in the construction of your header. For more information about the features and functions of the *PST Product Priority* dialog box, see the section within this document entitled “*Defining Product Priorities*.”



Set Bearing Properties for a Truss Girder

Bearing Type – Truss Girder

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field is disabled when *Truss Girder* is selected as the *Bearing Type*.

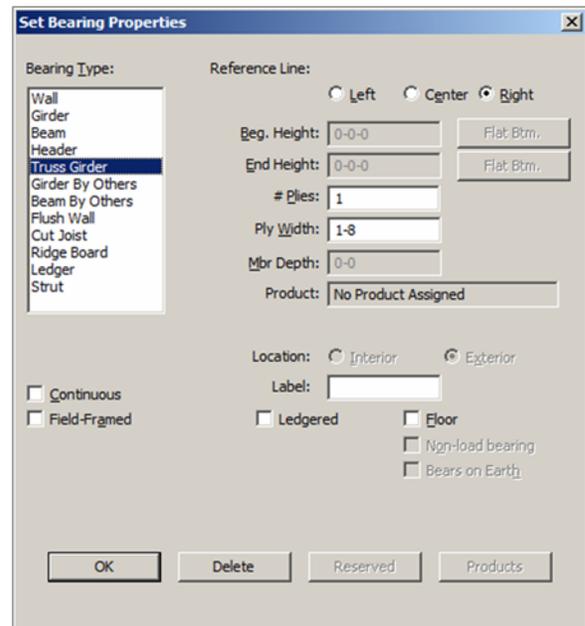
Flat Btm. – This field is disabled when *Truss Girder* is selected as the *Bearing Type*.

End Height – This field is disabled when *Truss Girder* is selected as the *Bearing Type*.

Flat Btm. – This field is disabled when *Truss Girder* is selected as the *Bearing Type*.

Plies – This field defines the number of plies of material to be used in the construction of your truss girder.

Ply Width – The *Ply Width* field defines the thickness of a single ply of the material that will be used to construct your truss girder.



Mbr Depth – This field is disabled when *Truss Girder* is selected as the *Bearing Type*.

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your truss girder.

Location – The *Location* radio buttons are disabled when *Truss Girder* is selected as the *Bearing Type*.

Label – This field is disabled when *Header* is selected as the *Bearing Type*.

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Truss Girder* that is created. The user will not be prompted to define the properties again for additional truss girders.

Field-Framed – When the *Field-Framed* checkbox is activated, the truss girder being defined will not appear on individual **panel plots** and will not be included in the materials lists for those plots.

Ledgered – If the *Ledgered* checkbox is activated, GS Plan permits framing members to hang from the side of the truss girder, rather than having to be supported by the top of the girder.

Floor – When the *Floor* checkbox is activated, the truss girder being defined is designated as supporting floor-framing members.

Non-Load Bearing – This field is disabled when *Truss Girder* is selected as the *Bearing Type*.

Bears on Earth – This field is disabled when *Truss Girder* is selected as the *Bearing Type*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Reserved – This command button is disabled when *Truss Girder* is selected as the *Bearing Type*.

Products – This command button is disabled when *Truss Girder* is selected as the *Bearing Type*.

Set Bearing Properties for a 'Girder by Others'

Bearing Type – *Girder by Others*

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field is disabled when *Girder by Others* is selected as the *Bearing Type*.

Flat Btm. – This field is disabled when *Girder by Others* is selected as the *Bearing Type*.

The screenshot shows the 'Set Bearing Properties' dialog box. The 'Bearing Type' list on the left has 'Girder by Others' selected. The 'Reference Line' section has radio buttons for 'Left', 'Center', and 'Right', with 'Right' selected. The 'Beg. Height' and 'End Height' fields are both set to '0-0-0' and have 'Flat Btm.' buttons next to them. The '# Plies' field is set to '1' and the 'Ply Width' field is set to '3-8'. The 'Mbr Depth' field is set to '12-0'. The 'Product' field is 'No Product Assigned'. The 'Location' section has radio buttons for 'Interior' and 'Exterior', with 'Exterior' selected. The 'Label' field is empty. There are several checkboxes: 'Continuous', 'Field-Framed', 'Ledgered', 'Floor', 'Define Posts', 'Non-load bearing', and 'Bears on Earth', all of which are currently unchecked. At the bottom of the dialog are four buttons: 'OK', 'Delete', 'Reserved', and 'Products'.

End Height – This field is disabled when *Girder by Others* is selected as the *Bearing Type*.

Flat Btm. – This field is disabled when *Girder by Others* is selected as the *Bearing Type*.

Plies – This field defines the number of plies of material to be used in the construction of a ‘*Girder by Others*.’

Ply Width – The *Ply Width* field defines the thickness of a single ply of the material that will be used to construct a ‘*Girder by Others*.’

Mbr Depth – The *Member Depth* field is used to specify the depth dimension of the ‘*Girder by Others*’ being defined. (See the figure at right.)



Member Depth

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your ‘*Girder by Others*.’

Location – The *Location* radio buttons are disabled when *Girder by Others* is selected as the *Bearing Type*.

Label – This field is disabled when *Girder by Others* is selected as the *Bearing Type*.

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Girder by Others* that is created. The user will not be prompted to define the properties again for additional girders by others.

Field-Framed – When the *Field-Framed* checkbox is activated, the girder being defined will not appear on individual **panel plots** and will not be included in the materials lists for those plots.

Ledgered – If the *Ledgered* checkbox is activated, GS Plan permits framing members to hang from the side of the girder, rather than having to be supported by the top of the girder.

Define Posts –

Floor – When the *Floor* checkbox is activated, the ‘*Girder by Others*’ being defined is designated as supporting floor-framing members.

Non-Load Bearing – This field is disabled when *Girder by Others* is selected as the *Bearing Type*.

Bears on Earth – This field is disabled when *Girder by Others* is selected as the *Bearing Type*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Reserved – This command button is disabled when *Girder by Others* is selected as the *Bearing Type*.

Products – This command button is disabled when *Girder by Others* is selected as the *Bearing Type*.

Set Bearing Properties for a ‘Beam by Others’

Bearing Type – *Beam by Others*

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field is disabled when *Beam by Others* is selected as the *Bearing Type*.

Flat Btm. – This field is disabled when *Beam by Others* is selected as the *Bearing Type*.

End Height – This field is disabled when *Beam by Others* is selected as the *Bearing Type*.

Flat Btm. – This field is disabled when *Beam by Others* is selected as the *Bearing Type*.

Plies – This field defines the number of plies of material to be used in the construction of a '*Beam by Others*.'

Ply Width – The *Ply Width* field defines the thickness of a single ply of the material that will be used to construct a '*Beam by Others*.'

Mbr Depth – The *Member Depth* field is used to specify the depth dimension of the '*Beam by Others*' being defined. (See the figure at right.)



Member Depth

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your '*Beam by Others*.'

Location – The *Location* radio buttons are disabled when *Beam by Others* is selected as the *Bearing Type*.

Label – This field is disabled when *Girder by Others* is selected as the *Bearing Type*.

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Beam by Others* that is created. The user will not be prompted to define the properties again for additional beams by others.

Field-Framed – When the *Field-Framed* checkbox is activated, the beam being defined will not appear on individual **panel plots** and will not be included in the materials lists for those plots.

Define Posts –

Floor – When the *Floor* checkbox is activated, the '*Beam by Others*' being defined is designated as supporting floor-framing members.

Non-Load Bearing – This field is disabled when *Beam by Others* is selected as the *Bearing Type*.

Bears on Earth – This field is disabled when *Beam by Others* is selected as the *Bearing Type*.

A screenshot of the 'Set Bearing Properties' dialog box. The 'Bearing Type' list on the left has 'Beam by Others' selected. The 'Reference Line' section has radio buttons for 'Left', 'Center', and 'Right', with 'Right' selected. The 'Beg. Height' and 'End Height' fields are disabled. The '# Plies' field is set to 1, 'Ply Width' is 3-8, and 'Mbr Depth' is 12-0. The 'Product' field is 'No Product Assigned'. The 'Location' section has radio buttons for 'Interior' and 'Exterior', with 'Exterior' selected. There are checkboxes for 'Continuous', 'Field-Framed', 'Define Posts', 'Floor', 'Non-load bearing', and 'Bears on Earth'. At the bottom are buttons for 'OK', 'Delete', 'Reserved', and 'Products'.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Reserved – This command button is disabled when *Beam by Others* is selected as the *Bearing Type*.

Products – This command button is disabled when *Beam by Others* is selected as the *Bearing Type*.

Set Bearing Properties for a Flush Wall

Bearing Type – Flush Wall

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field defines the height of your new flush wall at its starting point.

Flat Btm. –

End Height – This field defines the height of your new flush wall at its end point.

Flat Btm. –

Plies – This field is disabled when *Flush Wall* is selected as the *Bearing Type*.

Ply Width – The *Ply Width* field defines the thickness of a single ply of the material that will be used to construct your flush wall.

Mbr Depth – The *Member Depth* field is used to specify the depth dimension of the flush wall being defined. (See the figure at right.)

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your flush wall.

Location – The *Location* field designates your flush wall as existing on the *Interior* or *Exterior* of the structure. This classification effects the framing rules that will be applied to the flush wall's construction.

Label – This field is disabled when *Flush Wall* is selected as the *Bearing Type*.

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Flush Wall* that is created. The user will not be prompted to define the properties again for additional flush walls.



Member Depth

Field-Framed – This field is disabled when *Flush Wall* is selected as the *Bearing Type*.

Floor – When the *Floor* checkbox is activated, the flush wall being defined is designated as supporting floor-framing members.

Non-Load Bearing – This field is disabled when *Flush Wall* is selected as the *Bearing Type*.

Bears on Earth – When this checkbox is activated, the *Flush Wall* (whose properties are being defined within this dialog box) serves as the ultimate and final destination for all loads applied to it. No loads are transferred to bearings beneath this wall.

Wall Type – As the name implies, the *Wall Type* field identifies the kind of flush wall being defined. There are three options: *Framed*, *Foundation*, and *Non-Framed*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Reserved – This command button is disabled when *Flush Wall* is selected as the *Bearing Type*.

Products – This command button is disabled when *Flush Wall* is selected as the *Bearing Type*.

Set Bearing Properties for a Cut Joist

Bearing Type – *Cut Joist*

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field is disabled when *Cut Joist* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Cut Joist* is selected as the *Bearing Type*.

End Height – This field is disabled when *Cut Joist* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Cut Joist* is selected as the *Bearing Type*.

Plies – This field is disabled when *Cut Joist* is selected as the *Bearing Type*.

Ply Width – This field is disabled when *Cut Joist* is selected as the *Bearing Type*.

Mbr Depth – This field is disabled when *Cut Joist* is selected as the *Bearing Type*.

The screenshot shows the 'Set Bearing Properties' dialog box. The 'Bearing Type' list box has 'Cut Joist' selected. The 'Reference Line' section has 'Right' selected. The 'Beg. Height' and 'End Height' fields are both set to '0-0-0'. The '# Plies' field is set to '1'. The 'Ply Width' field is set to '0-0'. The 'Mbr Depth' field is set to '0-0'. The 'Product' field is set to 'No Product Assigned'. The 'Location' section has 'Exterior' selected. The 'Label' field is empty. The 'Continuous' and 'Field-Framed' checkboxes are unchecked. The 'Floor', 'Non-load bearing', and 'Bears on Earth' checkboxes are also unchecked. The 'OK', 'Delete', 'Reserved', and 'Products' buttons are visible at the bottom.

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your cut joist.

Location – The *Location* radio buttons are disabled when *Cut Joist* is selected as the *Bearing Type*.

Label – This field is disabled when *Cut Joist* is selected as the *Bearing Type*.

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Cut Joist* that is created. The user will not be prompted to define the properties again for additional cut joists.

Field-Framed – This field is disabled when *Cut Joist* is selected as the *Bearing Type*.

Floor – When the *Floor* checkbox is activated, the cut joist being defined is designated as supporting floor-framing members.

Non-Load Bearing – This field is disabled when *Cut Joist* is selected as the *Bearing Type*.

Bears on Earth – This field is disabled when *Cut Joist* is selected as the *Bearing Type*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Reserved – This command button is disabled when *Cut Joist* is selected as the *Bearing Type*.

Products – This command button is disabled when *Cut Joist* is selected as the *Bearing Type*.

Set Bearing Properties for a Ridge Board

Bearing Type – *Ridge Board*

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field is disabled when *Ridge Board* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Ridge Board* is selected as the *Bearing Type*.

End Height – This field is disabled when *Ridge Board* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Ridge Board* is selected as the *Bearing Type*.

The screenshot shows the 'Set Bearing Properties' dialog box. The 'Bearing Type' list box has 'Ridge Board' selected. The 'Reference Line' section has 'Right' selected. The 'Beg. Height' and 'End Height' fields are both set to '0-0-0', and the 'Flat Btm.' buttons are disabled. The '# Plies' field is '1', 'Ply Width' is '3-8', and 'Mbr Depth' is '12-0'. The 'Product' field is 'No Product Assigned'. The 'Location' section has 'Exterior' selected. The 'Label' field is empty. The 'Continuous' and 'Field-Framed' checkboxes are unchecked. The 'Floor', 'Non-load bearing', and 'Bears on Earth' checkboxes are also unchecked. The 'OK', 'Delete', 'Reserved', and 'Products' buttons are visible at the bottom.

Plies – This field defines the number of plies of material to be used in the construction of your ridge board.

Ply Width – The *Ply Width* field defines the thickness of a single ply of the material that will be used to construct your ridge board.

Mbr Depth – The *Member Depth* field is used to specify the depth dimension of the ridge board being defined. (See the figure at right.)



Member Depth

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your ridge board.

Location – The *Location* field designates your ridge board as existing on the *Interior* or *Exterior* of the structure. This classification effects the framing rules that will be applied to the ridge board's construction.

Label – This field is disabled when *Flush Wall* is selected as the *Bearing Type*.

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Ridge Board* that is created. The user will not be prompted to define the properties again for additional ridge boards.

Field-Framed – This field is disabled when *Ridge Board* is selected as the *Bearing Type*.

Floor – This field is disabled when *Ridge Board* is selected as the *Bearing Type*.

Non-Load Bearing – This field is disabled when *Ridge Board* is selected as the *Bearing Type*.

Bears on Earth – This field is disabled when *Ridge Board* is selected as the *Bearing Type*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Reserved – This command button is disabled when *Ridge Board* is selected as the *Bearing Type*.

Products – Clicking on the *Products* command button will open a *PST Product Priority* dialog box, which can be used to add to, or modify, the list of materials available for use in the construction of your ridge board. For more information about the features and functions of the *PST Product Priority* dialog box, see the section within this document entitled “*Defining Product Priorities*.”

Set Bearing Properties for a Ledger

Bearing Type – *Ledger*

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center of the Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field is disabled when *Ledger* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Ledger* is selected as the *Bearing Type*.

End Height – This field is disabled when *Ledger* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Ledger* is selected as the *Bearing Type*.

Plies – This field defines the number of plies of material to be used in the construction of your ledger.

Ply Width – The *Ply Width* field defines the thickness of a single ply of the material that will be used to construct your ledger.

Mbr Depth – The *Member Depth* field is used to specify the depth dimension of the ledger being defined. (See the figure at right.)



Member Depth

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your ledger.

Location – This field is disabled when *Ledger* is selected as the *Bearing Type*.

Label – This field is disabled when *Ledger* is selected as the *Bearing Type*.

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Ledger* that is created. The user will not be prompted to define the properties again for additional ledgers.

Field-Framed – This field is disabled when *Ledger* is selected as the *Bearing Type*.

Floor – When the *Floor* checkbox is activated, the ledger being defined is designated as supporting floor-framing members.

Non-Load Bearing – This field is disabled when *Ledger* is selected as the *Bearing Type*.

Bears on Earth – This field is disabled when *Ledger* is selected as the *Bearing Type*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Reserved – This command button is disabled when *Ledger* is selected as the *Bearing Type*.

Products – Clicking on the *Products* command button will open a *PST Product Priority* dialog box, which can be used to add to, or modify, the list of materials available for use in the construction of your ledger. For more information about the features and functions of the *PST Product Priority* dialog box, see the section within this document entitled “*Defining Product Priorities*.”

Set Bearing Properties for a Strut

Bearing Type – Strut

Reference Line – All objects drawn in GS Plan are placed in relation to a *Reference Line*. Objects can be rendered on *Center* of the *Reference Line*, or to its *Right* or *Left*.

Beg. Height – This field is disabled when *Strut* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Strut* is selected as the *Bearing Type*.

End Height – This field is disabled when *Strut* is selected as the *Bearing Type*.

Flat Btm. – This command button is disabled when *Strut* is selected as the *Bearing Type*.

Plies – This field defines the number of plies of material to be used in the construction of your strut.

Ply Width – The *Ply Width* field defines the thickness of a single ply of the material that will be used to construct your strut.

Mbr Depth – The *Member Depth* field is used to specify the depth dimension of the strut being defined. (See the figure at right.)

Product – This is a read-only field that displays the specific material, if any, that has been designated for use in the construction of your strut.

Location – This field is disabled when *Strut* is selected as the *Bearing Type*.

Label – This field is disabled when *Strut* is selected as the *Bearing Type*.

Continuous – When the *Continuous* checkbox is activated, GS Plan will use the properties within this dialog box, as currently defined, for each subsequent *Strut* that is created. The user will not be prompted to define the properties again for additional struts.

Field-Framed – This field is disabled when *Strut* is selected as the *Bearing Type*.

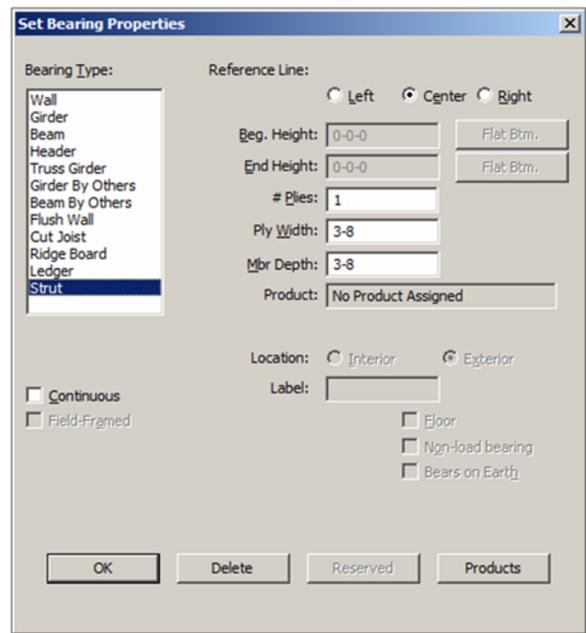
Floor – This field is disabled when *Strut* is selected as the *Bearing Type*.

Non-Load Bearing – This field is disabled when *Strut* is selected as the *Bearing Type*.

Bears on Earth – This field is disabled when *Strut* is selected as the *Bearing Type*.

OK – If you are creating a new bearing, clicking on the *OK* button will place the new bearing in accord with the properties defined in this dialog box. If you are modifying the properties of an existing bearing, clicking on the *OK* button will apply your changes and close the *Set Bearing Properties* dialog box.

Delete/Cancel – If you are creating a new bearing, clicking on the *Delete* button will close the *Set Bearing Properties* dialog box and return you to the main window without placing your bearing. If you



Member Depth

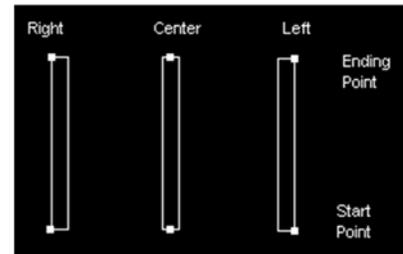
are modifying the properties of an existing bearing, this button will read *Cancel* rather than *Delete*. Clicking on the *Cancel* button will discard your changes and close the *Set Bearing Properties* dialog box.

Reserved – This command button is disabled when *Strut* is selected as the *Bearing Type*.

Products – Clicking on the *Products* command button will open a *PST Product Priority* dialog box, which can be used to add to, or modify, the list of materials available for use in the construction of your strut. For more information about the features and functions of the *PST Product Priority* dialog box, see the section within this document entitled “*Defining Product Priorities*.”

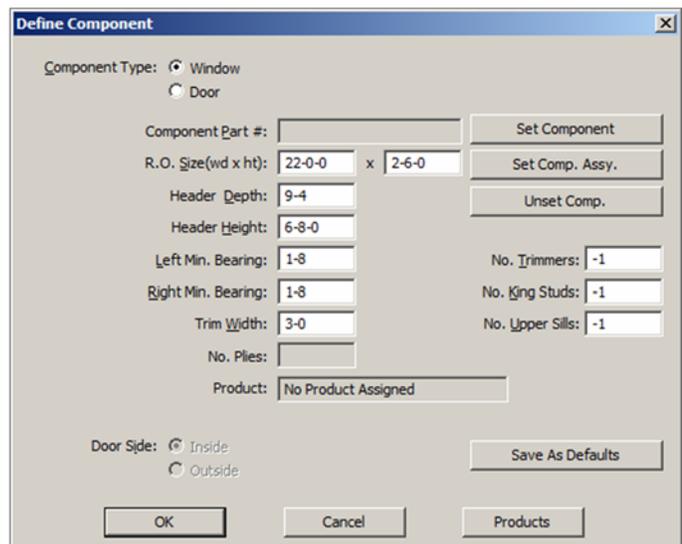
- **Continuous** – When drawing certain bearings (for example exterior walls), members with the same properties are placed one after another. When the *Continuous* checkbox is activated, all bearings being drawn will have the same properties. This means that you will not be prompted to define the bearing properties each time a new bearing member is input. They will receive the same properties as the previous bearing, until the *Bearing* command is exited (by picking another entity to input) or restarted (by selecting *Bearings* from the *Entities* menu, *Classic Sidebar*, or *Standard* toolbar).

- **Reference Line** – This is the side of a drawn line on which you want the body of an entity to fall. This is determined based on the start and ending points. *Center* keeps the *Reference Line* in the middle of the bearing and half of the bearing falls to either side. In the examples at right, a bearing of each orientation is displayed, with the start point at the bottom and the ending point at the top.



- **Non-Load Bearing** – If you are defining a partition wall, or any other wall that will not support loads transferred from above, the *Non-Load Bearing* checkbox should be activated.
- **Wall Type** – ...the list of potential values includes:
 - ◆ *Framed* –
 - ◆ *Foundation* –
 - ◆ *Non-Framed* – If you are defining a wall, and it is going to be built any way other than conventional framing, the *Non-Framed* flag should be set.

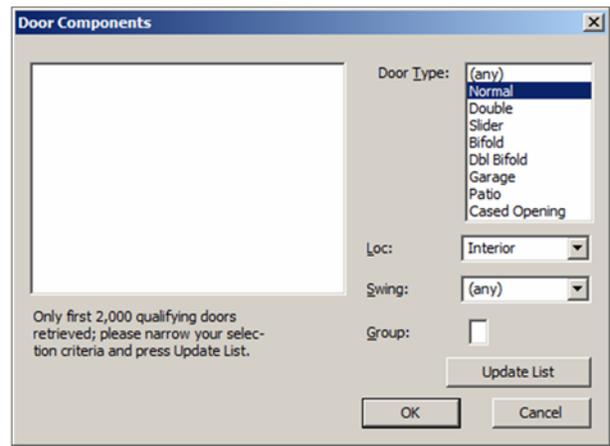
- **Header Info** – Clicking on the *Header Info* command button will open a *Define Component* dialog box, like the one pictured at right. For more information about the features and functions associated with this dialog box, see “*Defining Component Properties*” on page ____.



- Following is an explanation of the fields and buttons available in the *Define Component* dialog box:

- ◆ **Component Type** – Select the type of component you wish to use; *Door* or *Window*.

- ◆ **Component Part #** – This is the name of the component that you are currently viewing. To change, or choose a component, click the *Set Component* button. When you click on the *Set Component* button a *Window Components* or a *Door Components* dialog box will open (depending upon whether you have selected *Door* or *Window* as your *Component Type* in the *Define Component* dialog box). For more information on these component dialog boxes, see the section within this document entitled “*Selecting Door & Window Components*.”



- ◆ Note: It is not necessary to define a component. You may, instead, input an opening based on rough-opening dimensions only.

- ◆ **R.O. Size (Wd x Ht)** – This is the exact size of the opening to be placed in the wall. Enter the width of the opening in the first field, and the height in the second.

- ◆ **Header Depth** – This is the exact depth of the material that will be used as the header. This value is entered in inches and sixteenths of an inch.

- ◆ **Header Height** – This is the distance from the bottom of the wall to the top edge of the opening.

- ◆ **Left Min. Bearing** – This is the total width of any members that will be supporting the left side of the header. This value is entered in inches and sixteenths of an inch.

- ◆ **Right Min. Bearing** – This is the total width of any members that will be supporting the right side of the header. This value is entered in inches and sixteenths of an inch.

- ◆ **Trim Width** – This is the closest that either end of this opening can get to another opening or an intersection wall. This value is entered in inches and sixteenths of an inch.

- ◆ **No. Plies** – This field is used to define the number of plies used to construct the designated bearing.

- ◆ **Product** –

- ◆ **No. Trimmers** –

- ◆ **No. King Studs** –

◆ **No. Upper Sills –**

- ◆ **Door Side** – This is the direction that the door should swing. *Inside* and *Outside* refer to the room that the door is placed into.

COMMAND BUTTONS

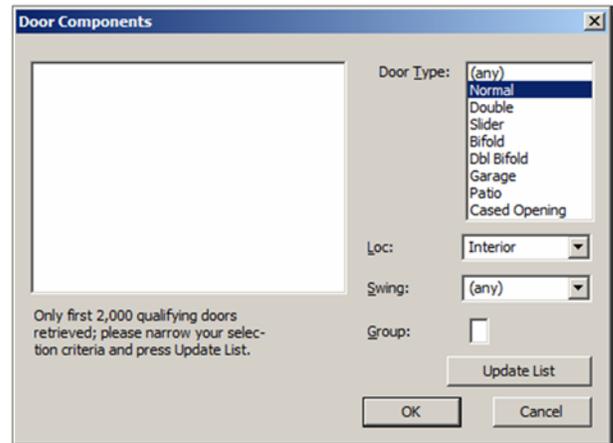
- ◆ **Set Component** – Click this button to open a dialog box that will allow you to select a component from the list of doors and windows you have created in KeyMan. For more information about the *Door Components* dialog box and the *Window Components* dialog box, see the section within this document entitled “*Selecting Door & Window Components.*”

◆ **Set Comp. Assy. –**

- ◆ **Unset Comp.** – When a component is selected, the rough opening size is pre-determined and therefore those fields are grayed out. If you wish to change this door or window from having a component to having a rough opening, click this button. The *Component Part #* field will be cleared and the rough opening fields will become available.
- ◆ **Save As Defaults** – Click the Save As Defaults button to store the information in this window and bring it up every time a new header is being defined.
- ◆ **OK** – Click the OK button to apply the information entered in this window and return to the Main Model™ window.
- ◆ **Cancel** – Click the Cancel button to return to the Main Model™ window without making any changes to the selected header or header being placed.
- ◆ **Products** – Clicking on the *Products* button opens a *View/Edit Product List* window like the one pictured at right. This window allows you to specify what materials you want to use for this header.

Selecting Door & Window Components

When you first enter this screen, all door or window components are displayed (depending on the component type you chose on the previous screen). To narrow the list, select the component type in the column on the right and/or other options such as *Swing* or *Location*. Then click on the *Update List* button to filter the component list based on the selected options. Highlight the component you wish to place and click on the *OK* button. If no components appear in the list, make sure that the KeyMan path is set correctly on the *File, Set Paths* screen.



APPENDIX C: GS PLAN MENUS & TOOLBARS

MENU	Icon	SUB-MENU(S)	Hot-Key	TOOLBAR
FILE		New		File
		Open...		File
		Save	Ctrl+S	File
		Save As...		
		Options Extract As...		
		Export DXF File...		
		Merge...		File
		Load Shadow Job...		File
		GS Build...		GS Build
		Print Drawing		File
		Plot Layout		File
		Print Trusses		File
		General Information		File
		Save Defaults		
		Reset Products		
		ReSync Job		File
		Set Paths		
		Auto Save & Undo Settings		
	Exit			
TOOLS		Point Snap	Ctrl+P	Tools
		Midpoint Snap	Ctrl+D	Tools
		Intersection Snap	Ctrl+I	Tools
		Perpendicular Snap	Ctrl+R	Tools
		XYZ Snap	Ctrl+E	Tools
		Grid Snap	Ctrl+G	Tools
		Ortho	Ctrl+O	Tools
		Set Snap Rules...		Tools
		Set Snap Grid...		Selection & Snap
		Set Visual Grid...		Selection & Snap
		Set Levels...		Tools; Standard
		Deselect All		Selection & Snap
		Select		Standard; Selection & Snap
		Select All Visible		Selection & Snap
		Enter Point...	Spacebar	Tools
		Set Top of Wall...	Ctrl+H	Tools
		Show Coordinates		Tools
		Show Messages		
		Show Structure Flags		
		Check Bearings		Tools
		Check Geometry		Tools
		Options		
		Create/Edit Option		Options
		Select Options		Options
		Copy/Xfer Option		Options
	Copy Selected Entities		Options	

MENU	Icon	SUB-MENU(S)	Hot-Key	TOOLBAR	
TOOLS (Cont'd)		Option Materials		Options	
		Toolbars			
		Classic Sidebar			
		Standard			
		File			
		Tools			
		Selection & Snap			
		Modify			
		Entities			
		Display			
		Macros			
		Headers			
		Roof Planes			
		Toggles			
		Lateral			
		Materials			
		Options			
		GS Wall			
		GS Wall Tools			
		Design			
		Structure Commands			
		Annotations			
		GS Build			
		View Schemes			
		Labeling			
		Output			
			Design Toolbar		Display
			T.O.P. Toolbar		
		Custom Toolbars			
MODIFY		Set Line Length...	l	Modify	
		Set Line Angle...		Modify	
		Move Endpoint	m	Modify	
		Split Line...	c	Modify	
		Join Lines	j	Modify	
		Trim/Extend Lines	t	Modify	
		Trim/Extend Two Lines	x	Modify	
		Offset Line/Array...	o	Modify	
		Reverse Reference Line	r	Modify	
		Make Polygon Edges...		Modify	
		Set Properties		Modify; Standard	
		Undo	Ctrl+Z	Modify	
		Redo	Ctrl+Y	Modify	
		Shift/Copy Selected...	s	Modify	
		Rotate Selected...		Modify	
		Mirror Selected...		Modify	

MENU	Icon	SUB-MENU(S)	Hot-Key	TOOLBAR	
MODIFY (Cont'd)		Delete Selected	Del		
		Change Level...		Modify	
		Shift/Copy Model..			
		Rotate Model [Temporary]...		Modify	
		Set Properties Mode		Modify	
		Relap Corners		GS Wall Tools	
		Relap Corners to Default		GS Wall Tools	
ENTITIES		Construct Lines		Entities	
		Bearings		Entities; Standard	
		Areas		Entities; Standard	
		Openings		Entities	
		Obstructions		Entities	
		Planes		Entities; Standard	
		Loads		Entities	
		Overhangs		Entities	
		Single Truss		Entities	
		Hip Ends		Entities	
		Annotations Plot		Entities	
		Additional Framing		Entities	
		Hangers		Entities	
		Joist Holes		Entities	
		Annotations			
		Model Entity Labels	Ctrl+L	Annotations	
		Draw Dimension Lines	Ctrl+J	Annotations	
		Draw Arbitrary Lines	Ctrl+Q	Annotations	
		Draw or Edit Text Box	Ctrl+T	Annotations	
		Edit Annotation Attributes...	Ctrl+O	Annotations	
		Options...			
		Point-to-Point Dimension Lines	Ctrl+1	Annotations	
		Horizontal Dimension Lines	Ctrl+2	Annotations	
		Vertical Dimension Lines	Ctrl+3	Annotations	
		Auto Dimensions Filter	Ctrl+F	Annotations	
	DISPLAY		Zoom Window		Display; Standard
			Zoom Full		Display
		Zoom Last		Display	
		Rotate...			
		Plan View		Display; Standard	
		Animate		Display; Standard	
		Solid Model		Display; Standard	
		Design Toolbar		Display	
		Show/Edit Truss Profiles		Display; Standard	
		Show Wall Profiles		Display	
		Show Planes & Overhangs		Display	
		Show Member Width			
		Show Panel Framing			

MENU	Icon	SUB-MENU(S)	Hot-Key	TOOLBAR
DISPLAY (Cont'd)		Force Panel <u>U</u> pdate		
		Set <u>C</u> olors...		
		Set <u>V</u> isibility...		Display; Standard
		Set View Scheme		
		View Scheme Settings		
MACROS		<u>G</u> enerate Members		Macros
		Re-Profile Members		Macros
		<u>U</u> ndo All Top Chord Edits		
		Undo All <u>B</u> ottom Chord Edits		
		Re-Elevate Roof Members		
		H eaders		
		<u>H</u> ader Products...		Headers
		<u>Q</u> uick Header...		Headers
		<u>M</u> ove Header		Headers
		<u>S</u> nap Header...		Headers
		<u>V</u> erify Components		Headers
		<u>G</u> lobal Change Components..		Headers
		<u>D</u> elete Header...		Headers
		Move Snap Header		Headers
		<u>E</u> xtrude Bearings		Macros
		Create <u>V</u> eneer Wall		Macros
		R oof Planes		
		<u>C</u> reate Planes...		Roof Planes
		Extend <u>M</u> anual		Roof Planes; Standard
		Extend <u>S</u> ingle		Roof Planes
		Extend <u>A</u> ll		Roof Planes
		Clip <u>O</u> ff Corner		Roof Planes
		<u>F</u> ive-Sided Bay Window		Roof Planes
		<u>T</u> hree-Sided Bay Window		Roof Planes
		<u>C</u> eiling Planes...		Macros
		T oggle		
		Toggle <u>L</u> oad Bearing		Toggles
		Toggle <u>R</u> oof/Floor		Toggles
		Toggle <u>I</u> nterior/Exterior		Toggles
		Toggle <u>W</u> all/Header Widths		Toggles
		Toggle <u>Q</u> verhang Lengths		Toggles
		Toggle <u>F</u> ramed/Not Framed		Toggles
		Toggle Top <u>T</u> rack Elevations		Toggles
		Toggle Wall/Header <u>H</u> eights		Toggles
		Toggle <u>B</u> earing Properties		Toggles
		Toggle Bears on <u>E</u> arth		Toggles
		S tick Framing		
		Ridge <u>B</u> oards (Roof)		
		<u>S</u> truts (Roof)		
		Stick <u>A</u> reas (Roof)		

MENU	Icon	SUB-MENU(S)	Hot-Key	TOOLBAR
MACROS (Cont'd)		Ridge Boards (Ceiling)		
		Stick Areas (Ceiling)		
		Skylight...		Macros
		Dormer...		Macros
		Panel Break Mode		Macros
		Floor Deck Edit Mode		Macros
		Stretch Model		Macros
LATERAL		Default Settings...		
		Insert SIP...		Lateral
		Move SIP		Lateral
		Delete SIP		Lateral
		Create Shear Lines		Lateral
		Erase Shear Lines		Lateral
		Wind Loads		
MATERIALS		Assign Non-Structural Materials		Materials
		Generate Non-Structural Materials List		Materials
		Generate Structural Materials List		Materials
		Output Window/Door Schedule		Materials
		Output Floors to GSS		Materials
		GSS Floor Output Settings...		
		Option Materials		Materials
		Structural Options Materials List		
	Output Members to LOC File			
DESIGN		Check Model		
		Edit Trusses		
		Reconsolidate All Trusses		
		Truss Label Presets		
		Import Hangers		
GS BUILD		GS Plan		
		GS Beam		
		GS Truss		
		GSS Output		
HELP		Contents		File
		What's New		
		Visit Website		
		About GS Plan		File