



RAM Advanse...

Masonry Tutorial

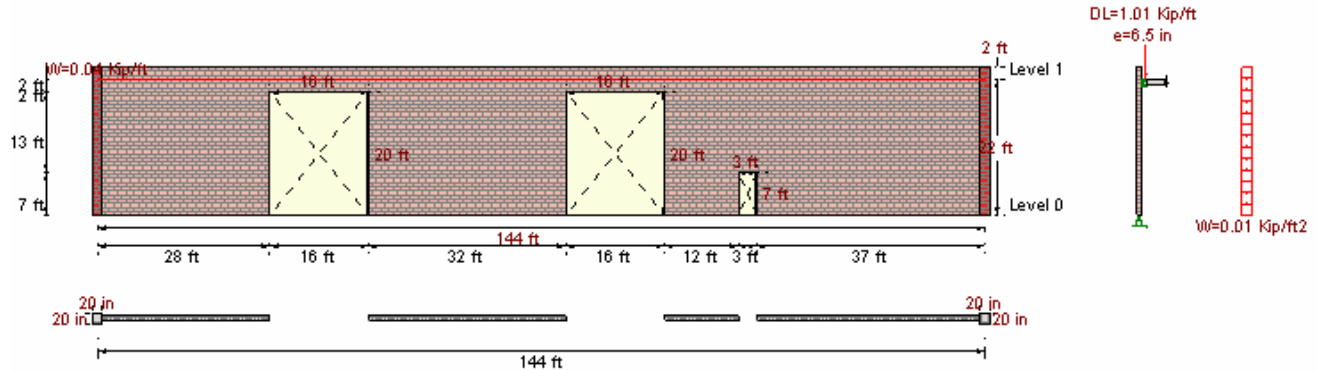
Disclaimer:

This file is a PDF version of the embedded masonry tutorial found in the help section of RAM Advanse and is subject to all licensing agreements, copyrights and other protections associated with the software. It is reproduced here as a convenience to the user. The software, including this tutorial, was developed for use as a design aid for qualified engineers. Considerable care was used in its preparation, but it should not be used in the absence of sound engineering judgment and knowledge. The engineer of record is always responsible for data used in design and results obtained whether from a calculator or from computer software. There are no warranties, either express or implied, associated with this software by the IMI, NCMA, Bentley Systems, Ryan-Biggs Associates, or distributors. The user must explicitly understand the assumptions inherent in the software, the procedures for structural design and must independently verify any results. Bentley Systems, NCMA, IMI and Ryan Biggs Associates do not assume responsibility for errors or omissions resulting from the use of the material in this document or in its use when preparing plans or specifications.

Example 8: Masonry Wall

This example shows systematically the creation of a masonry wall. This example will be most effective if the user practices the illustrated skills as they are presented.

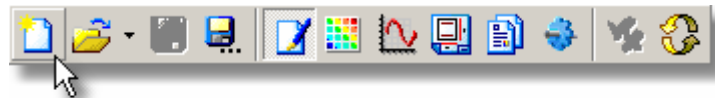
The structure is an example of one-story industrial building of reinforcement masonry walls; it includes the design of bearing walls, shear walls and lintels. It is an example presented in “Amrhein J. 1983, Reinforced Masonry Engineering Handbook, Fourth edition, Section 10”.



Example of masonry wall

Starting a new structure

If the Masonry Wall module is not already open, choose *Modules – Masonry Wall* from within RAM Advanse. To start a new structure click in the bottom “Create a new Masonry Wall” and a default wall will appear. If an existing model is open, the module will ask to save it.

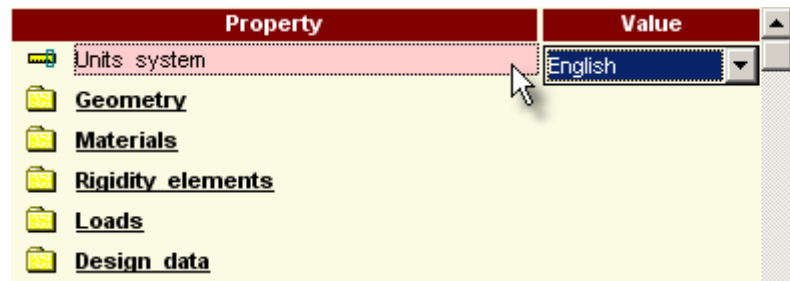


Select the button to create a new Masonry Wall.

Once the new file is open, proceed to the entering of data in the left window, following the order that is shown below.

Entering units

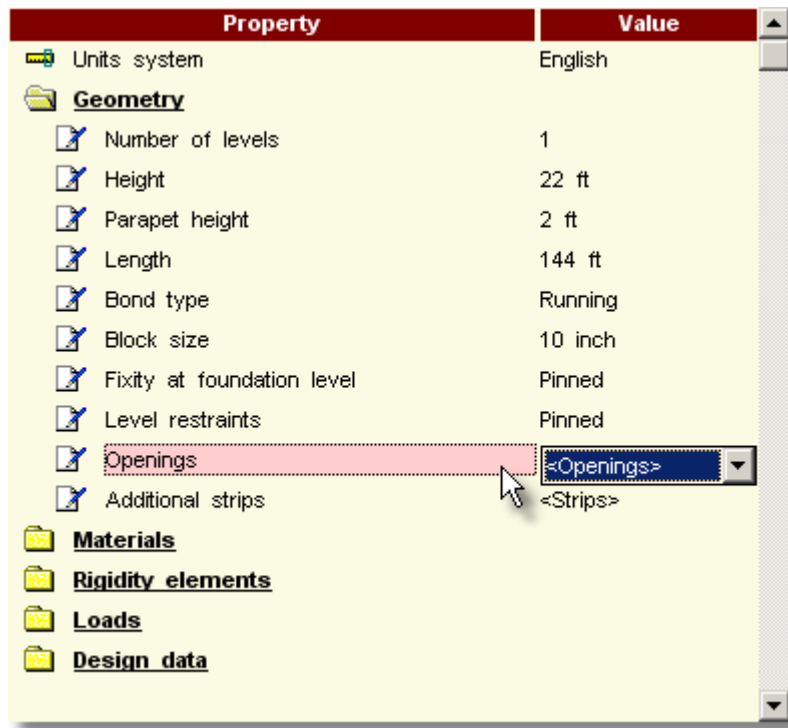
Select the option *Units system*. This action will allow the drop-down menu to be enabled.



Select the *English* units system.

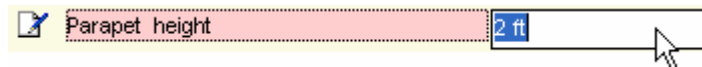
Entering geometry data

Then go to the option **Geometry**. This and the other files will drop-down for user's comfort.

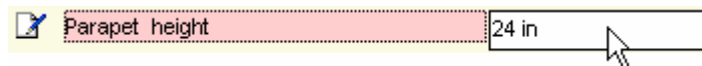


Enter wall dimensions as shown in the figure above.

Note - All entered values will correspond to the default units. If other units of the same system are required, type the value followed by the desired unit, as shown below.



Click in the cell to highlight the value.

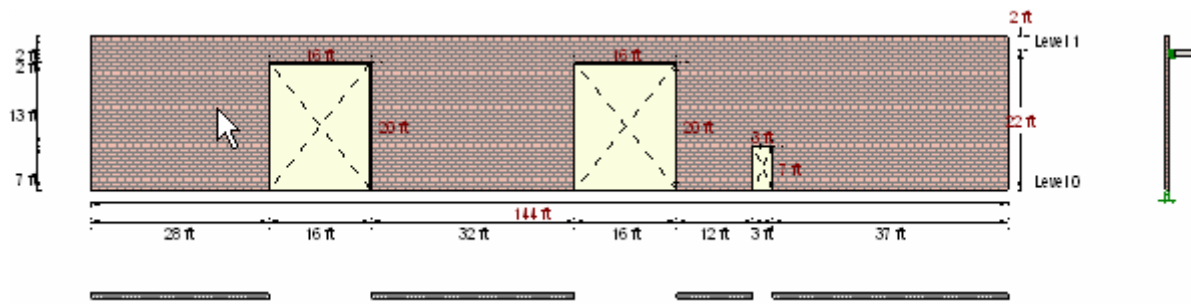


Type the value followed by its unit and press Enter.

To enter the **Openings**, choose this option and a spreadsheet will appear to define one or several openings in the wall at the same time.


Nro.	Referente Corner	X offset	Y offset	B: Width	H: Height
1	Lower Left	28	0	16	20
2	Lower Left	104	0	3	7
3	Lower Left	76	0	16	20

Note - Remember the bottom  can be used to undo changes.



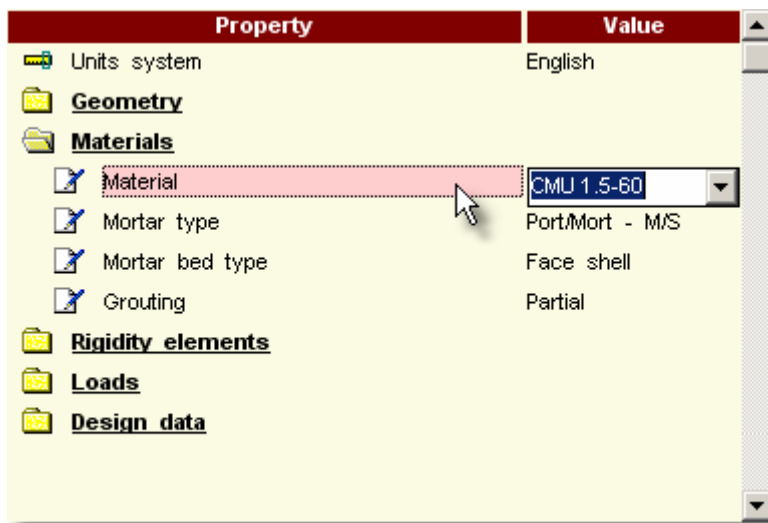
Now, the masonry wall has three different openings.

Note: Those values that appear in red can be modified directly in the plot.

The openings in this example have different shapes and are in the same level. In case the user needs to enter several openings in many level, it is possible to use the bottom .

Entering materials

There are four variables to define in this option.

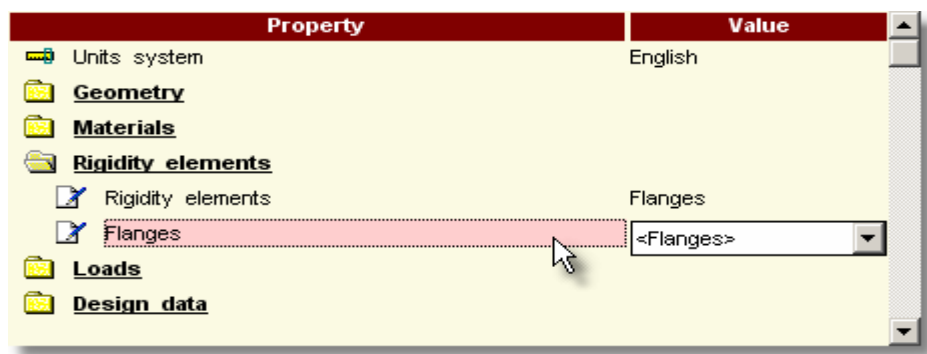


Enter the values shown in the figure above.

The first one, **Material**, defines the type and strength of the material. Choose *CMU 1.5-40*, which stands for brick masonry unit masonry with 1500 psi of specific compressive strength and grade 40 reinforcement. The second one is the **Mortar type** according to Table 2.2.3.2 on MSJC -02. The **Mortar bed type** can be complete or face shell, and the **grouting** can be complete or partial. If the partial grouting is chosen, only those cells that have reinforcement are going to be full.

Entering rigidity elements

Rigidity elements are commonly used to increase the stiffness of the masonry wall. The module allows choosing between None, Columns or flanges. For this examples choose flanges.



Select **Flanges** and enter the values that are shown in the table below.

Nro.	Distance	Width Z	Thicknes s	Position Z	Position X
1	0	55	9.63	Atrás	Derecha
2	144	55	9.63	Atrás	Izquierda

The flanges are defined by the **distance** from the left side of the wall, the flanges **width**, and the **position** along Z and X-direction. The flanges **width** is the flanges size along Z-direction, the user has to enter the real width and the module will consider only the effective width (6 times the flange thickness or the actual flange). The **position** along Z-direction defines if the flanges are at the back or front side from the wall face. The **position** along X-direction defines if the flanges are aligned at the left or right side from the point defined in **Distance**.


Entering loads

The module presents three different kinds of loads that may be applied to the wall, such as:

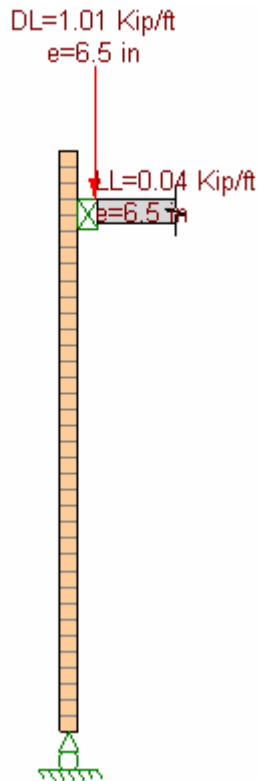
1. Vertical loads
2. In-plane lateral loads
3. Out-of-plane loads
4. Global forces

In this example, we have distributed dead and live vertical loads width eccentricity, in-plane lateral loads, and out-of-plane wind loads.

Before entering the loads, it is necessary to create the required load conditions. To do that, press the

button  to open the load condition manager, and create the Live Load (LL) and Wind (W) conditions as it is shown in the next figure.

Example 8: Masonry Wall



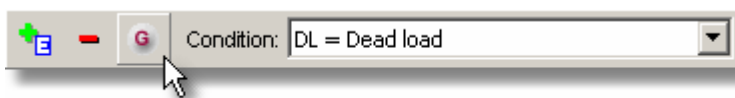
Distributed dead load defined.

Now, select **Lateral in-plane loads /Distributed**; introduce a distributed wind of 0.041 kip/ft and click OK.

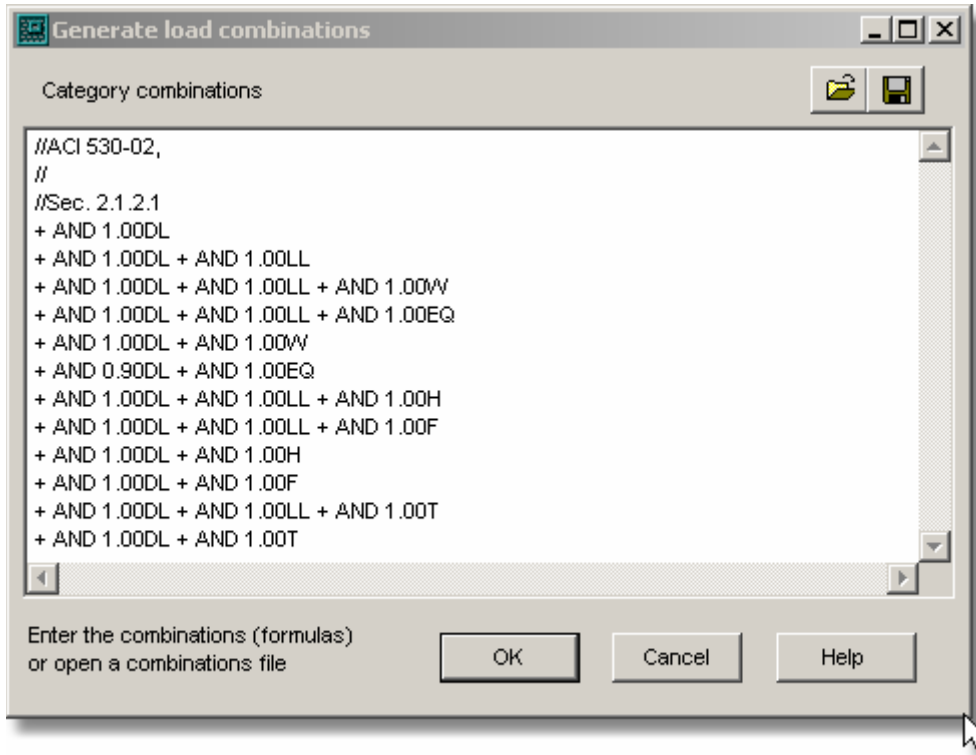
Enter the out-of-plane Out of plane load/pressure load of wind 0.012 kip/ft and click OK.


Generating load combinations

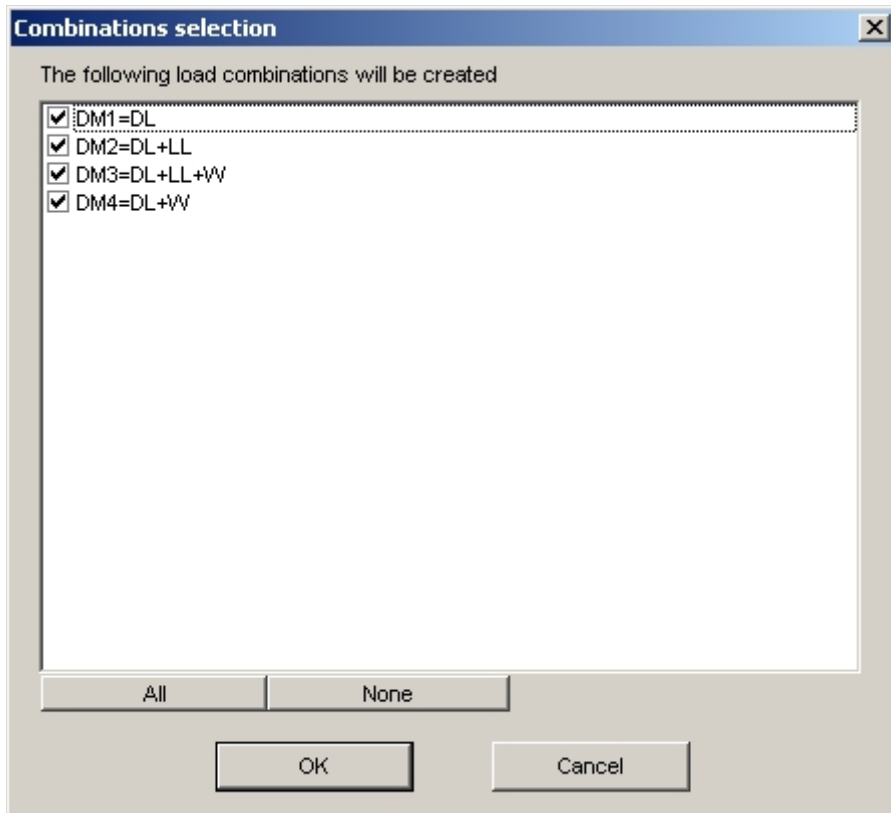
The program provides the different load combination files for each code (.mwg extension). The user will be able to generate these combinations automatically or define them manually.



*Click on the bottom **G** to generate load combinations.*

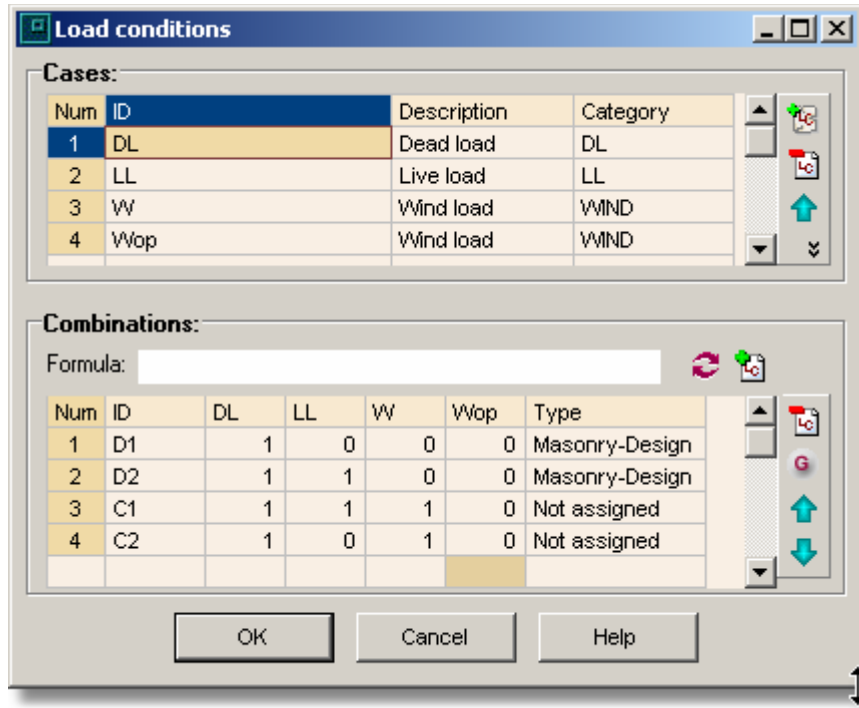


Press the button  to open a file and choose the file “ACI 530-02 LoadCombos_AllowableStressDesign.mwg”. Then press OK. And continue to choose the combinations.



Select the desired combinations and press OK.

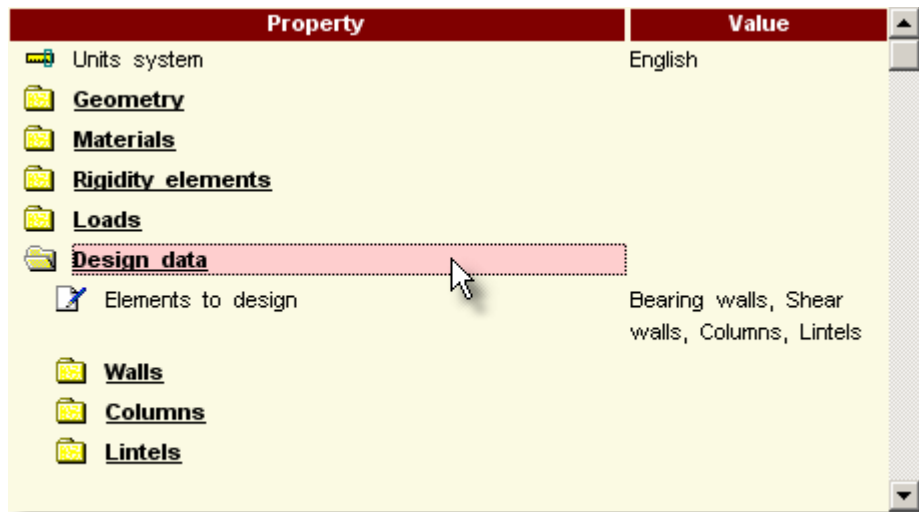
Finally, the Load conditions manager shows:



Generated load combinations

Entering design data

The first variable on the option *Design data* is *Elements to design*. This variable allows selecting which elements of the complete masonry wall are going to be designed.

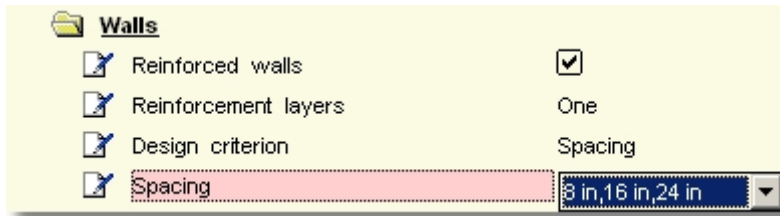


Select all the elements to design and press OK.

Although the option *Columns* was checked, the module is not going to design columns because there are no columns in the actual example. In the same way, the module identifies which elements the model should be designed.

The module has the option to design bearing and/or shear walls as reinforced or unreinforced. If the variable **reinforced wall** is unchecked the other variable disappears and the module only performs a validation of the wall under the applied loads. Only one reinforcement layer will be used.

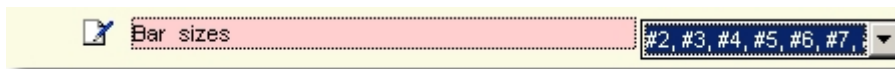
There are three criteria to design the walls: **Spacing, Bar size or Reinforced area**. Select **Spacing**. The spacing values are given by the spacing between block cells. Spacing of 8, 16 and 24 in will be considered.



Enter 16, 24 and 32 in as spacing.

Note: If **Spacing** is selected as **Design criterion**, the module will check every spacing value in a descendent order with the complete range of bar sizes until it finds an optimum result. If **Bar size** is selected as **Design criterion**, the module will check every selected bar size in an ascendant order with different spacing values until it finds an optimum result. Finally, if **Reinforced Area** is selected the module will find the optimum total reinforced area for each Bar size, and it will select the minimum as optimum.

It is also necessary to enter the Bar sizes for Columns and Lintel design.



Enter the Longitudinal Bar Sizes #2,3.....18 for the column design.

For columns design, the module will check an optimum result of tie bar size and tie spacing according to the reinforcement requirements of the design code. In the same way, the module will use the longitudinal bar sizes to find an optimum bar quantity.

For the lintel design, it is necessary to enter the lintel depth. In the case that the depth is the same for all the lintels, there is an option to select equal lintel depth. Besides, it is required to enter the longitudinal bar sizes.



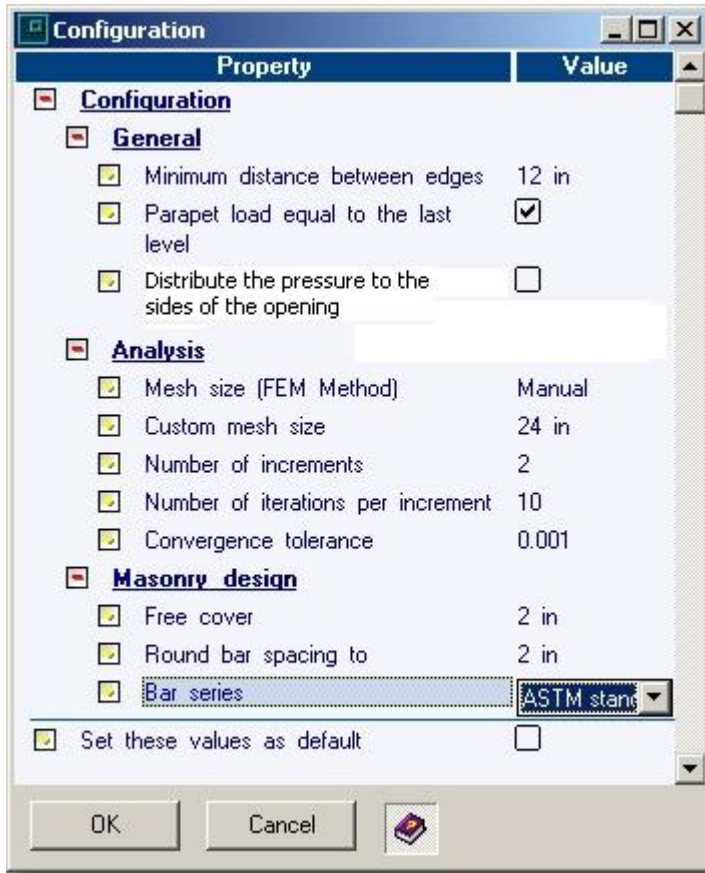
For the lintel design, enter the lintel depth and the list of longitudinal bar sizes.

Entering Configuration values

After entering all general data, the user should verify if all values by default of the **Configuration Screen** are correct for the model and the design requirements.



Press the button shown in the figure.

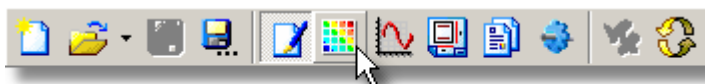


Verify that all values by default are correct for the model and the design requirements.





For this example, change the *Mesh size (FEM Method)* to *manual* and a new variable is going to appear, set 24 in for the *Custom mesh size* and press OK.

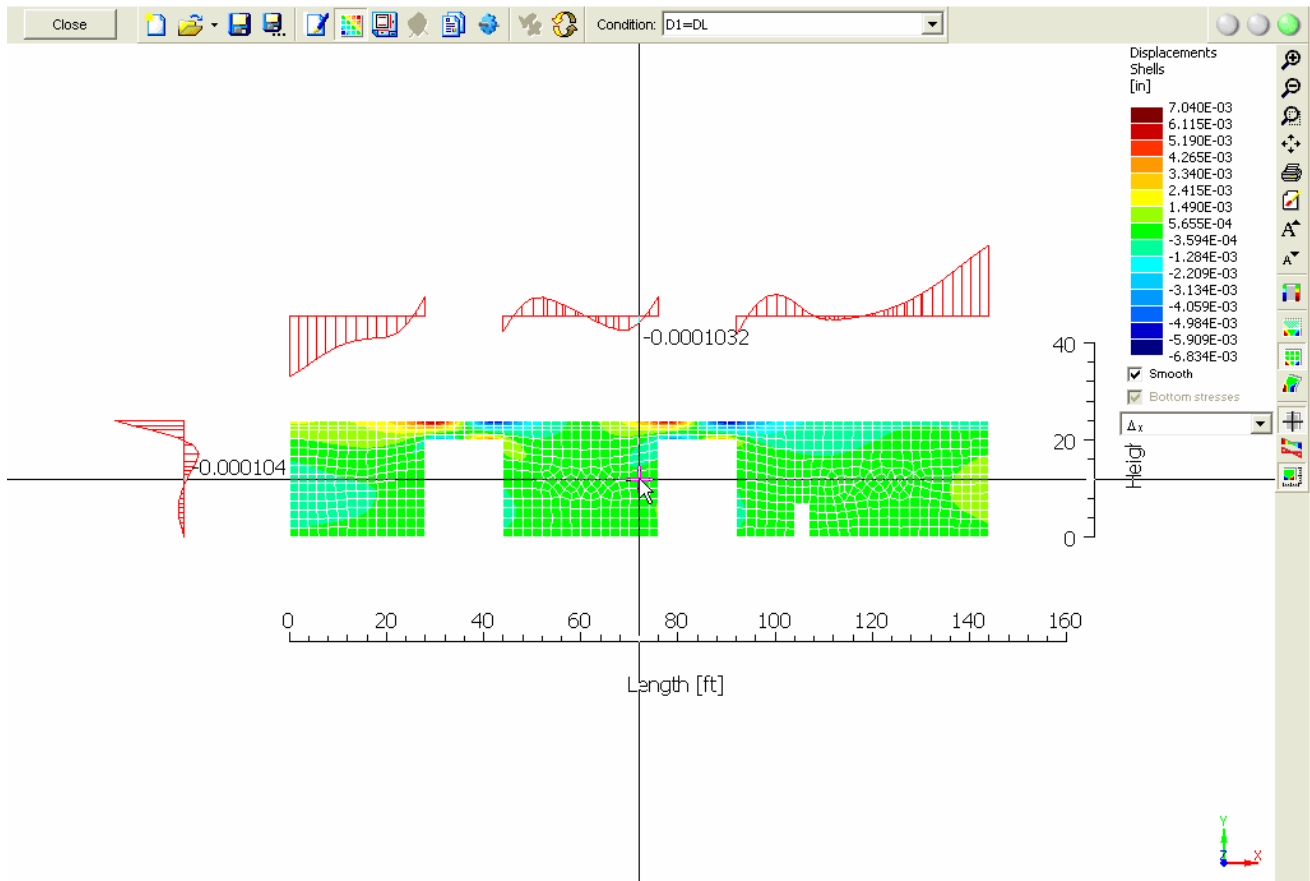
Seeing results graphically

Once all the data is entered, the module is ready for the analysis and design.



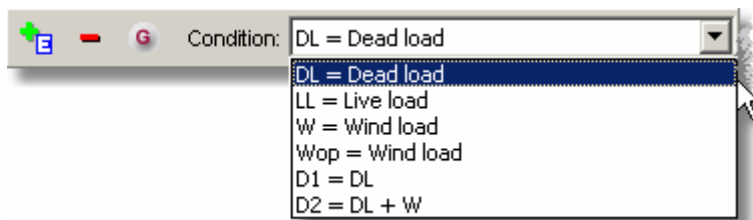
Click on the FEM diagram bottom  to see the analysis result.

Note: When one of the buttons *FEM diagram* , *Detailing* , *Report*  or *Optimize design*  is selected for the first time, the module analyze the model, perform a reinforcement optimization and verify the reinforcement according to the code; therefore it can take some minutes depending on the model size.



Analysis results obtained by the Finite Element Method (FEM)

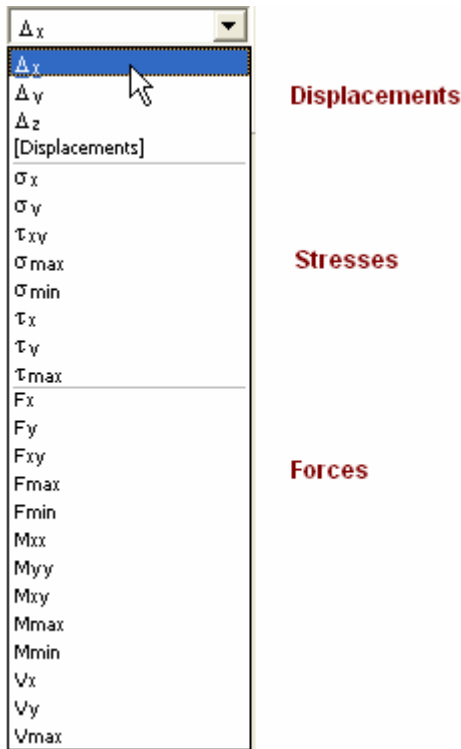
At the top side of the screen, the load **conditions** for these results are available; if another load condition is desired to see, dropdown the list and choose one:



Select the desired load condition.








At the right side several graph types can be chosen. The first four options show the wall displacements; the next eight options show the stresses; and rest show the resultant forces in the wall, remember that forces values are shown as force per length (e.g. kip/ft).

Example 8: Masonry Wall



Select the desired graph type

Note: Axial forces are given **Fy**; shear forces are given **Fxy**, out-of-plane moments are given by **Mxx** and in-plane moments are obtained using the **Fy** forces respect the middle of the wall portion in consideration.

At the right side of the screen, there are several options for the graphs, such as: see rigidity element graphs , visible levels , see mesh segmentation , see deformations , activate section axes , activate fill contour  and see the ruler .


Detailing the wall

Once the analysis and design of the wall have been run, the design can be reviewed in the detailing screen.


The screenshot displays a software window with a spreadsheet on the left and a technical drawing on the right. The spreadsheet is titled 'Vertical' and contains the following data:


Group	Strip	Quantity	Bar	Spacing	Level	Dist1	Dist2
1	Strip 1	5	#2	72	0	0	
2	Strip 1	5	#2	72	1	0	
3	Strip 2	3	#2	72	0	20	
4	Strip 2	3	#4	72	1	0	
5	Strip 3	6	#5	72	0	0	
6	Strip 3	6	#2	72	1	0	
7	Strip 4	3	#2	72	0	20	
8	Strip 4	3	#4	72	1	0	
9	Strip 5	2	#4	72	0	0	
10	Strip 5	2	#2	72	1	0	
11	Strip 6	1	#5	72	0	7	
12	Strip 6	1	#2	72	1	0	
13	Strip 7	7	#3	72	0	0	
14	Strip 7	7	#2	72	1	0	

The technical drawing on the right shows a cross-section of a masonry wall with various reinforcement strips and bars. Below the main drawing is a smaller, simplified diagram of the wall's reinforcement layout.


Press  to enter to the detailing screen.

This screen displays a spreadsheet with the reinforcement data of the design. During the design, the program makes an optimization of reinforcement, that is to say, it obtains the minimum steel area to satisfy the requirements according to the design parameters the user has entered in the data screen.

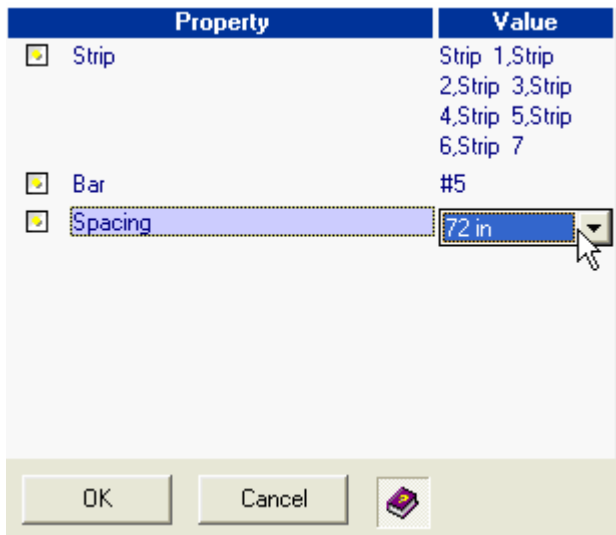
Remember that the user may change, edit or delete reinforcement at any moment according his requirements. After a change, choose the bottom  to verify quickly the influence of these changes.


The traffic light, at the upper right corner of the screen , is used to check the results of any changes.


This spreadsheet has four tabs: Wall **Vertical** reinforcement, Wall **Horizontal** reinforcement, **Lintel** reinforcement and **column** reinforcement. However the tab for columns is not shown because there is no column in this model.


The module has assigned different size of bars with 72 inches of spacing for vertical reinforcement. Let suppose the user needs spacing values of 72 inches with bar #5 for all vertical reinforcement. For this, initially delete all the vertical reinforcement using the bottom .

Example 8: Masonry Wall



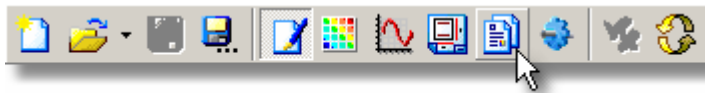
Click on the bottom  to enter a continuous reinforcement. Select all the strips, bar #5 and 72 inches of spacing, and then click OK.

A new vertical reinforcement was assigned to the whole wall and the traffic light is disabled due to these changes. Now the user can verify if those changes fulfill the requirements and the limitations of the code by clicking on the bottom . In the same way, it is possible to modify any reinforcement in the different tabs; just make any change and click on verify bottom. If the traffic light is not green, this means that some of the changes were not correct according to the code.

Note - If the user wants to return to the initial results of automatic design, the *Optimize design*  button can be used.

Seeing the report

The entire data and result sets can be seen in the report:



Press the button shown in the figure.

The report is divided in 5 parts: general information, bearing wall design, shear wall design, column design and lintel design.

- **General information.** This part of the report shows the wall geometry, material, openings, rigidity elements, load conditions, and loads.

Close [Icons] Help [] Show title block on each page

BENTLEY RAB

Units system: English
 File name: C:\Program Files\Ram\RAM Advanse\Data\Samples\MW\Example 9 - One-story industrial building (north).msw

Design Results

Masonry wall

GENERAL INFORMATION:

Global status : OK

Design code : ACI 530-02

Geometry:

Total height : 24.00 [ft]
 Wall Thickness : 9.63 [in]
 Total length : 144.00 [ft]
 Base support type : Continuous
 Wall bottom restraint : Pinned
 Column bottom restraint : Fixed
 Rigidity elements : Flanges

Materials:

Material : CMU 1.5-60
 Mortar type : Port/Mort - M/S
 Mortar bed type : Face shell
 Grouting : Partial
 Steel tension strength (Fs) : 2.4E4 [Lb/in²]
 Masonry compression strength (F'm) : 1.5E3 [Lb/in²]
 Steel elasticity modulus (Es) : 2.9E7 [Lb/in²]
 Masonry elasticity modulus (Es) : 1.35E6 [Lb/in²]
 Masonry unit weight : 0.14 [Kip/ft³]

Number of stories: 1

Story	Story height [ft]
1	22.00

General information.

- **Bearing wall Design.** This second part of the report shows the design results of the walls considered as bearing walls. It is subdivided in:
 - Bearing wall status and plot
 - Segment geometry
 - Vertical reinforcement
 - Results
- **Shear wall Design.** It shows the design results of the walls considered as shear walls. It is subdivided in:

Example 8: Masonry Wall

- Shear wall status and plot
- Segment geometry
- Vertical and horizontal reinforcement
- Results
- ***Column Design.*** It shows the design results of the columns. It is subdivided in:
 - Column status and plot
 - Column geometry
 - Reinforcement
 - Results
- ***Lintel Design.*** It shows the design results of the lintels. It is subdivided in:
 - Lintel status and plot
 - Lintel geometry
 - Reinforcement
 - Results

Design Status

The report presents a general status for bearing walls, shear walls, columns and lintels. There are three possible options:

- “OK” when all the elements fulfill the requirements and limitations of the code
- “Warnings.” when some elements fail one or more limitations of the code.
- “N.G.” when some elements fail one or more requirements of the code.