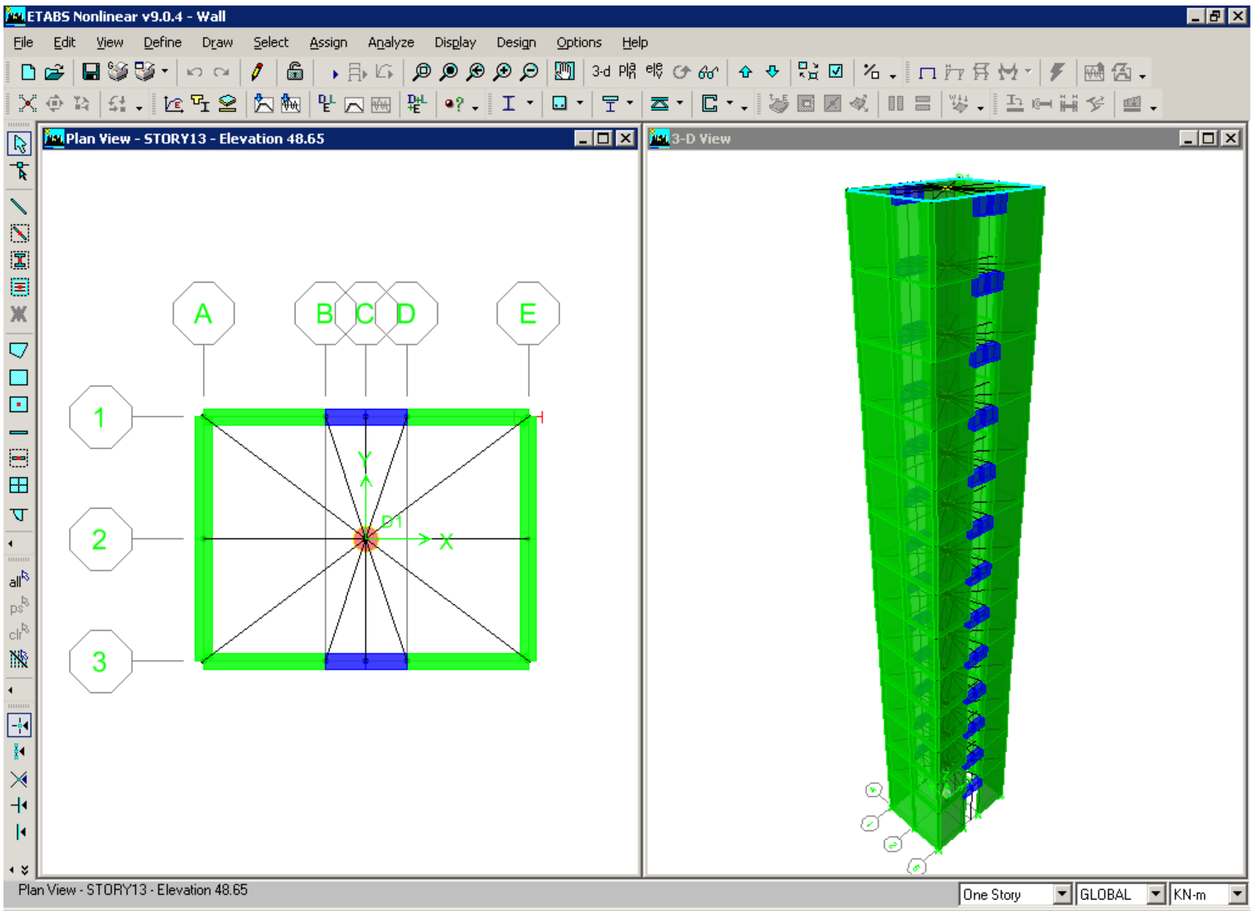
Tutorial for ETABS V9

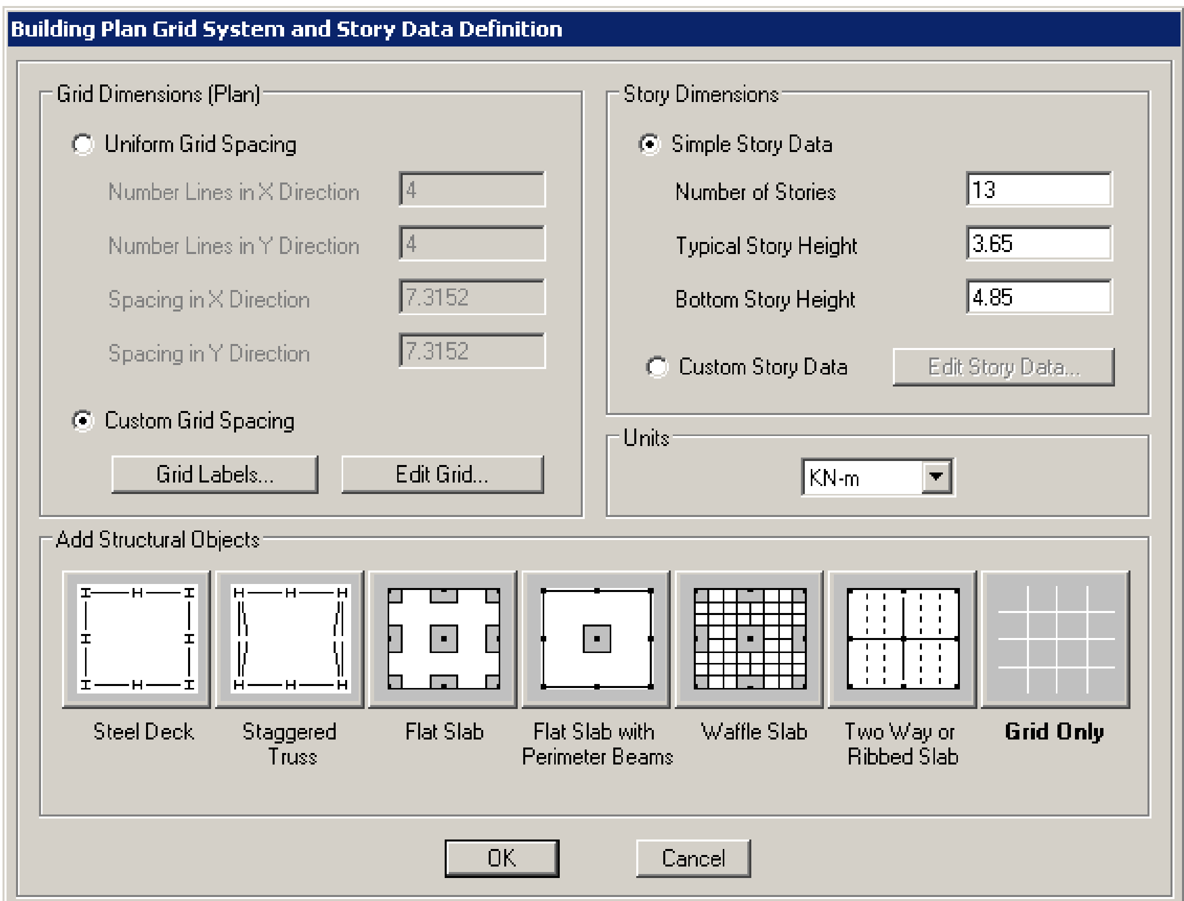
3-D Modelling and Dynamic Analysis of a RC Core Wall

This short manual provides step-by-step instructions to model and dynamically analyze the RC core wall seismically designed in the Third Edition of the Concrete Design Handbook (CDH) at Section 11.5 of Part II. The figure below shows the model resulting from these instructions (colours may be different).



Step-By-Step Procedure

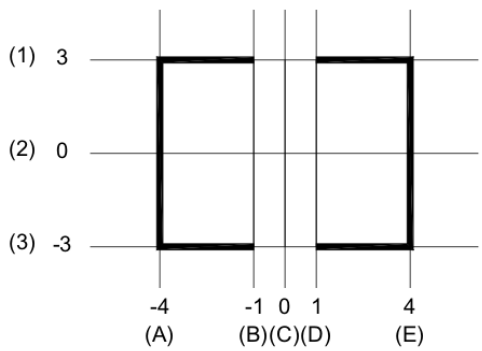
1. Click **File > New Model...**
2. In the form **New Model Initialization**, click **No**.
3. In the form **Building Plan Grid System and Story Data Definition**, set the parameters **Grid Dimensions (Plan)**, **Story Dimensions** and **Units** as shown in the figure below. Once set, click on **Edit Grid...**



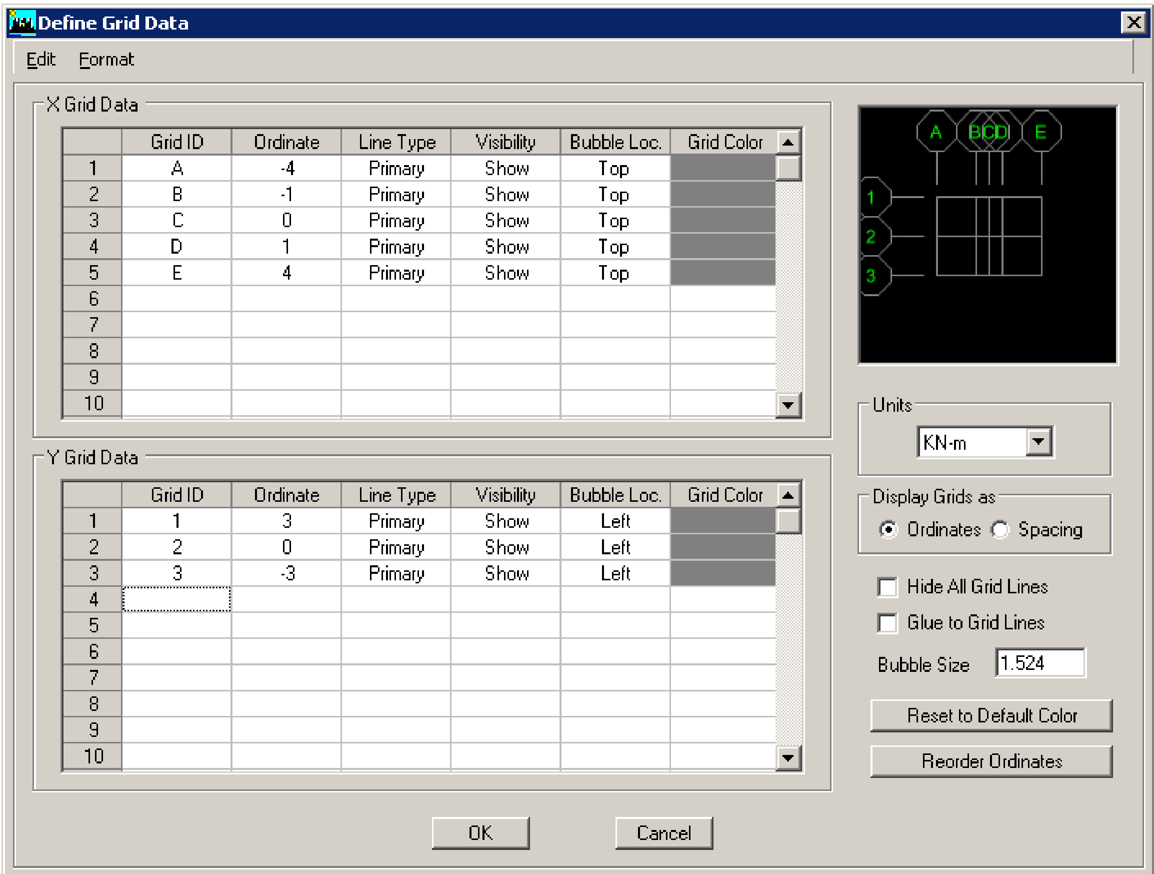
4. The form **Define Grid Data** appears. By default, ETABS defines four X-Grids numbered from A to D and four Y-Grids numbered from 1 to 4. From this default, one X-Grid will be added and one Y-Grid will be deleted for the present example.

To add a grid line, simply enter a **Grid ID** and an **Ordinate** in the line below those already defined. At this new line, double click on the empty spaces associated to **Line Type**, **Visibility** and **Bubble Loc.** to define these parameters. To delete a grid line, click on the number of the grid line to be deleted to highlight the line. Leave the mouse cursor over the highlighted line, right-click on your mouse and then select **Delete** in the form that appears.

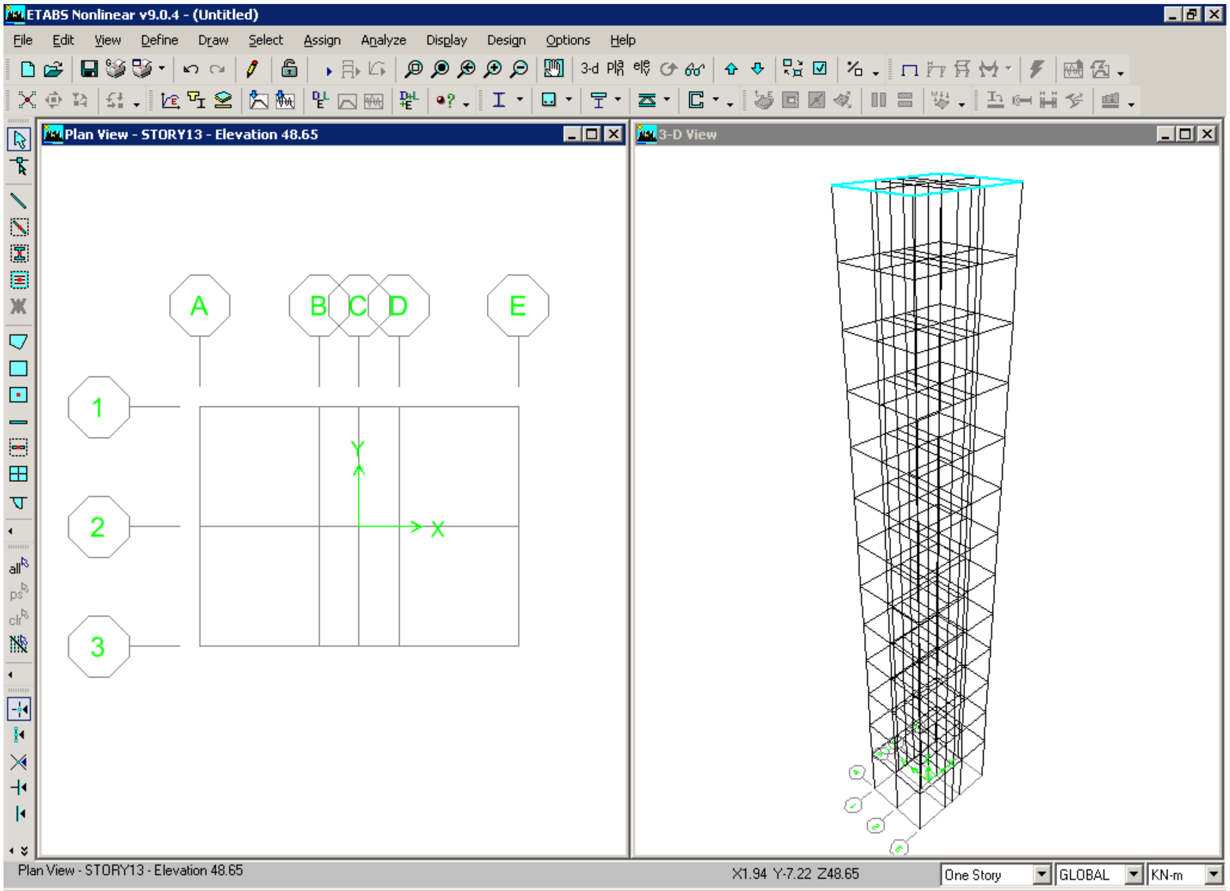
In this example, the grid data will be defined as shown below (the thick lines represent the wall cross-section to be defined later).



Based on the above figure, set the form **Define Grid Data** as shown in the figure below. Once set, click **OK**. Click **OK** to close the other form.

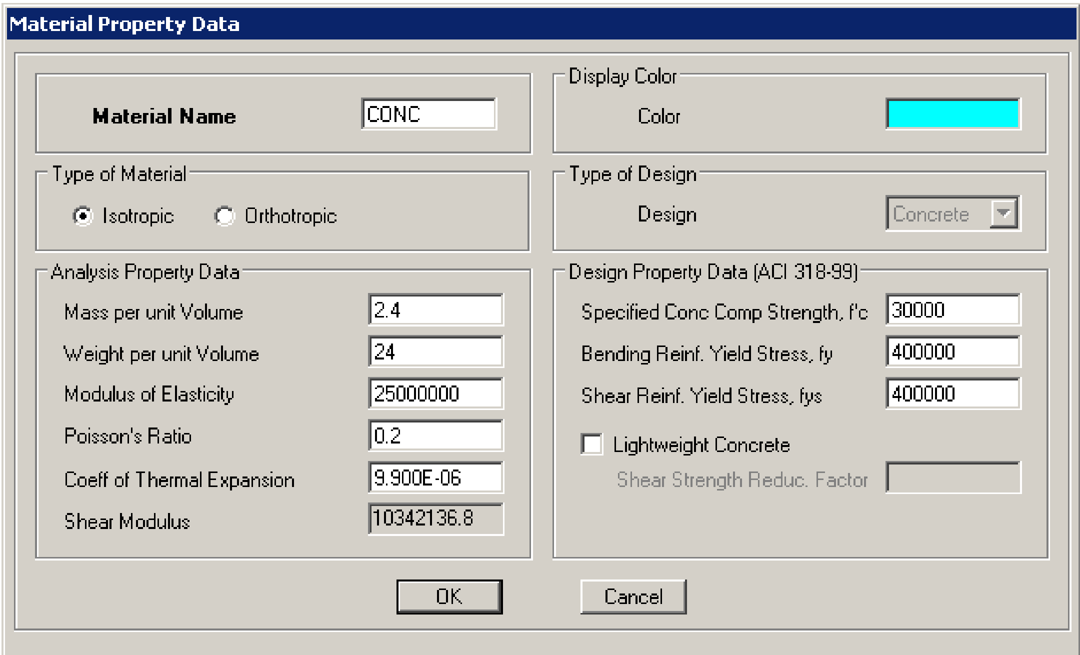


The figure below shows the result of the grid data definition.



5. Save your ETABS file by clicking **File > Save...** Prior to saving, make sure that the units are still in **kN-m** (see the lower right corner of the main window, as shown in the above figure)

6. Definition of the material properties. Click **Define > Material Properties...** Three materials are already defined in ETABS: **CONC**, for concrete, **STEEL** and **OTHER**. The material **CONC** will be modified for the example. Highlight **CONC** and click **Modify/Show Material...** Set the material properties as follows, click **OK** once completed and **OK** again to close the form **Define Materials**:

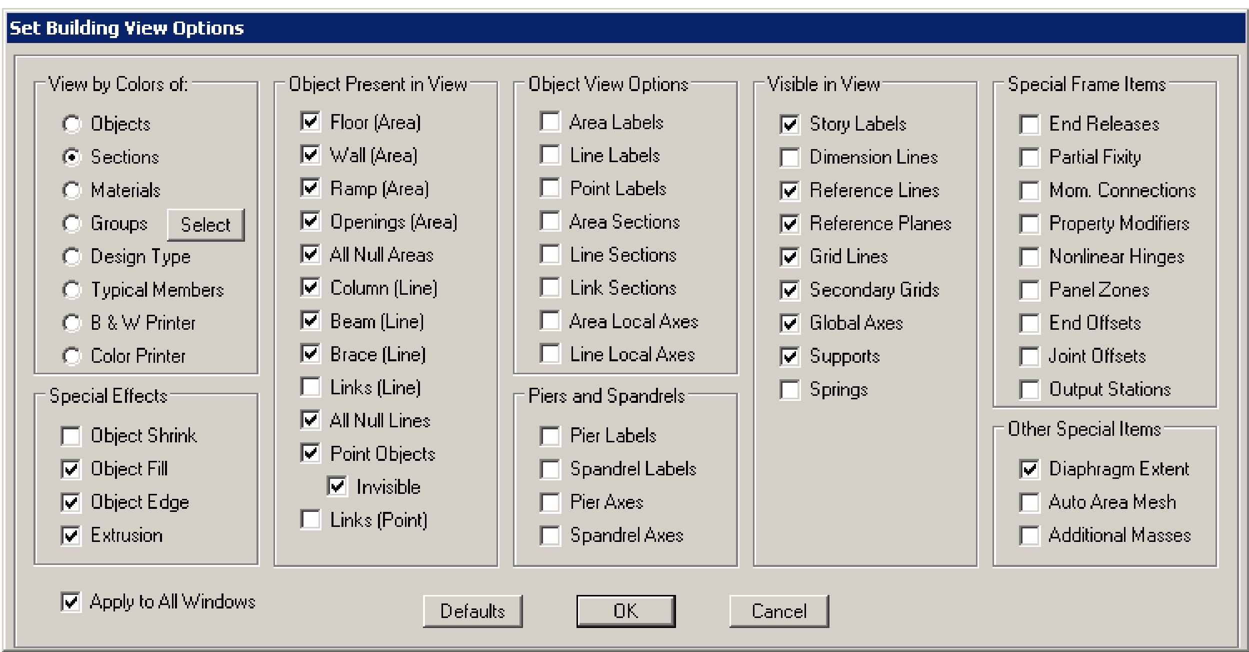


7. Definition of the member sections. Both the wall and the coupling beam members of the core wall could be modelled with plane elements. However, for the example, the wall members will be modelled with plane elements while the coupling beam members will be modelled with beam elements.

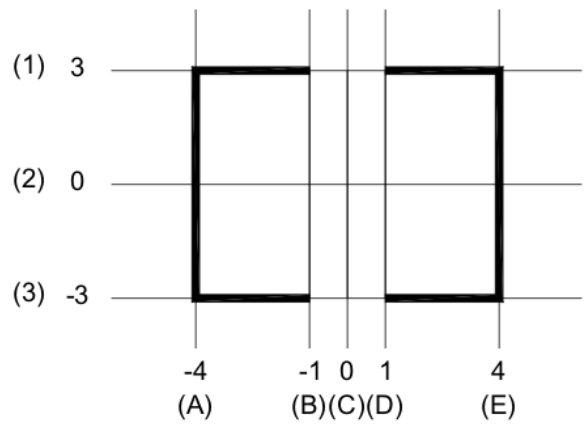
   To define sectional beam properties, click **Define > Frame Sections…** Several default frame sections are defined in ETABS. For the example, a new frame section is defined. In the form **Define Frame Properties**, click the drop-down list that reads **Add I/Wide Flange** in the **Click To** area. Scroll down the resulting list until **Add Rectangular** and then click on it. In the form **Rectangular Section**, give a section name, select the appropriate material and enter the dimensions of the beam section. Click **Set Modifiers…** and enter the ratios used to make allowance for cracking. Click **OK** three times to close all forms.

   To define sectional wall properties, click **Define > Wall/Slab/deck Sections…** Highlight **Wall1** in the **Sections** area of the form **Define Wall/Slab/deck Sections** and then click **Modify Show/Section…** in the **Click To** area. In the form **Wall/Slab Section**, give a section name, select the appropriate material and enter the thickness in membrane and bending of the wall section. Click **Set Modifiers…** and enter the ratios used to make allowance for cracking. Click **OK** three times to close all forms.

8. Prior to modelling members, make sure that the left window **Plan View** is at **STORY13**. If not, click **View > Set Plan View…** Highlight **STORY13** and click **OK**. In the right window **3-D View**, the top of the upper storey should be highlighted, indicating the plan view showed in the left window.
9. At the bottom of the main window, click the drop-down list that reads **One Story** and select **All Stories**. This command enables to apply at all stories what it is performed at the working story. This command is useful for this example because the wall cross-section is uniform over the entire height of the wall. Actually, with this command activated, the structural members generated at one story will be automatically generated at all stories.
10. In order to readily visualize the members generated, click **View > Set Building View Options…** In the associated form, check boxes as shown in the figure below. Click OK to close the form.

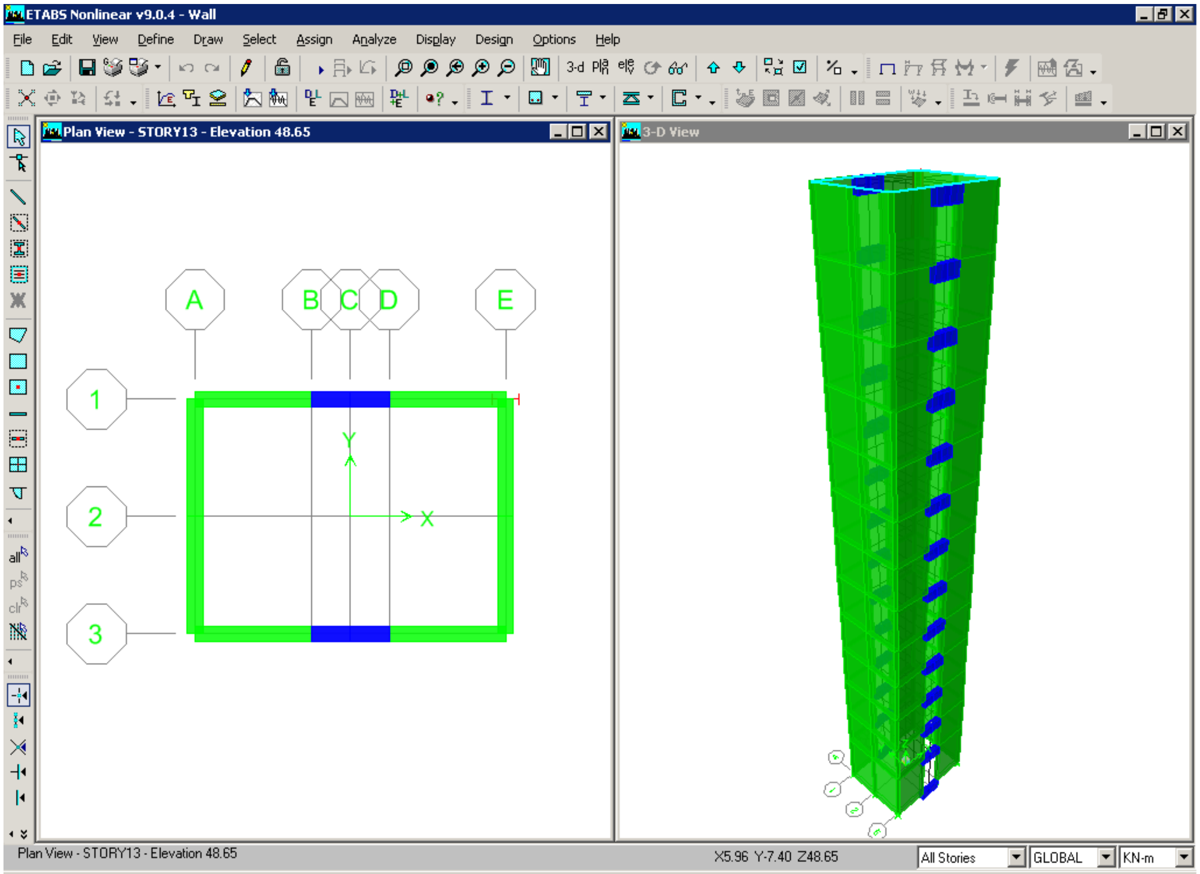


11. Modelling of the structural members. First the wall members are generated. Click **Draw > Draw Area Objects > Draw Walls (Plan)**. The form **Properties of Object** appears. Use the default values in this form but make sure that the name of the wall section defined previously is assigned to **Property**. In the **Plan View – STORY13**, select with the mouse cursor the grid points such to draw two unconnected C-shaped sections, as shown in the figure below. When one section is drawn, click the right button of the mouse to stop drawing the section. When both sections are drawn, press the **ESC** button to exit the **Draw Walls** command.



Secondly the coupling beam members are generated. Click **Draw > Draw Line Objects > Create Lines in Region or at Clicks (Plan, Elev, 3D)**. The form **Properties of Object** appears. In this form, assign to **Property** the coupling beam section previously defined by clicking on the default property and selecting the desired one. For the other parameters in this form, use the default values. In the **Plan View – STORY13**, move the mouse cursor over the line segments B-C and C-D of the grid line 1 and click on both. This generates coupling beams at all stories along grid line 1. Repeat the same steps for grid line 3. Once completed, press the **ESC** button to exit the **Draw Line** command.

The figure below shows the resulting wall structure.



12. Creation of a point object at the center of the whole wall section. **Click Draw > Draw Point Objects**. In the window **Plan View – STORY13**, select with the mouse cursor the grid point C2. Once completed, press the **ESC** button to exit the **Draw Point** command.

13. Generation of diaphragms. Click **Select > At Pointer/In Window**. In the window **Plan View – STORY13**, select with the mouse cursor all grid points by making a selection box enclosing the whole wall section. Click **Assign > Joint Point > Diaphragms…** Highlight the diaphragm **D1** and click **OK**. A diaphragm should appear at each story.

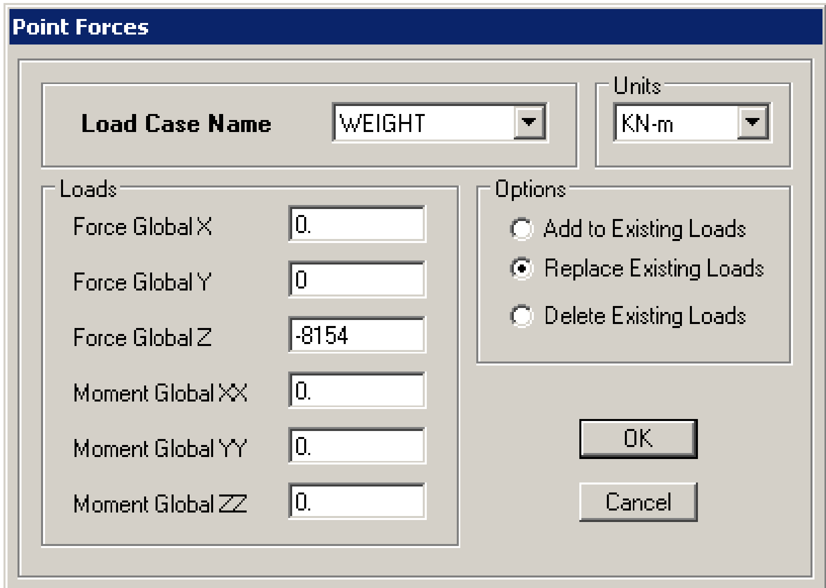
14. Removing of the beam elements at the base. Click **View > Set Plan View…** Select the plan level **BASE** and click **OK**. At the bottom of the main window, click the drop-down list that reads **All Stories** and select **One Story**. Prior to selecting beam elements, clear any possible undesired selections by clicking **Select > Clear Selection**. In the window **Plan View – BASE**, select with the mouse cursor all beam elements by enclosing them with a selection box. Once selected, click

the right button of the mouse and select **Delete Selected Objects**. Do the same operation to delete the point object at grid point C2.

15. Fixing the wall model at its base. In the window **Plan View – BASE**, select with the mouse cursor all grid points displayed (make sure that the selection is for **One story** only). Click **Assign > Joint Point > Restraints (Supports)…** In the form **Assign Restraints**, check all check boxes and click **OK** to close the form.

16. Mass definition. Different ways are possible to define mass. In this example, the mass is defined from loads, which are the seismic weights of the building. Consequently, a static load case must first be defined. Click **Define > Static Load Cases…** In the **Loads** area of the form **Define Static Load Case Names**, type a name (for instance WEIGHT) in the **Load** field, select **DEAD** in the **Type** field and set **Self Weight Multiplier** to 0. In the **Click To** area of the form, click **Add New Load**. Delete the default load cases **DEAD** and **LIVE** by highlighting them and clicking **Delete Load** in the **Click To** area. Once completed, click OK to close the form. Click **Define > Mass Source…** In the form **Define Mass Source**, select **From Loads** in the **Mass Definition** area and click **Add** in the **Define Mass Multiplier for Loads** area. Leave the check boxes checked. Click **OK** to close the form.

    Click **View > Set Plan View…** Select the plan level **STORY12** and click **OK**. In the window **Plan View – STORY12**, select with the mouse cursor the grid point C2 (make sure that the selection is for **One story** only). Click **Assign > Joint Point/Loads > Force…** In the form Point Forces, set the parameters as shown in the figure below (Note: the **Load Case Name** is that defined previously). Click **OK** to close the form. Repeat these steps for all stories where a seismic weight is given in the CDH.



In order to verify the force assigned at a point object, simply move the mouse cursor over this object to highlight it and click on the right button of the mouse. A **Point Information** form appears. Select the tab **Loads**. Click **OK** to close the form.

17. Definition of the design acceleration response spectrum. Click **Define > Response Spectrum Functions…** In the **Click To** area of the form, click the drop-down list and select **Add User Spectrum**. In the form **Response Spectrum Function Definition**, type a **Function Name** and enter in the **Define Function** area the period and design spectral acceleration values (in fraction of gravity) given in the CDH. Once completed, click **OK** and **OK** again to close the form.
18. Definition of two response spectrum cases, one for each design earthquake loading direction. Click **Define > Response Spectrum Cases…** In the form, click **Add New Spectrum…** In the form **Response Spectrum Case Data**, type a **Spectrum Case Name.** In the **Input Response Spectra area**, click on the drop-down list **Function** of the direction **U1** (Global X) to select the spectrum function previously defined and enter 9.81 as **Scale Factor** (make sure that **Units** is still **kN-m**). Click OK. Repeat the previous steps to define the response spectrum case in the orthogonal direction.
19. Definition of a pier entity to get wall forces in an appropriate format for design. At the bottom of the main window, click the drop-down list that

reads **One Story** and select **All Stories**. Click **View > Set Plan View…** Select the plan level **STORY13** and click **OK**. In the window **Plan View – STORY13**, select with the mouse cursor one of the two C-shaped wall sections. Click **Assign > Shell/Area > Pier Label…** In the **Wall Piers** area of the form **Pier Names**, type a name of pier. Click **Add New Name** in the **Click To** area. Leave the new name highlighted and click **OK** to close the form. Repeat these steps to define a pier entity for the other C-shaped wall section.

20. Modelling verification of the whole wall model. Click **Analyze > Check Model…** Check all boxes in the form **Check Model** and click **OK**. A **Warning** form appears. If no issue/problem is detected, the message "**Model has been checked, No warning messages**" appears in this form. Close the form. Otherwise, fix the detected problems.
21. Analysis setting. Click **Analyze > Set Analysis Options…** In the form **Analysis Options**, select **Full 3D**, check only **Dynamic Analysis** and click on **Set Dynamic Parameters…** In the form **Dynamic Analysis Parameters**, enter the number of modes to be calculated, set **Type of Analysis** to **Eigenvectors** and click **OK**. Click **OK** to close the form **Analysis Options**.
22. Run the requested analysis. Prior to running analysis, make sure that **Units** is **kN-m** and save the model. Click **Analyze > Run Analysis**.
23. Displaying analysis results. Once the dynamic analysis is completed, set first **Units** to **kN-m** if necessary. To display results, click **Display** and select the desired results.

ETABS enables to display results in tables that can be copied and pasted in worksheets of a calculator, such as Microsoft Excel. To display these tables, click **Display > Show Tables**… Select the desired results and load cases, and click **OK**.