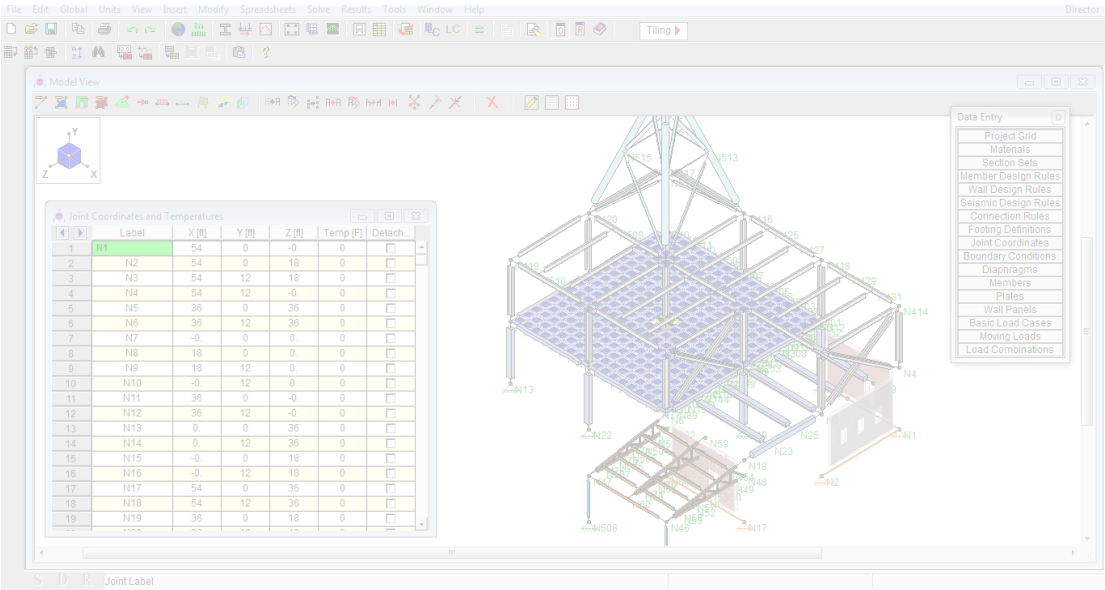
Chapter 2

Getting Started with RISA-3D

Learning Objectives

After completing this chapter, you will be able to:

- *Start a new project*
- *Configure application settings*
- *Configure model settings*
- *Create and set the grids*
- *Save and exit the project*
- *Open an existing project*



INTRODUCTION

In this chapter, you will learn to start a project, set up global and project parameters, and save a project. In RISA-3D, you can do preliminary settings for units, fonts, backup, graphics, result printing, analysis steps, and so on. You can also use these custom settings in other projects.

STARTING A NEW PROJECT

RISA Menu:	File > New
Shortcut Keys:	CTRL+N

On starting RISA-3D for the first time, the RISA-3D interface is displayed with a new project loaded by default. Also, the **Starting a Model** dialog box will be displayed along with it, refer to Figure 2-1.

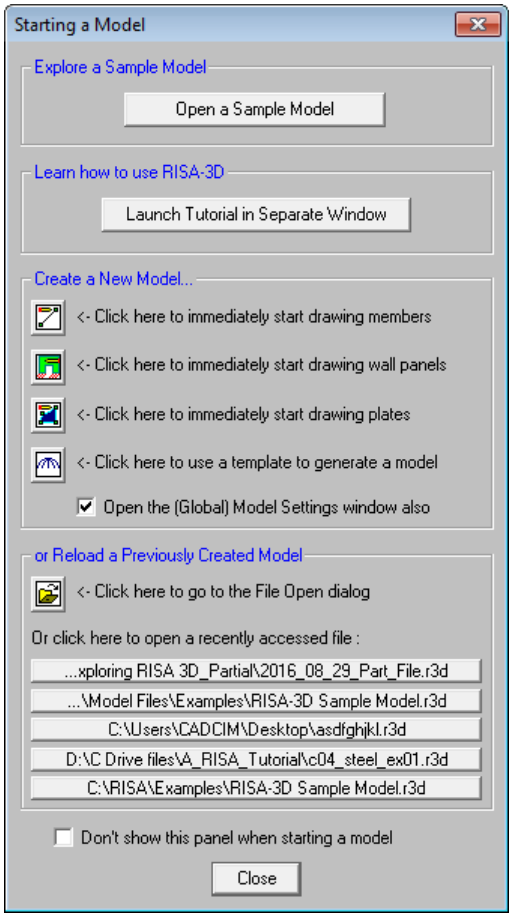


Figure 2-1 *The* ***Starting a Model*** *dialog box*

The **Starting a Model** dialog box contains options to open a sample model, launch RISA user's guide, create a new model or reload a previously created model. To open a sample model, choose the **Open a Sample Model** button in the **Explore a Sample Model** area. Similarly, to launch RISA user's guide, choose the **Launch Tutorial in Separate Window** button in the **Learn how to use RISA-3D** area. Various quick access buttons are available in the **Create a New Model**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

area in the **Starting a Model** dialog box. These quick access buttons allow you to create or sketch new members, wall panels, plates, and also to generate a model using a template. The **or Reload a Previously Created Model** area of this dialog box displays five recently accessed files. You can open a file by clicking on it.

CONFIGURING APPLICATION SETTINGS

You can customize any of the parameters, design, and analysis options in RISA-3D and thus modify the program to suit you and your work process. These options are discussed below.

Setting up the Application Parameters

RISA Menu:	Tools > Application Settings

To configure application settings, choose the **Application Settings** option from the **Tools** menu; the **Application Settings (Preferences)** dialog box will be displayed, as shown in Figure 2-2. The **Application Settings (Preferences)** dialog box contains all available options to customize the interface and graphics in RISA-3D. This dialog box contains six tabs named **General**, **Data Entry**, **Solution and Results**, **Fonts**, **Printing**, **File Locations**. These tabs and their options are discussed next.

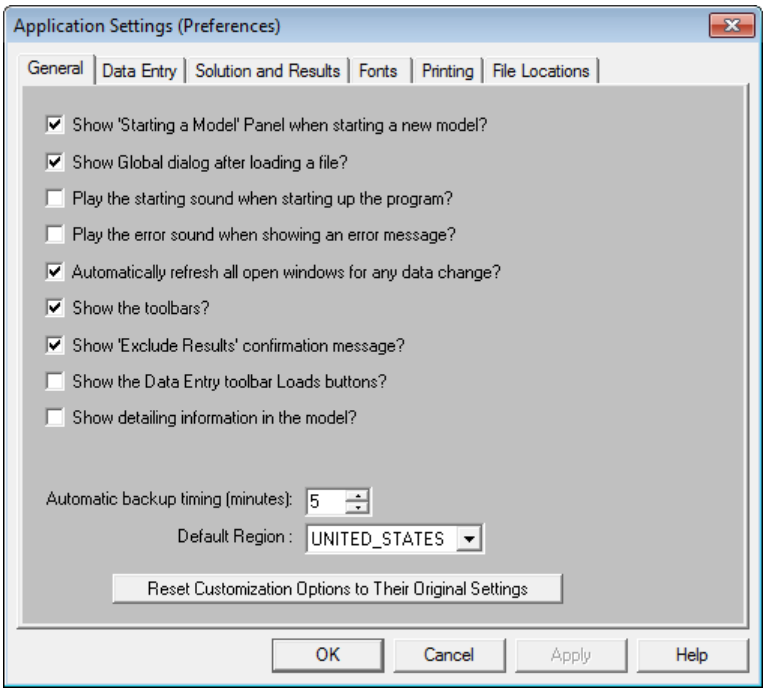


*Figure 2-2 The **Application Settings (Preferences)** dialog box*

General Tab

The **General** tab in the **Application Settings (Preferences)** dialog box contains the options related to the appearance, sound settings, backup file settings, and so on. Select the **Show "Starting a Model" Panel when starting a new model?** check box to display the "**Start a Model**" display box at startup.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

You can select the **Show Global dialog after loading a file?** check box to display the **(Global) Model Settings** dialog box containing the options related to the project details, codes of practice, shear ties and stress options for concrete material, seismic parameters, and more. To play the startup sound while starting up RISA-3D, select the **Play the starting sound when starting up the program?** check box. The error messages in RISA-3D are accompanied with an error sound. You can use the **Play the error sound when showing an error message?** check box to play or mute this sound at the time of display of the error. In RISA-3D, you can automatically update all the open windows for any changes made to the data by selecting the **Automatically refresh all open windows for any data change?** check box. The **Show the toolbars?** check box toggles on/off the display of all the toolbars. The **Show the Data Entry toolbar Loads buttons?** check box toggles on/off the display of the load buttons in the **Data Entry** toolbar. The **Automatic backup timing (minutes)** option is used to set the automatic backup frequency. You can manually enter any value directly in the edit box or you can use the spinner. The value should be entered in minutes. The **Default Region** drop-down list contains the list of countries to choose the default region for the project. The **Reset Customization Options to Their Original Settings** button is used to reset all the customized settings to its factory defaults.

Note

The customized settings cannot be rolled back to factory defaults. It is an irreversible process.

Data Entry Tab

In this tab, you can customize the height of the rows by entering a value in the **Row height for Data Spreadsheet** edit box or by increasing or decreasing the value set in the spinners. The **Decimal Places for Data Entry Fields** is used to specify the maximum number of digits to be displayed after the decimal point. You can select any value in the range of 1 to 4. In this tab, you can also customize the default prefix of joint, member, plate, wall panel, and solid labels.

Solution and Results Tab

If any instability is found in the joints at the time of analyzing the structure, it will be automatically locked and the program will be allowed to proceed. Rotational instabilities are commonly inconsequential and RISA-3D allows these instabilities to be locked without any warning. Select the **Lock isolated ROTATIONAL instabilities without notification?** check box to lock any rotational instability if present, refer to Figure 2-3. To create a warning before clearing the previous results, select the **Always warn before clearing results?** check box. To eliminate the slender check for steel structure design, clear the **Allow KL/r>200, l/r>300 for LRFD, CAN steel design?** check box or select it to allow the check for the design. Select the **Don't require P-Delta analysis for AISC, CSA, ACI, etc.?** check box to waive off the P-Delta analysis for certain design codes.

RISA-3D also warns you if the total of joint reactions is not equal to the summation of applied loads. To enable it, select the **Warn if total reaction does not equal total load?** check box. You can also adjust the row height for the result browser by choosing a value from the **Row height for Results Browsers** spinner in the range of 15 to 24. Specify a value in the **Decimal Places** edit box to set the maximum number of digits to be displayed after the decimal point. You can group the results under a particular load combination or you can group them under individual

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

items by selecting the **Batch results displayed by COMBINATION** or **Batch results displayed by ITEM** check box, respectively.

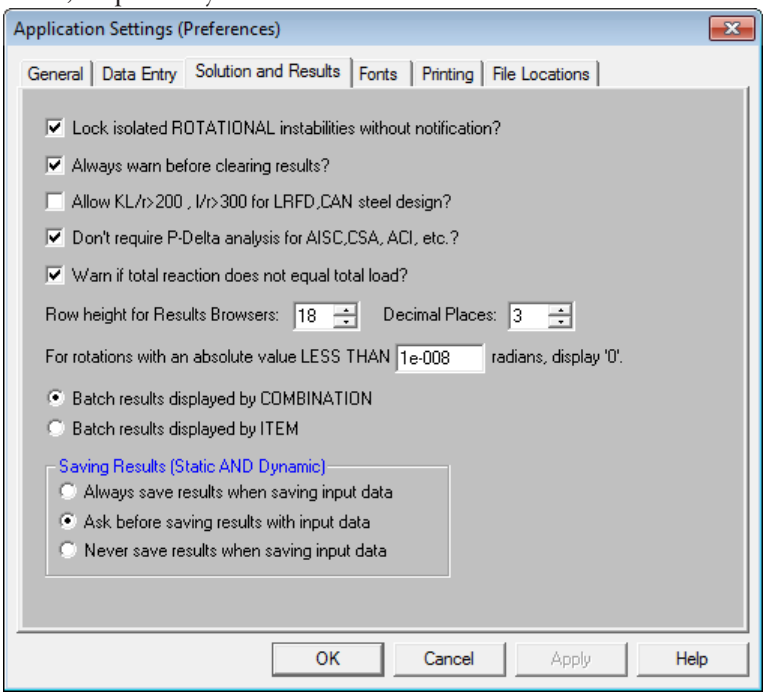


Figure 2-3 *The* ***Solutions and Results*** *tab of the* ***Application Settings (Preferences)*** *dialog box*

Fonts Tab

The **Fonts** tab contains options to personalize the annotation and spreadsheet formats such as font, font style, and font size. In RISA-3D, you can assign different font styles to spreadsheets, members and joints labels, reports, graphics, solid labels, and more. To do so, choose the button corresponding to the option in which you want to make changes. For example, to change format of spreadsheets, choose the **Select Spreadsheet Font** button from the **Fonts** tab; the **Font** dialog box will be displayed. The **Font** dialog box contains three areas for formatting. The **Font** area contains a list of all the fonts available in RISA-3D. Choose any font from the list. The **Font Style** area contains different font styles for the font selected. For example, if you choose the **Arial** font from the **Font** area, 10 format styles will be available for the selected font such as **Narrow**, **Narrow Italic**, **Regular**, **Bold**, **Black**, **Black Oblique,** and so on. You can use any of the font styles available in the **Font style** area. You can also modify the size of the font from the **Size** area in the **Fonts** dialog box. The list in this area contains default sizes available ranging from 8 to 20. You can choose any size from the list box or enter your own custom size by clicking in the edit box. The **Sample** area of the **Fonts** dialog box shows the preview format chosen.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Printing Tab

The **Printing** tab includes settings for printing graphics and reports. The **GRAPHICS Preferences** area of this tab contains settings for printing graphical sketches in the reports. You can select the **Portrait** or **Landscape** radio button to change the view orientation of the sketch.

To print the date and time along with the result, select the **Show Date and Time?** check box. You can also customize the sketch numbers and the prefix of the figures in the result. The **Margins (Inches)** area in the **Printing** tab is used to set the print margins. You can set the left, right, top, and bottom values for the margin in their corresponding edit boxes.

File Location Tab

The locations for files such as model data files, RISA section files, databases, temporary space, load combination lists, importing, back up files can be defined separately. To change the location of any of these files, choose the corresponding option from the **Path and Current Setting, Use Browse to Modify** drop-down, refer to Figure 2-4. Next, choose the **Browse** button; the **Select ANY file to set the path** dialog box will be displayed. In this dialog box, specify the path for the file location.

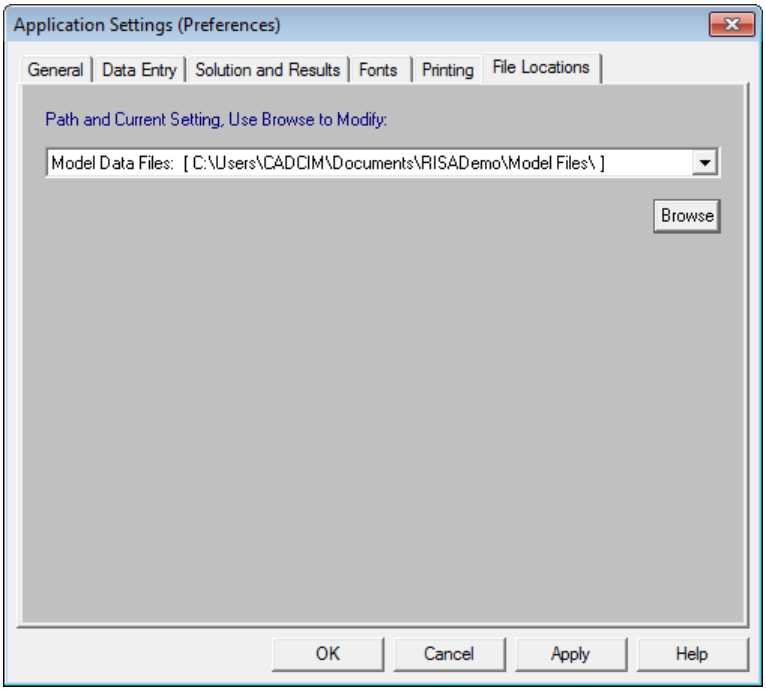


Figure 2-4 *The options in the* ***Path and Current Setting, Use Browse to Modify*** *drop-down list in the* ***Application Settings (Preferences)*** *dialog box*

Setting Project Units

RISA Menu: Units

You can set a unit system for your project. To do so, choose the **Units** menu from the

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Menu bar; the **Units Selection** dialog box will be displayed, as shown in Figure 2-5. Alternatively, choose the **Define units to be used** tool from the **RISA** toolbar.

The **Units Selection** dialog box contains different drop-down lists having both imperial and metric units. You can choose any option from the drop-down lists. For example, you can select the **inches** option from the drop-down list in the **Lengths** area to change the unit for length parameter from feet to inches. Select the **Converting Existing Data For Any Units Changes?** check box to convert the values already present in the current project. You can also save your custom unit settings as default. To do so, select the **Save these units settings as the default settings?** check box. To change the entire unit system to imperial or metric, choose the **Standard Imperial** or **Standard Metric** button, respectively.

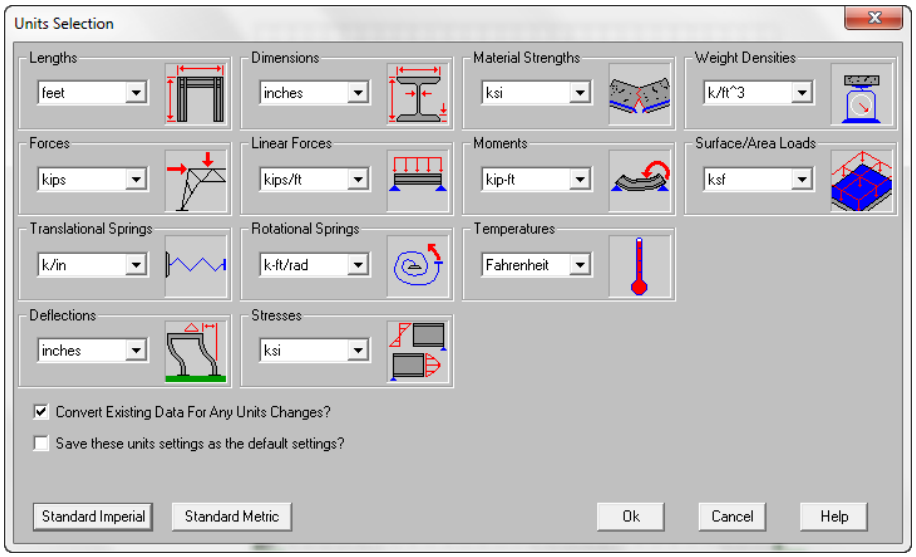


Figure 2-5 *The* ***Units Selection*** *dialog box*

Note

The ***Convert Existing Data For Any Units Changes?*** *check box is disabled until you change any of the units from the drop-down lists available.*

CONFIGURING (GLOBAL) MODEL SETTINGS

You can define the model parameters for a project in RISA-3D. To set the model parameters, choose the **Settings** menu; the **(Global) Model Settings** dialog box will be displayed with the **Description** tab chosen by default. This dialog box contains six tabs. These tabs are discussed next.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Description Tab

This tab contains the project information such as **Model Title**, **Company**, **Designer**, **Job Number**, and **Checked By**, as shown in Figure 2-6. You can add additional information in the **Notes** text box. This information can be saved as default, and then can be used in other projects. You can choose the **Save as Defaults** button to save the settings as default so that the specified settings are loaded the next time when you open RISA-3D.

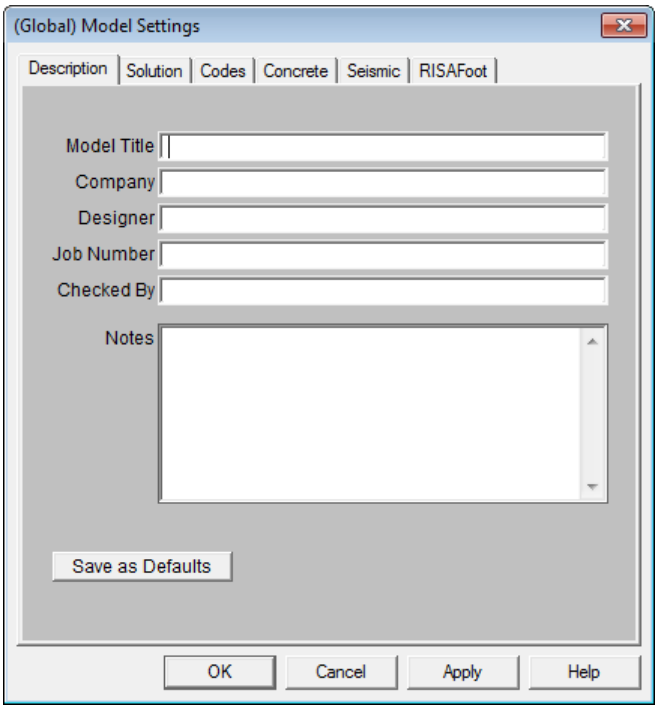


Figure 2-6 *The* ***(Global) Model Settings*** *dialog box*

Solution Tab

The parameters under the **Solution** tab are used to control the settings that affect general analysis of the model. You can adjust the number of sections defined for a single member. For each section defined, the report for member force, stress, torsion, and deflection results will be generated. This adds more detail to the output result files without affecting the analysis of the model or the code checks. To set the number of sections, choose a number between 2 to 20 from the **Number of Sections** spinner in the **Members** area, refer to Figure 2-7.

In the **INTERNAL Sections** edit box, you can specify the number of internal sections for which RISA-3D will calculate and save the results such as forces, stresses, code checks, and more. If you want to include the shear deformation and torsional warping in the model analysis, select the **Shear Deformation** and **Torsional Warping** check boxes. You can define the maximum size of

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

the area load mesh by entering its value in the **Area Load Mesh** edit box.

Note

Number of sections affect the analysis and design results. More the number of sections, more will be the output result.

In the **Wall Panels** area of the **Solution** tab, you can set the base mesh size for the wall panels before solving it. To define the base mesh size, enter a value in the **Mesh Size** edit box. To perform the P-Delta analysis on the wall panels, select the **Include P-Delta for Walls** check box.

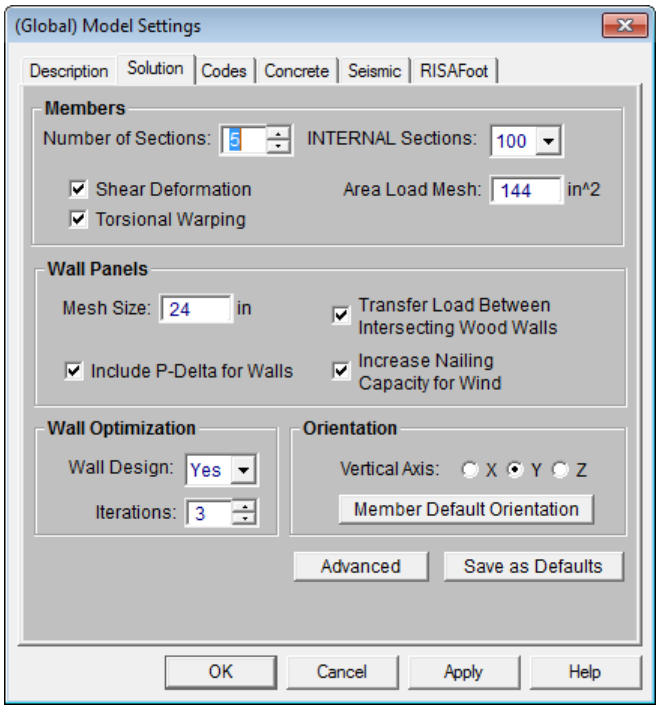


Figure 2-7 *The* ***Solution*** *tab of the* ***Global Parameters*** *dialog box*

You can also specify whether you want RISA-3D to automatically update the design or stiffness of wall panels with iterated solutions. To do so, select the **Yes** option from the **Wall Design** drop-down list in the **Wall Optimization** area. You can also set the number of iterated solutions you want by choosing the **Iterations** spinner.

You can set the vertical axis for the model by selecting the **X**, **Y**, or **Z** radio button from the **Orientation** area of the **(Global) Model Settings** dialog box. To define the default member orientation, choose the **Member Default Orientation** button from the **Orientation** area; the **Set Default Member Orientation** dialog box will be displayed, as shown in Figure 2-8.

In this dialog box, if you want the local z axis of the member to be parallel to global XZ plane, select the **Force member local z axis to be parallel to the global XZ plane** radio button. Similarly,

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

if you want the local z axis of the member to be parallel to global XY plane or global YZ plane, select the **Force member local z axis to be parallel to the global XY plane** or **Force member local z axis to be parallel to the global YZ plane** radio button, respectively. After setting the default member orientation, choose the **OK** button to save the setting and close the **Set Default Member Orientation** dialog box.

In the **(Global) Model Settings** dialog box, choose the **Advanced** button; the **Set Advanced Options** dialog box will be displayed. In the **P-Delta** edit box, specify the percentage tolerance for determining the convergence of P-Delta analysis. In the Eigensolution: I.E -edit box, you need to specify the volume which indicates that how close a solution must be to the previous solution for a model to be considered as converged..

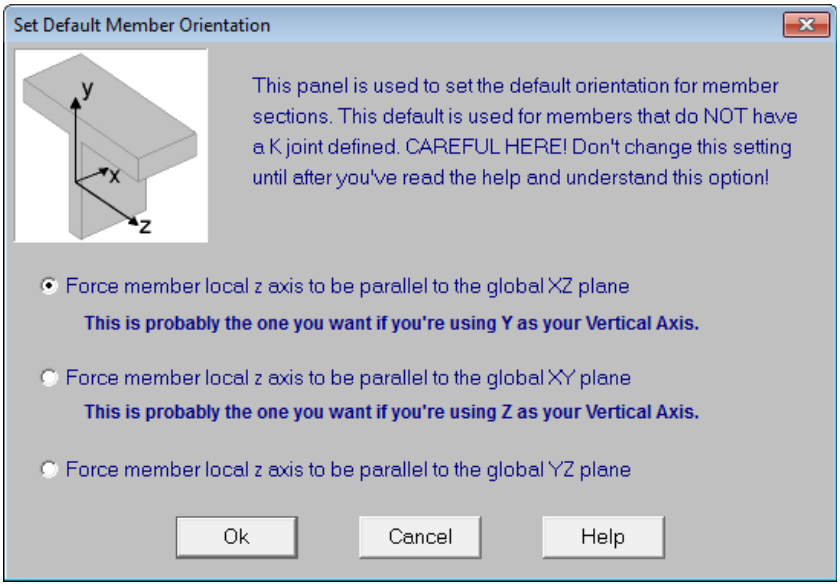


Figure 2-8 *The* ***Set Default Member Orientation*** *dialog box*

Specify the acceleration due to gravity and the model merge tolerance in the **Gravity Acceleration** edit box and **Merge Tolerance** edit box, respectively. Choose the solver types from the **Solvers** area for static and dynamic solutions. On selecting the **Standard Skyline** radio button for dynamic or static solution, the non-zero terms of the stiffness matrix are clustered around the main diagonal of the stiffness matrix. On selecting the **Sparse Accelerated** radio button from the **Static** area, the matrix size will be reduced to an absolute minimum by eliminating the storage of all zero stiffness terms. On selecting the **Accelerated Solver** radio button from the **Dynamic** area, the accelerated sub-space iteration with Lanzcos starting vector is used. After specifying the advanced options, choose the **OK** button; the **Set Advanced Options** dialog box will be closed. Choose the **Save as Defaults** button from the **Solution** tab in the **(Global) Model Settings** dialog box to save the settings as default for all further projects.

Codes Tab

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

The **Code** tab contains code options for the **HR Steel**, **Connections**, **CF Steel**, **Wood**, **Concrete**, **Masonry**, and **Aluminum**. You can choose an option from the drop-down list corresponding to these sections. You can also adjust the stiffness reduction provision of the defined **Hot Rolled steel** sections. To do so, select **Yes (Iterative)** or **Yes (Tau=1.0)** from the **Adjust Stiffness** drop-down list corresponding to the **HR Steel** section. To ignore the section stiffness reduction provision in the analysis, choose the **No** option from this drop-down list. You can also specify the temperature for the wood section by choosing an option from the **Temperature** drop-down list corresponding to the **Wood** section.

Concrete Tab

The **Concrete** tab contains options related to the analysis and design of concrete sections, refer to Figure 2-9.

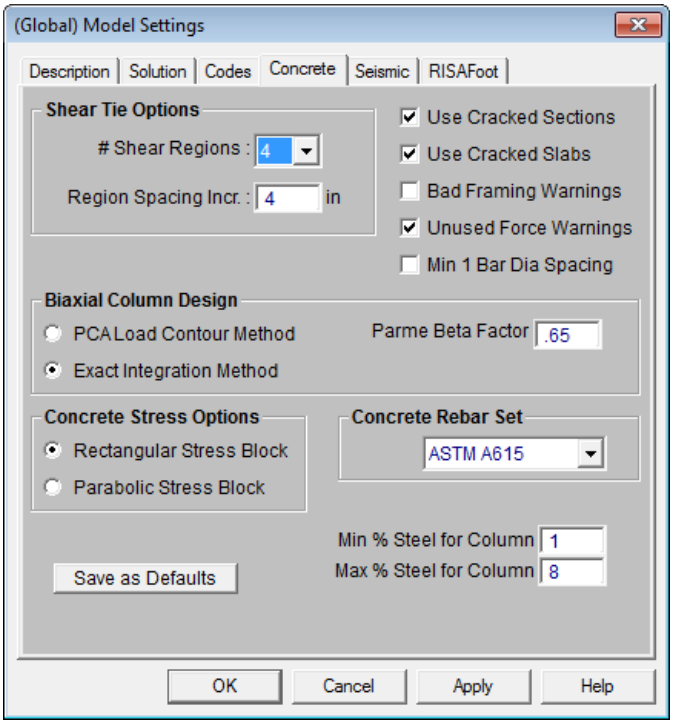


Figure 2-9 *The* ***Concrete*** *tab of the* ***(Global) Model Settings*** *dialog box*

From the **# Shear Regions** drop-down list of the **Shear Tie Options** area, choose an option from **2** or **4** to specify the number of shear regions you want to consider in analysis. At shear regions, you can also specify the shear tie increment to be considered while designing a concrete member. To specify it, you need to enter a value in the **Region Spacing Incr.** edit box of the **Shear Tie Options** area.

The **Biaxial Column Design** area of the **Concrete** tab controls the type of method that will be

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

used to determine the biaxial capacity of the column. If you want to determine the column's biaxial capacity by Exact Integration Method, select the **Exact Integration Method** radio button. To calculate the biaxial capacity by the PCA Load Contour Method, select the **PCA Load Contour Method** radio button. If you have selected the **PCA Load Contour Method** radio button, you can

also specify the column's approximate 3D interaction surface to be considered in the calculation. Specify this value in percentage in the **Parme Beta Factor** edit box of the **Biaxial Column Design** area. You can also modify the value for minimum and maximum percentage of steel in a column by specifying desired values in the **Minimum % Steel for Columns** and **Maximum % Steel for Columns** edit boxes, respectively. As per section 10.9.1 in ACI 318-11, the minimum and maximum steel values for a concrete column are set by default to 1% and 8% respectively.

RISAFoot Tab

The options in the **RISAFoot** tab are used to define parameters for design and analysis of the footings. In this tab, you can enter a value for factor of safety against sliding in the **OTM/Sliding Safety Factor** edit box. To optimize the solutions for OTM (overturning moment)/ sliding forces, select the **Optimize for OTM/Sliding** check box. The options in the **Concrete** area are used to specify the properties such as **Concrete Weight**, **Concrete fc** (strength), **Concrete Ec** (modulus of elasticity), and more. The **Steel** area of the **Footing** tab contains options related to the strength and minimum and maximum percentage of steel in an RCC member. You can also specify the cover for reinforcements in the **Footing Top Bar**, **Footing Bottom Bar**, and **Pedestal Bars** edit boxes. The default values for the concrete and steel parameters are as per the codes specified in the **Codes** tab of the **(Global) Model Settings** dialog box. Figure 2-10 shows the **RISAFoot** tab of the **(Global) Model Settings** dialog box.

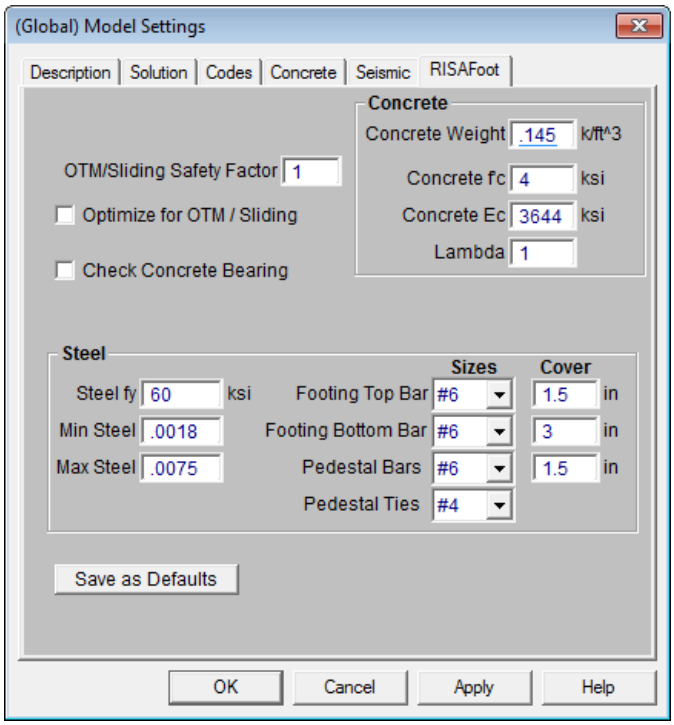


*Figure 2-10 The **RISAFoot** tab of the **(Global) Model Settings** dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Seismic Tab

The **RISA-3D Seismic Load Options** area of the **Seismic** tab contains the options for defining the earthquake load parameters. You can choose a desired code from the **Seismic Code** drop-down list for defining the earthquake load on the model. The other parameters for the design depends upon the code you select. For example, if you select the **None** option from the **Seismic Code** drop-down list, six basic parameters for the earthquake design will be displayed. The **Ct(X)** and **Ct(Z)** parameters are building period coefficient, **T(X)** and **T(Z)** which is the natural time period, and **R(X)** and **R(Z)** which is response modification factor in the x and z directions, respectively.

SETTING THE GRIDS

Grids are generally a series of vertical and horizontal reference lines at specified intervals. Grid lines are created for member references to facilitate the modeling process. You can add new members, plates, boundaries, and more to a project by drawing them from point to point at the grid intersection. In RISA-3D, you can draw two types of grids, project grid and drawing grid. These are discussed next in detail.

Project Grid

In RISA-3D, the project grid is part of the model and can be used to move the elements of the model simply by moving a grid line. In project grid, any change in the grid intervals is reflected in the repositioning of the corresponding hosted elements. You can specify the project grid only in the model plan view. The project grid is saved with the model view. Figure 2-11 shows project grids in 3D view.

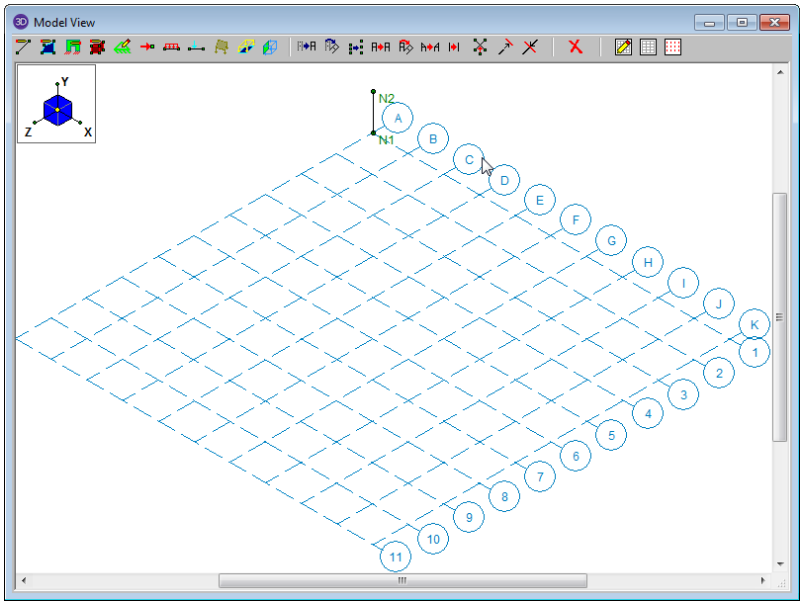


Figure 2-11 *Project grids in 3D view*

To define the project grid, choose the **Project Grid Spreadsheet** button from the RISA toolbar; The **Horizontal Project Grid Locations** dialog box will be displayed, as shown

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

in Figure 2-12.

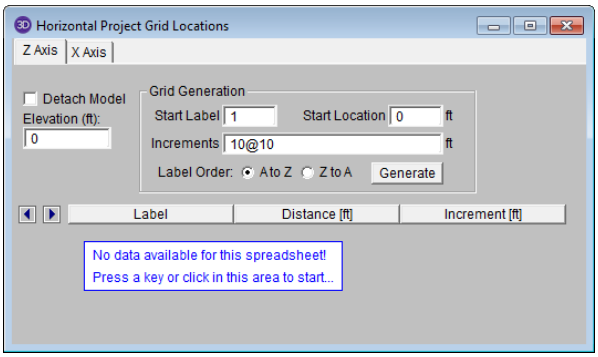


Figure 2-12 *The* ***Horizontal Project Grid Locations*** *dialog box*

The name of the dialog box will change based on the tab chosen. In the **Horizontal Project Grid Locations** dialog box, two tabs are present depending on your model plan view. By default, the **Z Axis** and **X Axis** tabs are available. This means your model elevation is parallel to Y axis. For more details, refer to the previous section of this chapter. In any or both of these available tabs you can set the location, label, increment, distance and more parameters defining the grid lines. To set the start label of the grid lines along any specific axis, enter a value in the **Start Label** edit box of the **Grid Generation** area. You can also specify the starting location of the grid with reference to the origin by entering a value in the **Start Location** edit box. The **Increments** edit box is used to specify the increment and the intermediate distance of the grids. For defining the grid lines, you can use symbols such as (@), or (,), or (/).

If you want to generate an equally spaced grid pattern of 8 increment with a distance of 10 units each, you should enter **8@10** in the increment edit box of the **Grid Generation** area. Next, choose the **Generate** button from the **Grid Generation** area; the pattern will be generated in the drawing window. Figure 2-13 shows the grid pattern generated in the plan view.

Figure 2-13 *Generated grid pattern with equal spacing*

Note

To generate the grid lines along an axis, you should define minimum one grid along its perpendicular axis.

To divide a large increment into smaller equally spaced increment, use the / symbol. For example if you want to divide a 10 unit long grid into 5 parts of 2 units each, the entry in the **Increments** edit box of the **Grid Location** area would be **10/5**. Choose the **Generate** button to generate the pattern. Figure 2-14 shows the generated grid pattern using the / symbol.

Figure 2-14 *Grid pattern generated using the / symbol*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

If you want to generate a grid pattern with variable increments, you can use the **,** (comma) symbol. For example, to generate a grid pattern with increment 2, 3, 4, and 5, enter **2,3,4,5** in the **Increment** edit box. This will generate five grid lines with increment 2, 3, 4, and 5 respectively. Figure 2-15 shows a grid pattern generated with variable increment.

Figure 2-15 *Grid pattern with variable increment generated using the, symbol*

Editing the Project Grid

After generating the project grid pattern, you can edit the spreadsheet or the grid pattern. The editing tools available for editing a grid appear only when the **Horizontal Project Grid Locations** dialog box is invoked. The tools will be present at the lower left corner below the RISA toolbar. Figure 2-16 shows the editing tools used for editing the grid pattern.

Figure 2-16 *Editing tools available for editing a grid pattern*

To add a new line in the spreadsheet, choose the **Insert a New Line Before the Current Line (F3)** tool from the editing bar or you can also press the F3 key. On choosing this tool, the **Insert Project Grid Line** dialog box will be displayed, as shown in Figure 2-17. In this dialog box, the currently selected spreadsheet line number is displayed at the top. You can specify the label of the new grid line and also the distance of the grid from the existing grid by entering values in the **Label for the new line** and **Distance from current line** edit boxes respectively. To add a new line before or after the current selected line in the spreadsheet, select the **Place new line BEFORE the current line** or **Place new line AFTER the current line** radio button respectively.

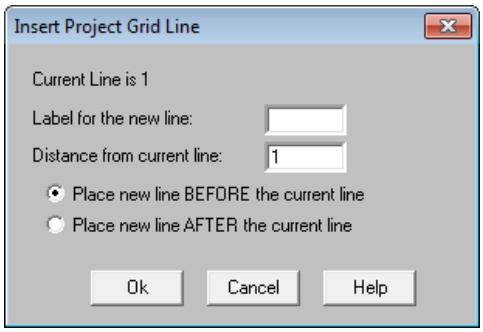


Figure 2-17 *The **Insert Project Grid Line** dialog box*

To delete a line from the spreadsheet, you can select the line and then choose the **Delete the Current line** tool or you can press F4 key. The **Fill the Currently Marked Block** tool is used to fill enteries in the cells of the spreadsheet. The **Mark All the Lines** tool is used to mark all the rows and columns in the spreadsheet. The **Delete the Currently Marked Lines** and **Unmark**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

All the Lines tools will be active only after selecting all the lines in the spreadsheet.

Drawing Grid

The drawing grid is yet another type of grid lines that can be created in RISA-3D. The main difference between project grid and drawing grid is that drawing grid is independent of the model and can be created in any view plane. You can modify the grid lines associated to a model anytime during modeling. Note that changing the grid lines will not change the model geometry. The drawing grid is available by default on opening a new project file. Figure 2-18 shows the default available grid pattern.

To define or modify a drawing grid, choose the **Modify the drawing grid and snap points** button from the Window toolbar. The **Drawing Grids** dialog box will be displayed, as shown in Figure 2-19. The **Drawing Grids** dialog box contains three tabs: **Drawing Grid**, **Snap To Options**, and **Import DXF**. These tabs and their options are discussed next.

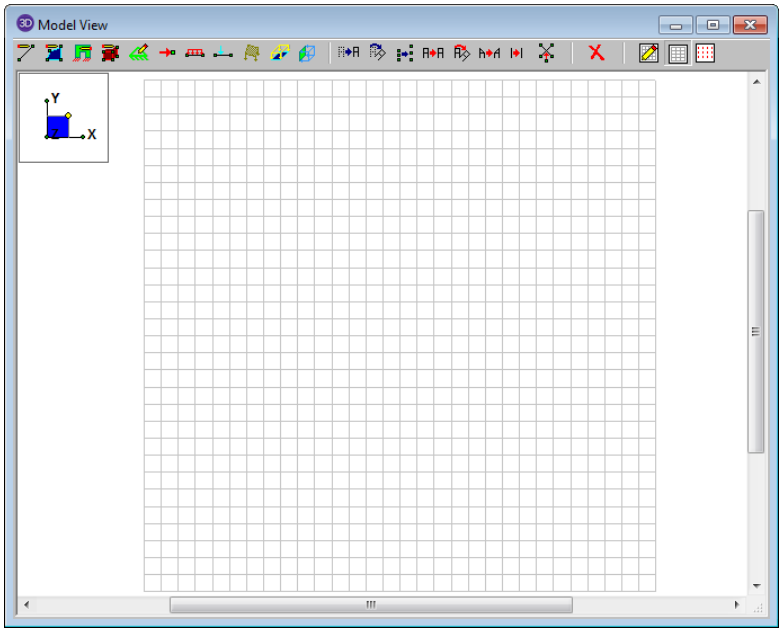
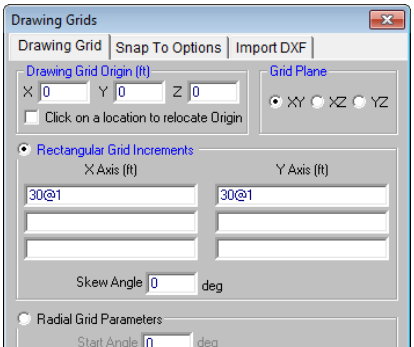


Figure 2-18 *The default drawing grid pattern*

Drawing Grids Tab

The **Drawing Grid** tab of the **Drawing Grids** dialog box contains six areas. These areas contain different options to generate or modify the grid lines, refer to Figure 2-19. The options in the **Drawing Grid Origin** area are used to set the origin of the grid line with respect to the global origin. To set the origin, enter the values in the **X**, **Y**, and **Z** edit boxes. You can also specify the location of the grid origin by clicking at any point in the drawing window. To do so, select the **Click on a point to relocate Origin** check box of the **Drawing Grid Origin** area. Next, choose the **OK** button



Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

and then click in the drawing window to specify the location. The options in the **Grid Plane** area are used to set the plane for placing the grid pattern. You can select the **XY**, **XZ**, or **YZ** radio button to set the grid pattern plane.

In RISA-3D, you can draw grid lines in rectangular and radial patterns. To draw rectangular grid lines, select the radio button corresponding to the **Rectangular Grid Increments** area if it is not selected by default. The options in the **Rectangular Grid Increments** area will be activated. The rectangular drawing grid is defined by increments in two directions. To define a rectangular grid pattern, you need to specify the number of grid lines and their intermediate distance. You can use the symbols such as (@), or (,), or (/) for defining the grid lines in a particular global direction. The use of these symbols is already discussed in the previous section of this chapter. You can also skew the rectangular grid by specifying the skew angle. The skew angle is the inclination of the rectangular grid from the global axis. To specify the angle of skew, enter a value in the **Skew Angle** edit box in the **Rectangular Grid Increments** area. Figure 2-20 shows default available rectangular grid pattern and Figure 2-21 shows the grid pattern after providing 135 degree skew angle.

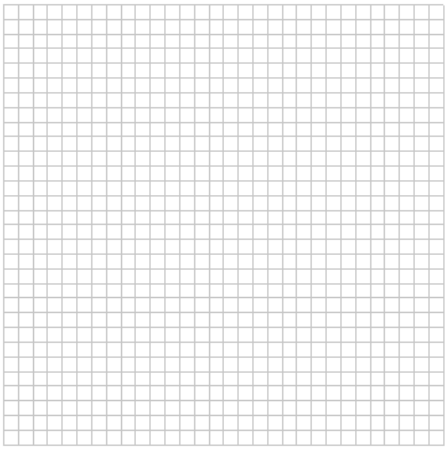


Figure 2-20 *The default rectangular drawing grid pattern*

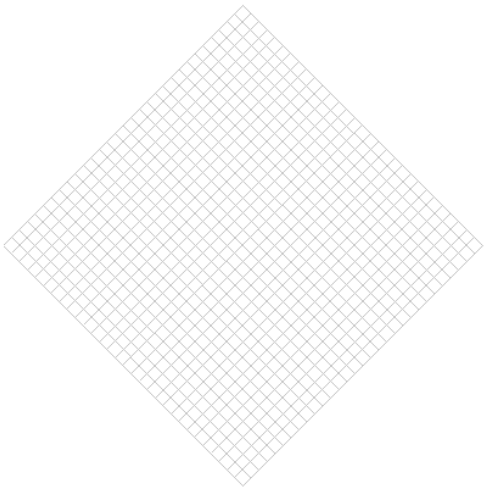


Figure 2-21 *The drawing grid pattern with 135 degree skew angle*

A radial grid pattern is defined by increments in two polar directions. To draw a radial grid, select the radio button corresponding to the **Radial Grid Parameters** area. The options in this area will be activated. Radial grids are defined by specifying the angular and radial increments. The value in the **Angle Increments** edit box controls the number and angular spacing of the spokes emerging from the origin. The value in the **Radial Increments** edit box is used to determine the number and spacing between the radial concentric grids. By default, the center point of the

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

radial grid pattern is its origin. The **Start Angle** is used to specify the angle of the first spoke from the global axis. The radial grid pattern is shown in Figure 2-22.

The **Save and Recall Grid Settings** area in the **Drawing Grids** dialog box contains options to save the grid pattern, retrieve a saved pattern, and delete them. You can also display the generated grid in the form of continuous lines or in the form of points. Figures 2-23 and 2-24 show the grid pattern represented by continuous lines and points respectively. To save the current grid settings as default grid setting for further sessions, select the **Save Current Settings as Defaults?** check box next to the **Show Grid As** area.

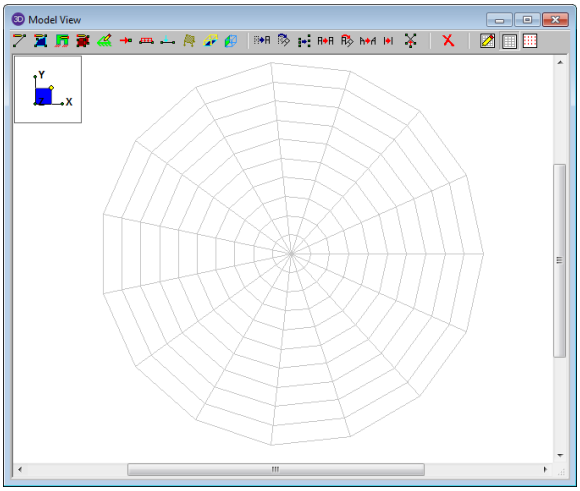


Figure 2-22 *The Radial grid pattern generated*

Figure 2-23 *The grid pattern represented by continuous lines*

Figure 2-24 *The grid pattern represented by points*

Snap To Options Tab

The **Snap To Options** tab contains the **Member Snap Locations** area and the **Universal Snap Increments** area. The settings in the **Snap To Options** tab help in sketching the model without using grids. In the **Member Snap Locations** area, you can select the **Quarter Points** or **Third**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Points check box to specify the automatic grid snapping with quarter or third point increment respectively. The **Universal Snap Increments** area contains options to draw grid line freely with any increment in the space. To enable universal snapping, select the **Use Universal Increments** check box in the **Universal Snap Increments** area. In RISA-3D, you can use this feature to set your own custom snap increments on the basis of the geometry of your model. To set the plane in which you want to draw grids, select the **XZ**, **XY**, or **YZ** radio button. For example, if you have set XY plane for drawing grids, you can set the snap incremental value in the **X Incremental** and **Y Incremental** edit boxes. The **Z Location** edit box defines the location of snap points perpendicular to the grid plane selected. For the XY plane selected, you can specify the snap location in Z direction.

Import DXF Tab

In RISA-3D, the **Import DXF** tab contains advanced options to import DXF drawing files as grid pattern. This imported grid pattern then acts as a grids providing snap points at their intersections. You can draw grid pattern using lines, polylines, circles, arcs, polylines with arcs, and points. In this tab, define the origin to which the imported file will be aligned. To define the origin, you can enter values in the **X**, **Y**, and **Z** edit boxes in the **Drawing Grid Origin (ft)** area. Alternatively, you can define the origin by clicking at the desired location. To define the origin, select the **Click on a location to relocate Origin** check box. You can also select the plane in which you want to place the grid pattern by selecting the **XY**, **XZ**, and **YZ** radio buttons available in the **Drawing Plane** area.

The **DXF Import Options** area contains options to change the scale factor, units of the DXF file, angle increment, and DXF plane. In this area, by default the value of DXF scale factor is 1. This implies that the objects in the DXF file will be imported on the scale of 1. For example, if an object is created on a scale of 1/2” = 12” in DXF file, RISA-3D will import it at the same scale factor of 1 by default. To import the object at full scale, enter 24 as the scale factor. Units of the imported DXF file can be changed from the **DXF Units** drop-down list in the **DXF Import Options** area. The value entered in the **Angle Increment** edit box specifies the number of degrees in which the imported arcs or circular objects should be broken into straight line segments. These segments allows you to snap points. You can enter the degree range between 0.1 to 10.

In the **Select a DXF file** area, you can select the recent file from the list. You can choose the **Browse For File** button to import an existing file. On choosing the **Browse For File** button, the **Select DXF File to Import for Drawing Grid** dialog box will be displayed. Browse to the desired file and choose the **Open** button; the **Select Layers for Drawing Grid** dialog box will be displayed. It has two radio buttons, the **Use All Layers** and **Select Layers From the List** to use all the default layers of the imported DXF file as grid pattern, select the **Use All Layers** radio button. To select Layer(s) from the list of layers, select the **Select Layers From the List** radio button. Choose the **Done** button; the objects in the DXF file will be imported as drawing grids.

SAVING A PROJECT

Before you close or exit a RISA-3D session, it is recommended to save the project file. You can save a project file in a permanent storage device, such as a hard disk or a removable storage device like CD or USB. Also, you must save your work at regular intervals to avoid data loss due to any error in the computer’s hardware or software.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Using the Save As Tool

RISA Menu:	File > Save As
Shortcut Keys:	CTRL+S

In RISA-3D, you can save your project file at the desired location. To do so, choose the **Save As** option from the **File** menu; the **Save RISA-3D Data File** dialog box will be displayed, as shown in Figure 2-25. Alternatively, you can save the project file by choosing the **Save the current model data to disk** button from the RISA toolbar.

In the **Save RISA-3D Data File** dialog box, the drop-down list at the top displays the current drive and path in which the project file will be saved. The **File name** edit box is used to specify the name of the file to be assigned to the project. The list on the left in this dialog box contains shortcuts for the folders that are frequently used.

In the **Save as type** drop-down, you can select the format in which you want to save your file. By default the files are saved in the **RISA-3D Data Files** format. Choose the **Save** button to save the file at the specified location.

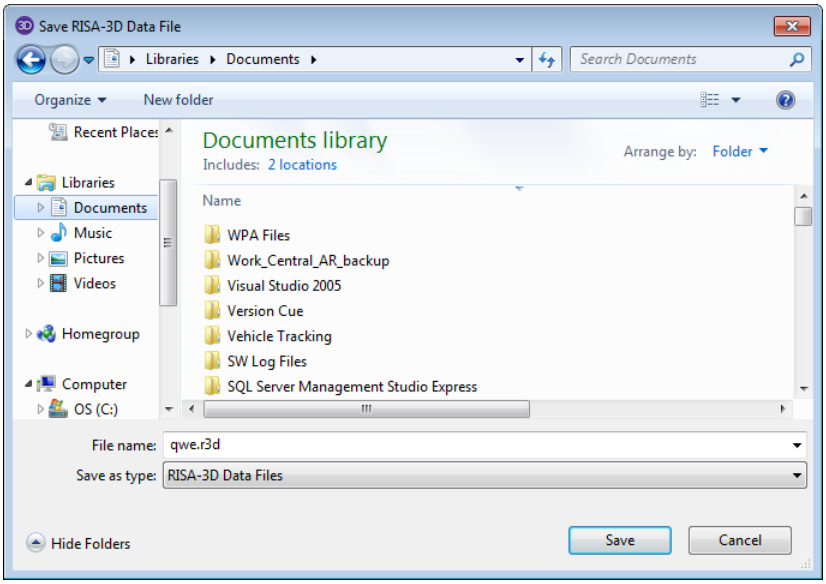


*Figure 2-25 The **Save RISA-3D Data File** dialog box*

EXITING A PROJECT

To exit a project, choose the **Exit** option from the **File** menu. If you have already saved the latest changes, the project file will be closed. Otherwise, RISA-3D will prompt you to save the changes through the **Check for Save** dialog box. You can save the changes by choosing the **Yes** button or discard them by choosing the **No** button. You can also choose the **Cancel** button to return to the interface and continue working on the project file. You can also use the **Close** button in the drawing to close the project.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

OPENING AN EXISTING PROJECT

RISA Menu:	File > Open
Shortcut Keys:	CTRL+O

In RISA-3D, there are many options available to open an existing project. These options are discussed next.

Opening an Existing Project Using the Open Tool

To open an existing project file, choose **File > Open** from RISA menu, as shown in Figure 2-26. Alternatively, you can open the project file by choosing the **Open** button from RISA toolbar or by pressing the CTRL+O keys. On invoking the **Open** tool, the **Open RISA-3D Data File** dialog box will be displayed, as shown in Figure 2-27. Using the drop-down list at the top of this dialog box, you can access the desired folder and open the desired file.

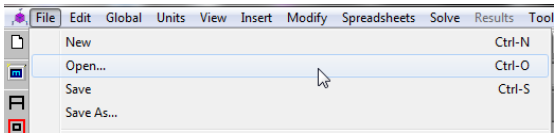


*Figure 2-26 Choosing the **Open** option from **File** menu*

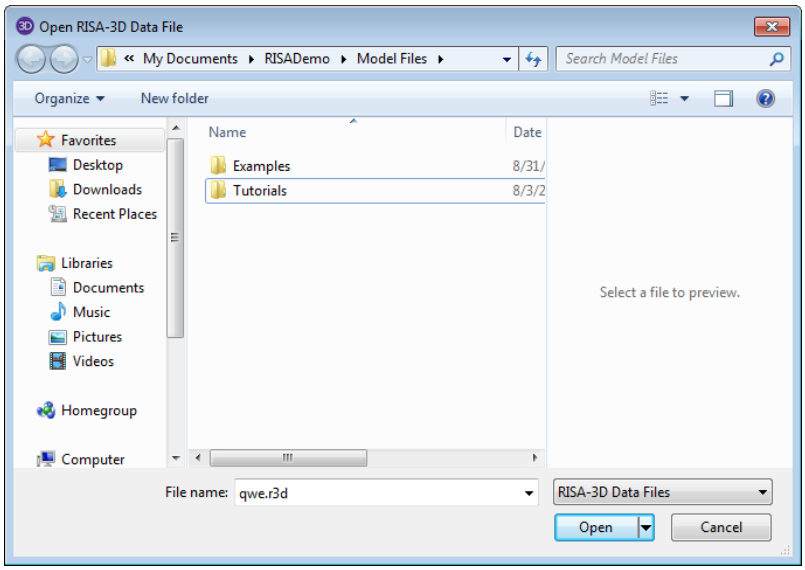


*Figure 2-27 The **Open RISA-3D Data File** dialog box*

The **Change your view** button located on the left of the **Show the preview pane** button, helps you select a project file based on its size, type, or date when it was last saved. The location list is located on the left side in the **Open RISA-3D Data File** dialog box. In this list, you can navigate to drives and folders available in your system. Choose the file located at the path or enter file name in the **File Name** edit box. Choose the **Open** button to open the desired file.

TUTORIALS

Tutorial 1 — Project Parameters

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

In this tutorial, you will create a new project file for a project with the following parameters and project specifications: **(Expected time: 30 min)**

1. Model Parameters

Model Title	**G+1 Concrete Framed Structure**
Company	**CADCIM Technologies**
Vertical Axis	**Y**
Concrete Code	**ACI 318-11**
Concrete Rebar Set	**ASTM 615**

2. Application Settings
3. Units System : Standard Metric
4. File name to be assigned: *c02_concrete_tut1.r3d*

The following steps are required to complete this tutorial:

a. Start a new session of RISA-3D.
b. Set the model parameters.
c. Define application settings.
d. Set the units for the project from the **Units Selection** dialog box.
e. Save the project.
f. Exit the project.

Starting a New Session of RISA-3D

1. Start RISA-3D by choosing **Start > All Programs > RISA-3D > RISA-3D 14.0**. On doing so, the RISA-3D interface window with an active **Starting a Model** dialog box is displayed, as shown in Figure 2-28.

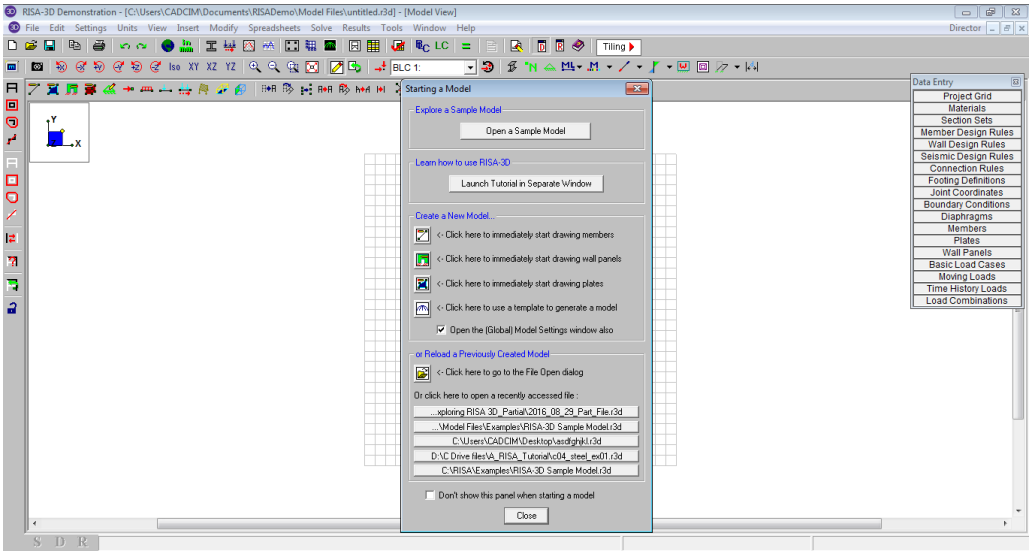


Figure 2-28 *RISA-3D interface displaying the* ***Starting a Model*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

2. Choose the **Close** button to close this dialog box.

Setting the Model Settings

1. Choose the **Settings** menu from the RISA-3D interface; the **(Global) Model Settings** dialog box is displayed.

2. In the **Description** tab, enter **G+1 Concrete Framed Structure** and **CADCIM Technologies** in the **Model Title** and **Company** edit boxes, respectively.

3. Choose the **Solution** tab. Next, select the **Y** radio button in the **Orientation** area it is not selected by default.

4. Choose the **Codes** tab and then select the **ACI 318-11** option from the **Concrete** drop-down list if not selected by default.

5. In the **Concrete Rebar Set** area of the **Concrete** tab, select the **ASTM A615** from the drop-down list if is not selected by default.

6. Choose the **Apply** and then the **OK** button; the **(Global) Model Settings** dialog box is closed.

Setting the Application Parameters

1. Choose the **Application Settings** option from the **Tools** menu; the **Application Settings (Preferences)** dialog box is displayed with the **General** tab choosen.

2. In the **General** tab, select the check boxes as represented in Figure 2-29.

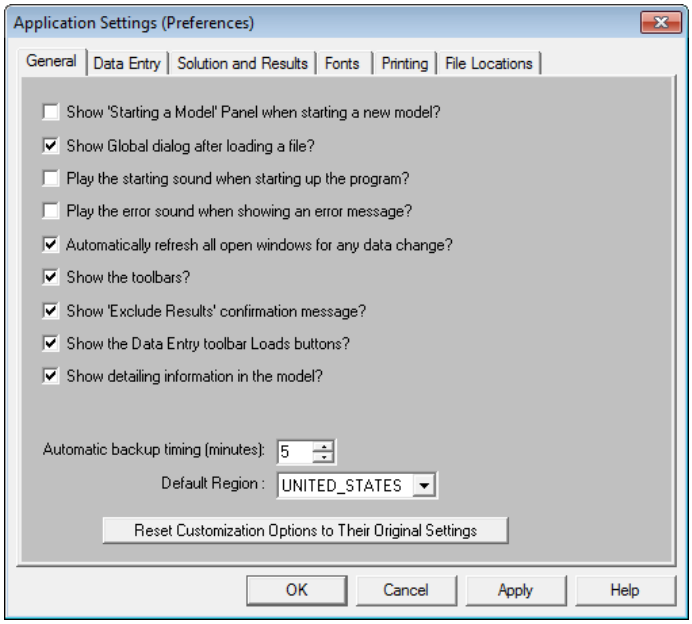


Figure 2-29 *The application settings in the* ***General*** *tab of the* ***Application Settings (Preferences)*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

3. Set the **Automatic backup timing (minutes)** spinner to **10**. Also, ensure that the **UNITED_STATES** option is selected in the **Default Region** drop-down list.

4. Choose the **Data Entry** tab and specify the values for the parameter as given below:

   Row height for data spreadsheets: **23**
   Decimal places for data entry fields: **3**

5. Ensure that the **Lock isolated ROTATIONAL instabilities without notification?**, **Warn if total reaction does not equal total load?**, and **Always warn before clearing results?** check boxes are selected in the **Solution and Results** tab.

6. Enter **21** in the **Row height for Results Browsers** edit box, or you can use the spinners to set the value. Ensure that **3** is specified as the value in the **Decimal Places** edit box.

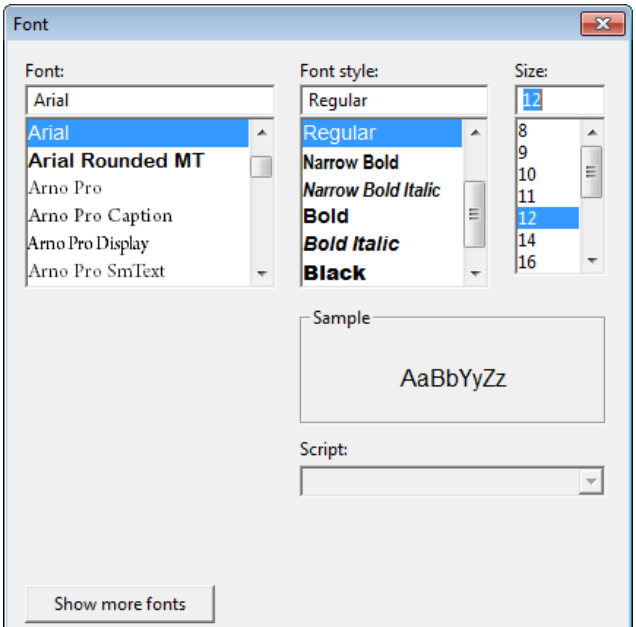
7. Select the **Batch results displayed by COMBINATION** radio button. Ensure that the **Ask before saving results with input data** radio button is selected in the **Saving Results (Static AND Dynamic)** area.

8. Choose the **Select Spreadsheet Font** button from the **Fonts** tab; the **Font** dialog box is displayed. In this dialog box, choose the **Arial** font option from the **Font** area, **Regular** in the **Font style** area, and **12** in the **Size** area, as shown in Figure 2-30. Choose the **OK** button to close the **Font** dialog box.

9. Choose the **Select Printed Report Font** button from the **Font** tab; the **Font** dialog box is displayed.

10. Specify the same settings as in step 8, refer to Figure 2-30 and then choose the **OK** button; the **Font** dialog box is closed.

11 Next, in the **Application Settings (Preferences)** dialog box, choose **Apply** and then **OK** buttons to save the changes and close the **Font** dialog box.



Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Setti

1. Cl

2. Cl metric fo

3. Se

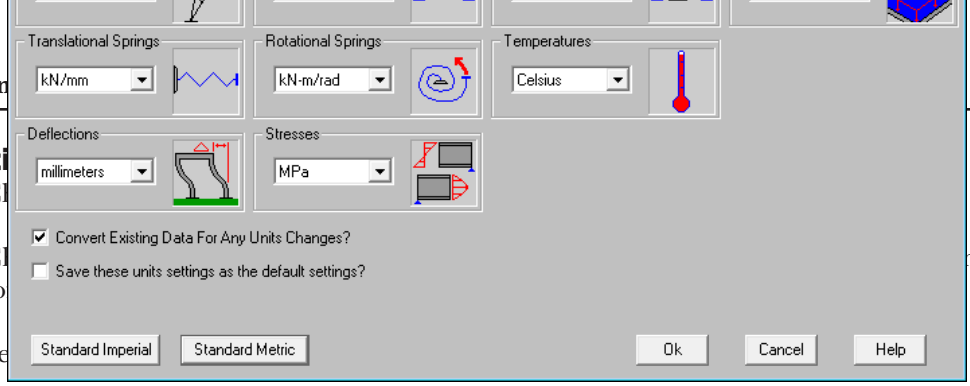


Figure 2-31 Settings units in the ***Units Selection*** *dialog box*

4. Ensure that the **Convert Existing Data For Any Units Changes** check box is selected. Choose the **OK** button to save the changes and close the **Units Selection** dialog box.

Saving the Project

The project parameters set in the previous steps are the global settings of the project file. To save this project file, use the **Save As** tool.

1. Choose the **Save** option from the **File** menu; the **Save RISA-3D Data File** dialog box is displayed.

2. In this dialog box, browse to the **C:** drive and then create a folder with the name **RISA_Tutorial**.

3. Open the *RISA_Tutorial* folder and then create a sub-folder with the name *c02*. Next, open the created folder and enter the file name as **c02_concrete_tut01**, refer to Figure 2-32.

4. Choose the **Save** button; the dialog box is closed and the file is saved.

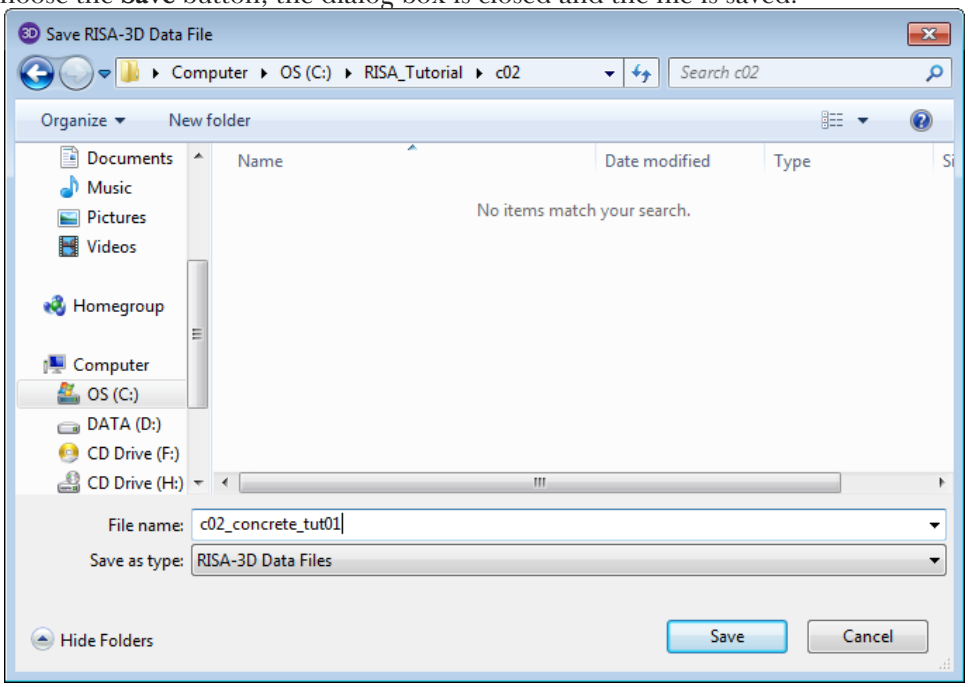


Figure 2-32 The ***Save RISA-3D Data File*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Exiting the Project

1. To exit the project, choose the **Exit** option from the **File** menu.

Tutorial 2 Grids

In this tutorial, you will create and set up the project grid for the *concrete* project created in the previous tutorial by using the following parameters: **(Expected time: 20 min)**

1. Project Grid
   Horizontal 2 spans at 5 meters each
   Vertical 1 span at 5 meters

The following steps are required to complete this tutorial:

a. Open the existing project.
b. Create the project grid.
c. Save the project.
d. Exit the project.

Opening the Existing Project

1. Start a new RISA-3D session and then choose **Open** from the **File** menu; the **Open RISA-3D Data File** dialog box is displayed.

2. Browse to *c:\RISA_Tutorial\c02* and select the file with the name *c02_concrete_tut1.r3d*.

3. Choose the **Open** button; the **Open RISA-3D Data File** dialog box is closed and the **(Global) Model Settings** dialog box is displayed.

4. Choose the **OK** button; the **(Global) Model Settings** dialog box is closed.

Creating the Project Grid

1. Choose the **Project Grid Spreadsheet** button from the RISA toolbar; the **Horizontal Project Grid Locations** dialog box is displayed.

2. In the **Z Axis** tab, make sure that the **Start Label** value is set to **1**, the **Start Location** value is set to **0**, and the **Label Order** is set to **A to Z** radio button.

3. Enter **2@5** in the **Increments** edit box and choose the **Generate** button; the horizontal grid entries are generated in the **Horizontal Project Grid Locations** dialog box.

4. Similarly, in the **X Axis** tab, make sure that the **Start Label** value is set to **A** and the **Start Location** value is set to **0**, and the **Label Order** is set to **A to Z** radio button.

5. Enter **1@5** in the **Increments** edit box and choose the **Generate** button; a vertical grid pattern is generated in the **Vertical Project Grid Locations** dialog box. Close the dialog box.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

6. View the project grid in top view by choosing the **Snap to an XZ Planar View** button from the Window toolbar. The project grid is visible, as shown in Figure 2-33.

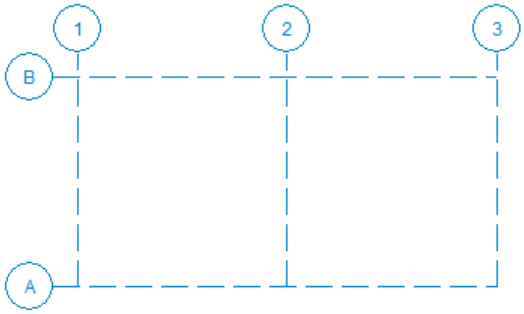


Figure 2-33 *Viewing the project grid in top view*

Saving the Project

1. Choose the **Save As** option from the **File** menu; the **Save RISA-3D Data File** dialog box is displayed.

2. In this dialog box, browse to the following location:

   C:\RISA_Tutorial\c02

3. In the **File name** edit box, enter **c02_concrete_tut02**.

4. Choose the **Save** button; the file is saved with the name *c02_concrete_tut02.r3d* at the specified location.

Exiting the Project

1. To exit the project, choose the **Exit** option from the **File** menu.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. Which of the following options is used to move elements of the given model simply by moving a grid line?

   a) Drawing grid
   b) Project grid
   c) Both the drawing and project grids
   d) None

2. Which of the following toolbars helps you to select and unselect items in the given model?

   a) RISA toolbar
   b) Window toolbar
   c) Selection toolbar
   d) Drawing toolbar

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

3. In RISA-3D, the _____ toolbar can be customized.

4. The use of the _____ grid is limited to the model plan.

5. In a RISA-3D session, you can open only one structural project at a time. (T/F)

6. In RISA-3D, you cannot change the font styles used in the spreadsheets. (T/F)

7. You can customize the RISA toolbar by double-clicking on the toolbar. (T/F)

8. A project file can be opened by double-clicking on the file name in the Windows Explorer. (T/F)

Review Questions

Answer the following questions:

1. Which of the following file formats can be imported as grid in RISA-3D?

   a) *.dwg b) *.dxf
   c) *.r3d d) None

2. The _____ dialog box is displayed when a RISA-3D session is initiated.

3. You can modify the project unit settings anytime during a project. (T/F)

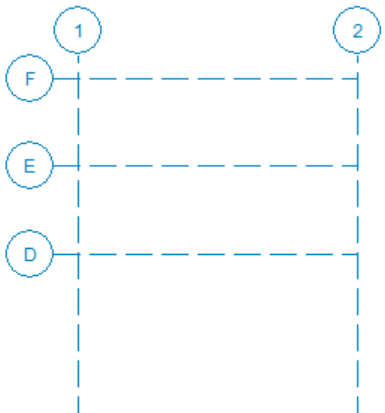
4. The name and path of a project file should be specified each time you save the project. (T/F)

5. If you choose the **Exit** button without saving the changes made in a project file, RISA-3D will prompt you to save the changes before getting closed. (T/F).

EXERCISE

Exercise 1

In this exercise, you will create a new project file specifying the following parameters. Figure 2-34 shows the project grid in top view. **(Expected time: 20 min)**



Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

1. Model Settings :
   - Model Title **Steel Roof Truss**
   - Company **CADCIM Technologies**
2. Units System : Standard Metric
3. Project grid :
   - X-axis **2@2.5**, **1@6**, and **2@2.5**.
   - Z-axis **1@8**.
4. File name to be assigned: *c02_steel_ex1.r3d*

Answers to Self-Evaluation Test

1. b, **2. c**, **3. Window**, **4. Project**, **5.** T, **6.** F, **7.** F, **8.** T

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com