The RAM Structural System V8*i*

SELECTseries 5™

RAM Manager™

Release 14.05
July 2012

Bentley Systems, Inc.
2744 Loker Avenue West, Suite 103
Carlsbad, CA 92010
Telephone: (760) 431-3610
Toll Free: (800) 726-7789
Fax: (760) 431-5214

DAA036830-1/0012

DISCLAIMER

RAM Manager

Table of Contents

1 Introduction

The RAM Structural System is powerful and versatile special purpose software for the analysis and design of building structures. It is useful in the design and analysis of commercial, institutional and industrial buildings. The RAM Structural System automates the process of calculating tributary loads, live load reduction, gravity member selection, frame analysis, drift control, frame member and joint code checking, special seismic provisions member and joint checking and foundation design. By automating these tedious and time consuming processes, the engineer can quickly obtain an accurate design. Different framing configurations may be examined in a short period of time, resulting in substantial timesaving for the Engineer and a more economical design for the client. The interface with CAD software permits rapid generation of framing plans, saving significant drafting time and reducing the errors associated with manual information transfer.

The RAM Structural System is composed of a number of special purpose modules, each of which is invoked from the RAM Manager. The RAM Modeler provides for the creation of a model of the entire structure, including roof and floor loads; beam, column, brace and wall geometry and locations; and slab properties, openings and edges. Powerful, yet easy to use graphical model generation features are provided which allow complex floor and building systems to be modeled in a short time. The result is a comprehensive database of building data which can be accessed by the analysis and design modules, providing a completely integrated solution.

The RAM Steel Beam Design module provides a powerful capability for the gravity design of composite and noncomposite beams and girders as well as the selection of Steel Joists and CMC SMARTBEAM and Westok Cellular beams. In addition to the automated optimization of beam sizes, existing conditions can be checked. Tributary loads from the user-defined surface, line and point load patterns, loads on girders due to beams which frame into them, live load reduction factors based on one of several available building codes, and effective flange width are all automatically calculated. Special design considerations, such as depth restrictions, can be specified. Designs can be performed using one of several included steel design codes.

The RAM Steel Column Design module provides a powerful capability for the design of gravity columns and their baseplates. Axial loads, unbalanced moments, live load reductions and bracing conditions are automatically calculated. Optimum sizes may be obtained or existing conditions analyzed.

The RAM Concrete Design module provides a powerful capability for the analysis and design of concrete shear walls and gravity and frame columns and beams. Design loads, including live load reductions and pattern loading, are automatically calculated. Effective flange width of T-beams is automatically calculated. Reinforcement is automatically selected. Optimum designs may be obtained or existing conditions analyzed.

RAM Frame provides the capability to perform a full three-dimensional static and dynamic frame analysis of the lateral system in the structure. Member locations and geometry, gravity loads with their corresponding live load reduction factors and story mass properties are obtained directly from the database. Lateral wind and seismic loads may be generated based on Building Code requirements or specified as user-defined story or nodal loads. In the Analysis mode frames of any material and type, including moment frames, braced frames and walls can be analyzed. In Steel mode a code check based on a selected steel design code can be performed for all lateral steel members and moment frame joints. The seismic provisions of the American Institute of Steel Construction and of the Uniform Building Code and can also be checked

for steel members and frame joints. In the Drift Control Mode the lateral members of the structure can be analyzed for drift participation and the member sized changed to efficiently control drift.

RAM Foundation provides the capability to design all spread footings, continuous footings and pile caps located beneath both lateral and gravity members. Gravity member reactions can be taken from the RAM Steel Gravity Column module, the RAM Concrete Analysis module or a combination of the two based on the material type of the supported member. Lateral member reactions are taken from the RAM Frame analysis. The gravity and lateral reactions are combined in RAM Foundation to automatically produce the foundation forces necessary to accurately design concrete footings. Spread footings, continuous footings and pile caps can be designed and optimized based on the American Concrete Institute (ACI-318) and British (BS 8110) Building Code Requirements.

1.1 Using the Documentation

The documentation is divided into several distinct manuals, one for each individual module: RAM Manager, RAM Modeler, RAM Steel Beam Design, RAM Steel Column Design, RAM Concrete, RAM Frame and RAM Foundation. In addition there is an Installation manual and a Tutorial manual. The Tutorial example provides a step-by-step guide demonstrating how to create a model, analyze and design the elements in the various design modules and obtain the various outputs. The manual for each module describes each function of the program in greater detail. A technical section is included in the manuals for each of the analysis and design modules which explains engineering related issues such as Building Code interpretation and implementation, assumptions, etc. The user should become thoroughly familiar with the documentation.

The purpose of these manuals is to explain the assumptions and methodology of the RAM Structural System. Every effort has been made to include a discussion of significant decisions and assumptions made by the program. Generally, if there is any question as to how the program handles any particular condition, a small model can be quickly created and analyzed, and the results verified with the appropriate hand calculations.

In the design of a structure a great number of decisions must be made. What is acceptable to one engineer may not be acceptable to another. It is crucial that the user understands the decisions and assumptions being made by the program. If these are not appropriate for the specific conditions of a particular building, the user should augment or replace the results from the program with those of some other tool.

The RAM Structural System has been extensively tested and used. It is impossible, however, to anticipate every possible configuration that could be encountered by the program. Ultimately the engineer is responsible for the safety and adequacy of the design of the building.

Do not hesitate to contact Bentley/RAM Technical Support with any questions that arise. The staff is always willing to spend time answering questions to assure each user understands and utilizes the RAM Structural System to its fullest potential.

This manual describes the function and capabilities of the RAM Manager, and its relationship to the various design modules.

Chapter 2 addresses the use of the RAM Manager including creating a new database and accessing the analysis and design modules. A discussion on creating DXF files of floor framing plans and column and footing schedules is also included.

Chapter 3 describes the Model Status feature. Elements of this feature are available in all of the modules.

Chapter 4 provides an in-depth look at the Reports that can be produced using the Post-Processing menu option in the RAM Manager.

Chapter 5 addresses several options for customizing the RAM Structural System working environment.

Chapter 6 takes an in-depth look at customizing the RAM Structural System through creating new member tables or modifying the existing ones.

2 The RAM Manager

The RAM Manager is the gateway to the RAM Structural System Software. It contains commands that serve five main functions: creating, selecting and deleting databases, specifying design criteria, invoking the RAM Modeler for creation or revision of models, invoking the Analysis and Design modules, and invoking Post-Processing commands. These functions are described in this chapter essentially in the order they appear in the RAM Manager menu. This is also, generally, the order followed when using the Structural System to analyze and/or design a structure: create a database, specify design criteria, generate a model, invoke the analysis and/or design modules, and obtain output.

2.1 Invoking the RAM Manager

The RAM Manager is invoked by double clicking on the RAM Structural System icon or by selecting **Programs – Bentley Engineering - RAM Structural System – RAM Structural System** from the Windows **Start** menu.

Upon initiating the RAM Manager, the program window appears displaying the RAM International Logo, a menu bar with a series of menu commands, a tool bar and an RSS (Really Simple Syndication) feed view pane. Selecting one or more of the commands from this menu or tool bar performs the functions described in the remainder of this chapter.

2.1.1 Tip of the day

A ‘Tip of the Day’ feature has been added to RAM Manager. The tip dialog will display each time RAM Manager is started but may be turned off by unchecking the “Show Tips on StartUp” checkbox in the dialog. The tip dialog may also be launched at any time by selecting “Tip of the Day…” from the “Help” menu.

In general, the ‘tip’ will highlight new features added to any of the RAM Structural System Modules in the most recent releases. Some ‘tips’ will also present not commonly known features that can help the engineer to be more productive.

2.1.2 RSS Feed

The RAM Manager has a view pane in the bottom portion of the main screen. This pane displays items from a custom Bentley Really Simple Syndication (RSS) feed. The feed items are used to provide timely information on new updates, releases and general product news. The intent of this feature is to get timely and relevant information to our users. The items in the list may be updated at any time so they should be checked regularly for new announcements.

Clicking on a feed item will launch the default browser and load the associated web page (link), if an internet connection is available on the computer. If the RSS feed is unavailable, or if there is no internet connection, a single feed item will be listed, with a link to the RAM Structural System web page on the Bentley website (the link won’t be active if there is no internet connection available). If this is consistently the only Feed displayed, important announcements and information are being missed.

Items in the feed are icon coded. Icons have the following general meanings:

 - Critical messages, patches, new releases of the RAM Structural System

 - Notification of available downloads, releases of related products

 - Messages from Bentley not specifically related to RAM products

 - Educational announcements, upcoming seminars, webinars, etc.

 - Everything else, general important information

Some feed items may be specific to certain regions. Those items may be filtered out by de-selecting the appropriate options in the **Tools – RSS Feeds Options** command. The general items ("Worldwide") and those in the region corresponding to the location of the user's computer cannot be filtered out.

The RSS items pane may be resized and will automatically show scroll bars if more items are listed than can be displayed in the current pane size.

2.2 Database Operations

The RAM Structural System Software utilizes a database format for storing data. Each model has its own database, composed of several files. When starting a new project, a database is created and a name is associated with it. Once the database has been created, all of the data for a given model can be accessed through that name.

To improve the speed and manageability of RAM Data files two features have been implemented. First, the numerous RAM Database files have been consolidated into a single compressed .rss file. Second, all models are run from a 'Working' folder rather than the 'Data' folder. If it hasn't already been specified, the 'Working' folder will be specified by the user the first time any model is opened from RAM Manager. ***This working folder should be located on the local drive even if the data files (.rss) themselves are stored on the network, and it should not be the same as the 'Data' directory (it can be a subdirectory of the 'Data' directory).*** It is a temporary directory and should not be used for any other purpose. When a database is opened in RAM Manager it is copied from the Data directory (or whatever directory it is stored in) and uncompressed in the local 'Working' folder. All model changes are saved to the local working files. When a Save is performed, the files are compressed and stored back in the .rss file in their original location. This feature allows databases to be stored in a network folder, but to be worked on locally.

2.2.1 Creating a New Database

To create a new database:
Select the **File – New** command.

A dialog box will appear prompting for the name of the database to be created, a job description, and the system of units (English, SI, or Metric) to be used.

The Units selected should be those desired on the final output reports. Units may be changed at any time while working with a model, but an initial system of units must be specified so that correct model defaults can be selected. Some of the default values that are dependent upon the system of units selected include material properties such as steel Fy or concrete f'c and DXF drawing information defining grid extensions and bubble size.

Clicking the **OK** button causes the new database to be created in the Data directory (or the directory specified). Entering a database name that already exists in the Data directory will result in a warning message being issued.

The system administrator specifies the data directory and units when installing the RAM Structural System Software. Section 2.4.10 of this manual provides information on modifying the Units on a model-by-model basis. Additional information on modifying program defaults, such as Units, is available in Chapter 5 Customizing the RAM Structural System and in the on-line help.

2.2.2 Opening an Existing Database

To open an existing database:
Select the **File – Open** command.

A dialog box will appear, containing a list of all existing databases in the current Data directory. A database may be opened by either of the following methods:

- Click on a database name in the list box and then click the OK button.
- Double click on a database name in the list box.
- Type in the name of the database in the File Name edit box.

To modify the job description, click on a database name in the list box and edit the description before clicking the **OK** button.

For information on opening a database that was not properly saved due to an abnormal program termination, see Section 2.2.4 Saving a Database.

2.2.3 Multi-User File Prevention

The program prevents two or more users from editing the same database simultaneously. When a user attempts to open a database that is currently being edited, a dialog listing the current "owner" is shown (see Figure 2-1 below). The database is not available to other users until the current owner exits the database.

Note that this dialog may erroneously be displayed if, during the previous editing session with this database, the program abnormally terminated ("crashed"). When this occurs a temporary file used by the program to determine whether or not the database is currently being edited will not be properly deleted, and any attempts by any other user to access this database will result in this dialog being displayed, even though the

database is not, in fact, currently being edited by the original owner. This problem can be resolved by having the original owner open, then close, the database. If the original owner is not available to open and close the model, the “modelname~.usr” file (where “modelname” is the name of the database) may be deleted from the data folder to allow another user to open the model. Only manually delete the “modelname~.usr” file if the owning user is not available and is not currently editing the file. Warning: if one user is currently editing the database and the “modelname~.usr” file is manually deleted, a second user can then simultaneously access the database, resulting in data corruption. Only delete that file if no users are currently editing that database.

The following shows an example of the dialog that is displayed if the model is currently opened by a user whose login name is “Gabe” and a different user attempts to open the same model:

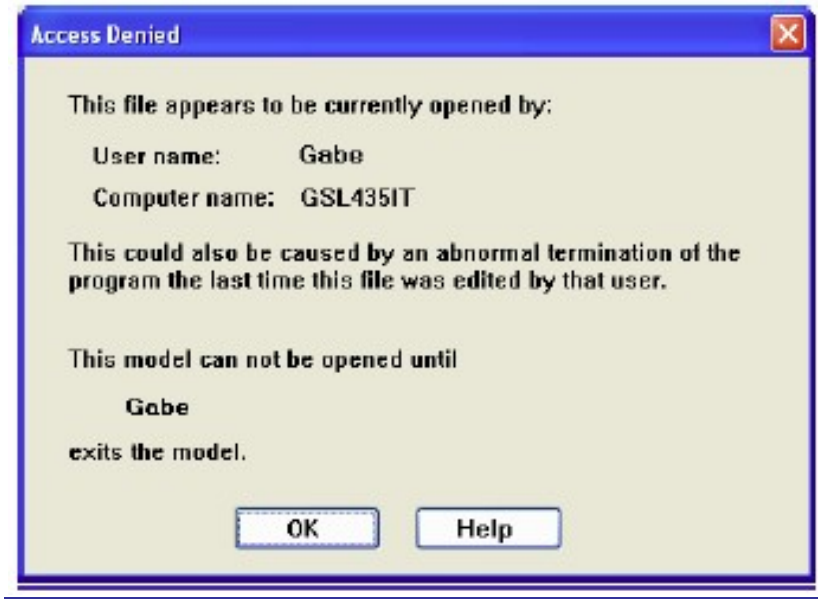


Figure 2-1

2.2.4 Saving a Database

To save a database under the existing name:
Select the **File – Save** command.

To save a database under a new name: Select the **File – Save As** command.

Each module has a **File – Save** command allowing the user to save the current database. It is not necessary to invoke the Save command when going from one module to another. Any changes made to Criteria or assignments, or any analyses or designs performed are only saved temporarily until the Save command is invoked. This allows the user to work with the database, saving or discarding changes or results as desired. The RAM Manager requires that the Save command be invoked prior to exiting the RAM Structural System or prior to opening another database, otherwise the changes made since the last Save will be lost. The other modules do not require that the Save command be invoked before exiting that module and going to another module. It is recommended that the Save command be invoked periodically, especially when exiting the Modeler.

If it is desired to discard any modifications or changes made to a database since the most recent Save, invoke the **File – Revert** command (see Section 2.2.5) or exit the RAM Structural System without saving the data. Opening a different database without saving will also cause any changes to be discarded.

There is no explicit command to Copy a database, but this can be accomplished by opening the database, invoking **File – Save As** and specifying the new name and/or directory.

Issuing the **File – Exit** or **File – Open** commands before the current database has been saved will cause a message to be given warning the user that changes have been made since the last Save was invoked, and gives the user a chance to save work before exiting. Select **Yes** to save the changes to the database, **No** to discard the changes, or **Cancel** to continue with the current database.

If the program crashes or otherwise abnormally terminates at any time before the database can be properly saved, a message will be given the next time that database is opened indicating that a temporary backup file for that database has been found. The backup file contains the database as it existed at the last Save, before the most recent changes were made. The user is given the option to either open the database as it occurred at or just previous to the time that the program terminated (using the **Most Recent Database** option) or to open the backup database which contains the database as it existed at the time of the last Save (using the **Backup Database** option). The user is also given the option to cancel opening either one.

If the **Backup Database** option is selected, any changes made since the last proper Save will be lost.

If the **Most Recent Database** option is selected, the user should carefully inspect the model. The most recent database contains all or most of the changes since the last Save, but it may also contain whatever data errors or corruption that may have caused the program to terminate. If the data is corrupted, exit without saving. This will cause the most recent changes to be lost and the backup database to be restored to the database (the same as if the **Backup Database** option had been selected initially).

Alternatively, select the **Most Recent Database** option and then save to a different name using the **File – Save As** command. By doing this, both versions of the database will be available for further inspection or use if necessary.

2.2.5 Discarding Changes

There is no corresponding toolbar button for this command.

To revert a database back to its state at the previous save:
Select the **File – Revert** command.

Any modeling done in the Modeler, any changes made to Criteria or assignments, or any analyses or designs performed are only saved temporarily until the Save command is invoked. This allows the user to work with the database, saving or discarding changes or results as desired. Such changes can be discarded by using the **File – Revert** command. This command discards all changes and actions taken since the last Save command was invoked, returning the database to that former state.

A dialog box will appear showing the file name and the date and time of the most recent Save, and prompting the user to verify that the Revert command is to be performed. Select **OK** to Revert to the previous state, or select **Cancel** to continue with the current state of the database.

2.2.6 Deleting a Database

To delete a database:
Select the **File – Delete** command.

A dialog box will appear, containing a list of all existing databases in the current Data directory. A database may be deleted by either of the following methods:

- Click on a database name in the list box and then click the Delete button.
- Double click on a database name in the list box.

After selecting a database name, a warning will appear stating that the entire database will be deleted. To continue the deletion, select **OK**. To cancel the deletion, select **Cancel**.

2.2.7 Archive/Zip a Model

There is no corresponding toolbar button for this command.
The engineer has the ability to automatically compress all the files of the current open database into a single .zip file. This file can be used for archival purposes or when a model needs to be transferred electronically. It is not necessary to first Save the database, this command will archive the database in its current state. To compress all files of the model into one .zip file:

Select the **File – ZIP Model** command.

Upon issuing the **File – Zip Model** command on a model that has been analyzed or designed, a dialog providing the option to exclude results file from the archive will be shown. For an analyzed model there may be many large analysis result files in the database. These files can typically be regenerated by issuing an analyze command in the appropriate program. To reduce the size of the zip file the user is given a choice as to which results files to include in the zip file.

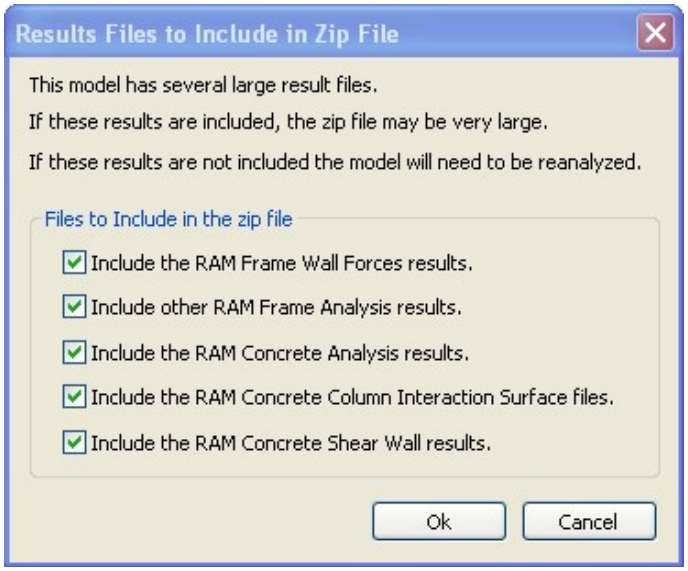


Figure 2-2

Upon clicking OK, the Save-As dialog is presented.

When the command is completed, the database will remain open and available to continue editing.

2.3 Information about the Database

2.3.1 Model Status

To view the current status of the database
Select the **File – Model Status** command

The **File – Model Status** command provides information about the status of the current model. This command lists the current design status for each module, critical information available to the module and, in cases where a model cannot be run in the module, the missing information. This feature is common to all of the modules in the RAM Structural System.

For more information, see Chapter 3, Model Status.

2.3.2 Model Notes

To view or edit model notes:
Select the **File – Model Notes** command

The Model Notes command opens a text file that may be used for entering any notes that the user wishes to keep on the currently loaded model. The model notes file is stored in the same directory as the model's files and will have the format 'modelname.txt'.

2.3.3 3-D View

To view the model in 3D:
Select the **View – 3D View** command

Issuing the **View – 3D View** command initiates the 3D Viewer.

For more information about the 3D View, see the 3D Viewer Manual.

2.4 Database Criteria

The user can specify several criteria in the RAM Manager. These are criteria that affect the overall model. Criteria that are specific to Steel Beam Design (stud criteria, camber, etc.), Steel Column Design (bracing, etc.), Concrete Design (reinforcement selection, etc.), Frame Analysis (rigid end zones, P-delta, etc.) or Foundation Design (reinforcing, soil data, etc.) are located within their respective modules.

2.4.1 Member Loads Criteria

A building code by which Live Load Reduction will be calculated must be selected. The program also needs to know how to treat Roof Live Load versus Snow load, and how to count accumulated stories for the calculation of the column live load reduction. The system administrator can define the default options at the time of installation. To modify or inspect the current options for the selected model:

- Select the **Criteria – Member Loads** command.

A dialog box will appear showing the current Building Code, Roof Live Load designation and Number of Stories selections. To modify these selections, make the desired changes and click the **OK** button. If no changes are desired, click the **Cancel** button.

If the Code selections are modified after one of the Design Modules has been run, optimized member designs will automatically be removed from the database and the 'framing tables' will be rebuilt based on the newly specified codes. A warning message to this effect will be posted providing the option of canceling the change.

See the Technical Notes chapter in each Design Module manual for a description of how the Live Load Reduction, Snow Load and Roof Live Load criteria are used by each module.

When the program calculates the loads on beams it calculates the load for each segment along the beam and on both sides of the beam. To simplify the output, these segments of loads are combined when possible into a single load on the beam. This is the default behavior of the program, but an option is available, Combine Beam Loads, that can be deselected to prevent the program from combining these load segments. This will have no impact on designs but will result in a fuller listing of member loads which can aid in the verification of those member loads. In general it is recommended that the option to combine the loads be selected, to simplify and shorten the listing of gravity loads in the reports.

Some terminology used in the Modeler is dependent upon the Live Load Reduction Code Selection. For example, the yield strength of steel will be labeled in the Modeler as Fy, fy, or py, depending on which Code is selected here.

2.4.2 Self-Weight and Self-Mass

The RAM Structural System offers the automatic calculation of self-weight and self-mass for beams, columns, walls and slabs. The system administrator defines the default option selections at the time of installation. To modify or review these criteria

- Select the **Criteria – Self Weight** command.

A dialog will appear showing the current self-weight and self-mass selections. The automatic calculation and inclusion of self-weight and self-mass may be specified separately, and furthermore the calculation and inclusion for Columns, Beams, Walls and Slab/Deck may be specified separately. To change the current settings for beams, columns, walls or slabs, select the associated check box. The boxes will display a check mark when self-weight is to be included for the member type. For columns and walls there are options to either include all of the mass of the wall or column below with the mass of the story or to split the mass of the wall or column between the two adjacent levels, half up and half down.

To modify these selections, make the desired changes and click the **OK** button. If no changes are desired, click the **Cancel** button.

If the Self-Weight selections are modified after one of the Design Modules has been run, optimized member designs will automatically be removed from the database and the 'framing tables' will be rebuilt based on the newly specified selections. A warning message to this effect will be posted providing the option of canceling the change.

See the Technical Notes chapter in each Design Module manual for a description of how the Self-Weight criteria affect each module.

Important Note: When a database created in a version previous to V7.2 is converted to the recent version, the self-weight options for Columns and Walls is selected, but the self-weight option for Beams is not. This is to maintain consistency of the analysis with previous versions, but may be modified by the user.

2.4.3 Selecting a Master Steel Table

A Master Steel table that will be accessed when assigning member sizes in the RAM Modeler or when designing steel members in the Design modules must be specified. Master Steel Tables must contain every steel size to be used by all modules and their associated properties. See the RAM Structural System Tables chapter for more information.

The system administrator makes the default Master Steel Table designation at the time of installation. To select a different Master Steel table:

- Select the **Criteria - Master Steel Table** command.

A dialog box showing the current table selection will appear. To modify the selections, make the desired change and click the OK button. If no change is desired, click the Cancel button.

The engineer can also choose to create a master table that will be stored with the model data. If there is a master table file with the same name as the model and file extension .tab then the master table dialog will change in appearance and the modelname.tab file can be selected to be used by the engineer. To create or edit a master table select the **Tools - Steel Table Editor** command.

If the table selections are modified after one of the Design Modules has been run, optimized member designs will automatically be removed from the database. A warning message to this effect will be posted providing the option of canceling the change.

Users of the RAM Steel Beam Design module and/or the RAM Steel Column Design module must use caution when making changes to the Master Steel Table selection. When a Master Steel Table selection is modified, the Design Steel Tables selections for beams and columns must also be modified to correspond to the selected Master Steel Table. This is due to the fact that the Design Tables refer to the corresponding Master Tables for section property information. If this is not done, error messages will be encountered when the design module attempts to optimize a member. No design will be created.

2.4.4 Selecting Design Steel Tables

A Deck table from which deck assignments will be made within the RAM Modeler must be specified and Design Steel Tables from which steel member sizes will be selected in the RAM Steel Beam and Column Design modules must be specified.

The system administrator makes default table designations at the time of installation. To change the Deck or Steel Member table selections:

- Select the **Criteria - Design Steel Table** command.

A dialog showing the current table selections will appear. For Beams, Steel Joists and SMARTBEAM, a default table and an alternate table selection is made. During the design process, member sizes are selected from the default tables unless otherwise specified in the RAM Modeler. For more information on specifying Default and Alternate tables on a member-by-member basis, see the RAM Modeler manual.

To modify these selections, make the desired changes and click the OK button. If no changes are desired, click the Cancel button.

If the table selections are modified after one of the Design Modules have been run, optimized steel member designs will automatically be removed from the database. A warning message to this effect will be posted providing the option of canceling the change.

Licensees of the RAM Steel Beam Design module and/or Column Design module must use caution when making changes to the Design Steel Tables selections. When a Design Steel Table selection is modified, the Master Steel Table selection may also need to be modified to correspond to the Design Steel Tables selected. If this is not done, error messages will be encountered when the design module attempts to optimize a member using sizes from the Design Steel Table that are not included in the Master Steel Table. No design will be created.

Deck tables and Steel tables can be modified or new tables can be created to meet the specific needs of the engineer. For further explanations on customizing these tables, see Chapter 6.

2.4.5 Selecting Concrete Design Tables

A reinforcement table must be specified for the concrete design modules, RAM Concrete and RAM Foundation. If Pan joists and beams are to be modeled, a Pan Form table must also be specified. The system administrator specifies default values at the time of installation. To change the Reinforcement Table or the Pan Form Table selection:

- Select the **Criteria** – **Concrete Tables** command.

A dialog showing the current table selections will appear. To modify these selections, make the desired changes and click the OK button. If no changes are desired, click the Cancel button.

If the table selections are modified after one of the Design Modules have been run, optimized concrete member designs will automatically be removed from the database.

2.4.6 Eurocode Factors

EC A number of factors used in the Eurocode may vary from country to country. Their values are specified in the individual country's National Annex (NA). The user may specify these values as appropriate. The system administrator specifies default values at the time of installation. To change Eurocode factors:

- Select the **Criteria - Eurocode Factors** command.

A dialog box will appear showing the current Eurocode factor values. To modify these values, make the desired changes and click the **OK** button. If no changes are desired, click the **Cancel** button.

If the factors are modified after one of the Design Modules has been run, optimized member designs will automatically be removed from the database and the 'framing tables' will be rebuilt based on the newly specified values. A warning message to this effect will be posted providing the option of canceling the change.

2.4.7 Canada Parameters

CN

Deflection:

In the determination of the deflection of Composite beams, CAN/CSA-S16-01 requires that the effects of creep and shrinkage be considered. For creep, the post-composite Dead and long-term Live Loads are multiplied by 1.15. The user specifies the portion of the total Live Load to be considered long-term.

For shrinkage, the modular ratio, nt, must be specified. The modular ratio nt equals E/Ect where Ect is the effective modulus of concrete in tension. The value normally ranges from 40 to 60. A percentage of the resulting shrinkage deflection will be added to the Live Load, as specified by the user; the remainder will be added to the Dead Load.

These parameters only affect composite beam design in RAM Steel Beam Design.

Steel Material:

The steel material type (based on Table 6-3 from CSA G40.21) for each type of structural member must be selected. This steel type is combined with the nominal yield strength assigned to each individual member to determine the steel grade and the design yield strength of the section. For example, a section of type W with a nominal Fy of 350N/mm2 is assigned a steel grade of 350W. A nominal Fy of slightly less than 350 will result in a steel grade of 300W. Specifying a nominal Fy and material type that has no matching steel grade will result in a design yield strength of 0.0 and no grade assignment. A steel of unknown grade should be designated as a '?' (unknown) steel grade and the program will use the assigned nominal Fy value for the design Fy.

Built-Up:

Flange sections can be designated as being built of Universal Milled Plate or of Flame Cut plate. The type of plate has an effect on the axial capacity of the section in both the RAM Steel Column program and RAM Frame, as described in CAN/CSA-S16-01 Section 13.3.1. Sections of type WWF are considered as being made with flame-cut flange plates, in accordance with G40.20.

HSS-Class:
Hollow structural sections (round, square and rectangular) can be designated to be Class C (cold-formed non-stress-relieved) or Class H (hot or cold-formed stress-relieved). This designation applies to all HSS in the model. The class designation has an effect on the axial capacity of the HSS section in both the RAM Steel Column program and RAM Frame, as described in CAN/CSA-S16-01 Section 13.3.1.

2.4.8 BS 5950 Parameters

Built-Up:
Flange sections can be designated as being built of milled plate or of flame cut plate. The type of plate has an effect on the axial capacity of the section in both the RAM Steel Column program and RAM Frame, as described in the footnotes of table 25 of BS 5950-1:1990 and of table 23 of BS 5950-1:2000.

Hollow Structural Sections
BS 5950-1:2000 stipulates different design requirements for cold-formed and hot-finished hollow structural sections (HSS). The HSS type affects the classification of the cross section (see table 12 in BS 5950-1:2000), and the web shear interaction as described in H.3 of BS 5950-1:2000. The engineer can stipulate which type of HSS they are using in the Section Properties tab of the **BS5950 Parameters** dialog.

Distance to Axis of Restraint
The BS 5950-1:2000 has changed the method in which the lateral torsional bending capacity of beams is calculated. In certain circumstances, where the tension flange of a beam is fully braced and the compression flange unbraced, the height of the restraint to the tension flange affects the bending capacity of the member (See BS 5950-1:2000 4.3.5.3c, Annex G.1. and G.2.). This will typically affect the design of cantilever or continuous members in the zone where the unbraced lower flange is in compression. The engineer can stipulate the distance from the top – of –flange (tension), to the center of the restraint of that flange, in the Section Properties tab of the **BS5950 Parameters** dialog.

Modular Ratio:
For the modular ratio used for the calculation of the composite section properties, the user may specify a value to be used explicitly, or the user may specify the proportions of the live loads that are to be considered long term versus short term. If the Use option is selected, that value would be used for all composite beams. If the Calculate option is selected, the value of the modular ratio is determined based on Clause 4.1 of BS 5950:Part 3:Section 3.1:1990. The dead load applied to the composite section is all considered to be long term load. This is done for each beam individually based on its specific loads. These parameters only affect composite beam design in RAM Steel Beam Design.

2.4.9 Australia Parameters

Combination Factor Ψv:
Load combinations considered in the Steel Column module are based on AS/NZS 1170:2002 if Australia is the selected design code. The combinations include the consideration of the following:

[1.2*G*, *S*u, *ψc*Q]

Expanded as

1.2DL + Psic * LL + Snow

The engineer can set an appropriate ψc in the RAM Manager dialog, to be used to generate load combinations appropriately in Steel Column when roof live load is considered as snow (See Member Loads Criteria command).

Beam Lateral Torsional Buckling

When loads are applied transverse to a member (typically gravity loads applied to a beam) the engineer should indicate where those loads are applied for consideration in member capacity calculations (Table 5.6.3(2) in AS 4100-98). In the RAM Manager the engineer can designate if loads applied transverse to a member are applied at the shear centre or top of section. This setting is considered only for beam design in RAM Frame and RAM Steel Beam. For columns and braces, where forces are only applied at the ends of a member, they are assumed to be through the shear center. This setting will only affect the capacity of a beam in segments in which they are not braced laterally (i.e. they are subject to lateral torsional buckling).

Residual Stress Welded Shapes

When calculating member capacity (See Section 5 of AS 4100-98) it is necessary to know the residual stresses in the section which are a function of the manner in which the section is created. For welded sections the engineer can specify the manufacturing method used. Rolled I sections are assumed to be Hot Rolled (HR) per table 5.2. Rolled tubes and pipes are assumed cold formed (CF) per Steel Designer Handbook. All welded sections must be designated as either Heavy Welded (HW) or Light Welded (LW) in this dialog.

2.4.10 Selecting Model Units

When first creating a database, the system of units is specified as English, SI or Metric. The proper initial selection of model units is those units desired in the final output reports (see the Output Description chapters in each Design Module manual). The system administrator specifies default units at the time of installation. To change model units:

- Select the **Criteria - Units** command.

A dialog box will appear showing the current Units selection. Select the desired model Units and click the **OK** button. To cancel the command, click the **Cancel** button.

At any time during the modeling or design phases, the system of units may be changed. This capability is useful in the case where a job requires the production of designs in units that the engineer is unaccustomed to working. A building can be modeled, analyzed and output reports obtained in the familiar system of units. Once the design is completed to the satisfaction of the engineer, the units can be changed for the final output reports.

2.5 Invoking the RAM Modeler

Structural models used in analysis and design are created and modified using the RAM Modeler. To invoke the RAM Modeler:

- Select the **Model** command.

Control is then passed to the RAM Modeler. The program window appears with a black view screen and a new series of menu commands displayed on the menu bar. The use of these commands is addressed in the RAM Modeler manual and in the on-line help.

2.6 Invoking the RAM Steel Beam Design Module

The RAM Steel Beam Design Module performs either batch or interactive design of all gravity steel beams and joists in the currently selected model, using the currently specified design criteria. Numerous output reports are available which are designed to be included with the Building Calculations. To invoke the Beam Module:

- Select the **Design – RAM Steel Beam** command.

Control is then passed to the RAM Steel Beam Design module. The program window appears with a black view screen and a new series of menu commands displayed on the menu bar. The use of these commands is addressed in the RAM Steel Beam Design manual and in the on-line help.

2.7 Invoking the RAM Steel Column Design Module

The RAM Steel Column Design Module performs either batch or interactive design of all gravity columns and base plates in the currently selected model, using the currently specified design criteria. Numerous output reports are available and are designed to be included with the Building Calculations. To invoke the Column Module:

- Select the **Design – RAM Steel Column** command.

Control is then passed to the RAM Steel Column Design module. The program window appears with a column layout on the view screen and a new series of menu commands displayed on the menu bar. The use of these commands is addressed in the RAM Steel Column Design manual and in the on-line help.

2.8 Invoking RAM Frame

RAM Frame performs a finite element analysis on the lateral framing systems of the currently selected model, using the currently specified criteria. Load case and combination generators, a drift analyzer, steel stress check post processors and other tools help the engineer arrive quickly at an optimal frame design. A variety of on-screen view options are available for investigating the results of an analysis. Numerous output reports are available which are designed to be included with the Building Calculations. To invoke the RAM Frame module:

- Select the **Design – RAM Frame** command

Control is then passed to RAM Frame. The program window appears with a three dimensional view of the model on the screen and a new series of menu commands displayed on the menu bar. The use of these commands is addressed in the RAM Frame manual and in the on-line help.

2.9 Invoking RAM Concrete

RAM Concrete performs the analysis and design of Concrete members in both the gravity and the lateral framing systems. Numerous output reports are available which are designed to be included with the Building Calculations. To invoke the RAM Concrete module:

- Select the **Design – RAM Concrete** command

2.10 Invoking RAM Foundation

RAM Foundation is a foundation design program using the superstructure analysis and design results to help analyze and design the foundation substructure. Column and Wall loads and sizes as well as base plate dimensions are automatically transferred from the database to the RAM Foundation module.

RAM Foundation designs single column footings with uniaxial or biaxial moments and continuous footings using an elastic foundation model. The analysis takes into consideration the fact that the soil under the footing can only take compression forces. The program optimizes both the footing size and reinforcement, and then provides an interactive method of modifying the design to suit the engineer's needs. Numerous output reports are available which are designed to be included with the Building Calculations. To invoke the RAM Foundation module:

- Select the **Design – RAM Foundation** command

Control is then passed to RAM Foundation. The program window appears with a collapsed plan view of the model on the screen and a series of menu commands displayed on the menu bar. The use of these commands is addressed in the RAM Foundation manual and in the on-line help.

NOTE: Before a design can be achieved in RAM Foundation, column and/or wall forces must be available. Forces on gravity members can come from RAM Steel Column or from RAM Concrete Analysis and forces on lateral members come from RAM Frame. Without available forces, RAM Foundation will open but foundations will not be designable.

2.11 Invoking RAM Concept

RAM Conce*pt* is an analysis and design program that uses the finite element method for elevated concrete floor systems, or mat foundations. The floors or mats can be post-tensioned concrete, reinforced concrete or a hybrid of the two.

In this context, the term "design" means that the user defines the following: structural geometry, loads, load combinations, design strips or sections, and post-tensioning layout (if applicable), and Conce*pt* calculates the required amount of reinforcement for flexure and one-way shear according to relevant code requirements; the stud shear reinforcement (SSR) for punching shear, stresses for flexure, and deflections.

Note that the option to invoke RAM Conce*pt* will be grayed (inactive) if RAM Conce*pt* is not installed.

To invoke the RAM Conce*pt* module:

- Select the **Design – RAM Conce*pt*** command

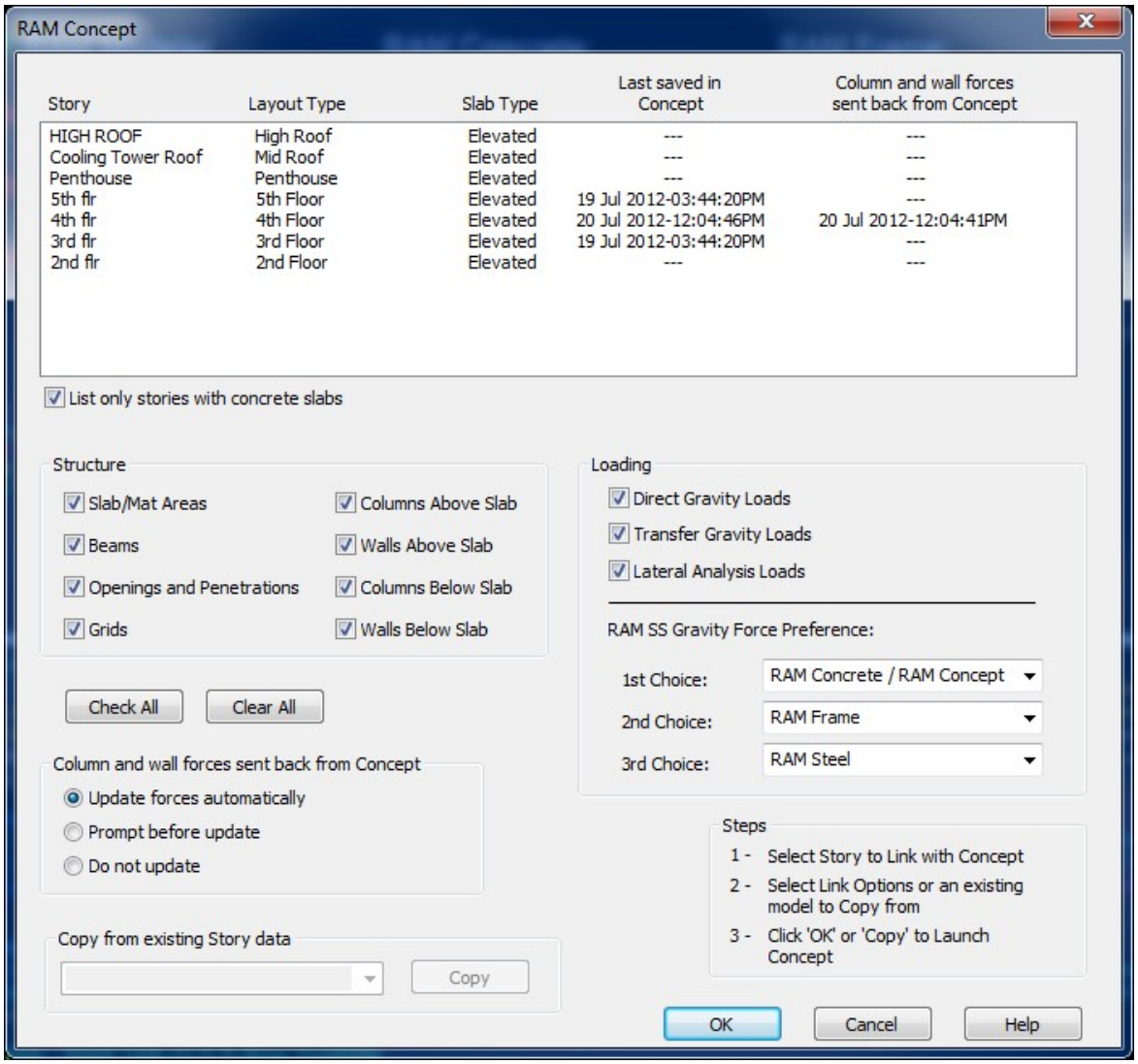


Figure 2-3

The RAM Conce*pt* export dialog will then display. At the top of the dialog is a list of stories in the model. The list shows if a story has any slabs or mat foundations, as well as the slab type. It also shows the last time a story's data was saved in Conce*pt* and the last time forces were sent back from Conce*pt* to be included in the model. There is also a checkbox option to only list stories that have concrete slabs.

In the center of the dialog are options to specify structure, loading and force update options.

The "Structure" section allows selection of the structural objects to be included. Select any combination or all from the available choices:

- Slab/Mat Areas
- Beams
- Openings and Penetrations
- Grids
- Columns Above Slab
- Walls Above Slab
- Columns Below Slab
- Walls Below Slab

Generally speaking all of the structure objects should be exported to Conce*pt* unless the Conce*pt* model already exists and modifications to those objects have been made in Conce*pt*, in which case the modified items should excluded from the subsequent export.

The "Loading" section allows selection of load types to be transferred. Select any combination or all from:

- Direct Gravity Loads, the surface, line and points loads in the model
- Transfer Gravity Loads, the loads from columns or walls on higher stories that bear upon the slab
- Lateral Analysis Loads, the axial forces and unbalanced moments from the Ram Frame analysis

If "Transfer Gravity Loads" is selected the "RAM SS Gravity Force Preference" section is enabled. Select 1st, 2nd and 3rd as desired from the following:

- RAM Concrete / RAM Conce*pt*
- RAM Frame
- RAM Steel

The 3 selections must be unique, no duplicate references.

In the "Column and wall forces sent back from Concept" section, specify whether to:

- Update forces automatically
- Prompt before update
- Do not update

Currently these updated reactions are used in the Concrete Analysis module and affect the concrete column design only. When the Conce*pt* model includes post-tensioning or more precise load modeling, it's advisable to send column and wall reactions back.

If a model has already linked one or more stories to Conce*pt*, the "Copy from existing Story data" option will be available when a non-linked story is selected. Select an existing story from the drop-down and select the "Copy" button to launch Conce*pt* for the new story using the reference story as a template including all layers and objects other than Mesh Input.

After selecting “OK” or “Copy”, Control is then passed to RAM Conce*pt*. The program window appears on the screen and a series of menu commands displayed on the menu bar. The use of these commands is addressed in the RAM Conce*pt* manual and in the on-line help.

NOTE: Stories analyzed in Conce*pt* through this dialog and backup files will be included as part of the RAM Structural System model. They can only be opened by RAM Conce*pt* through this link. Conce*pt* will not be able to open these files when running stand-alone unless they are extracted from the RAM model and renamed.

2.12 Invoking the Bentley Structural Dashboard

The Structural Dashboard is a free program from Bentley Systems, Inc. that assists in managing the data and workflow of projects from start to finish. The Structural Dashboard provides a single interface to utilize Bentley's integrated products for a complete project workflow.

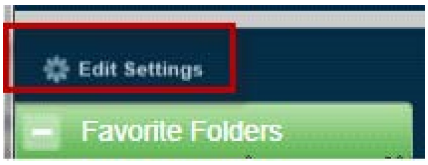
- Manage workflows for common project type (including building workflows and plant workflows)
- Use a unified interface to launch all your Structural applications
- Create customized workflows specific to your projects
- Receive Structural News from customizable RSS feeds
- Join Structural online community and access blogs, wikis and forums
- Download product upgrades from Bentley's SELECTservices Center
- Manage project files and links within your workflow
- Consolidate common project information and dynamic links to project documents

For more information on the Structural Dashboard, go to:
http://communities.bentley.com/Wiki/view.aspx/Structural_Integration

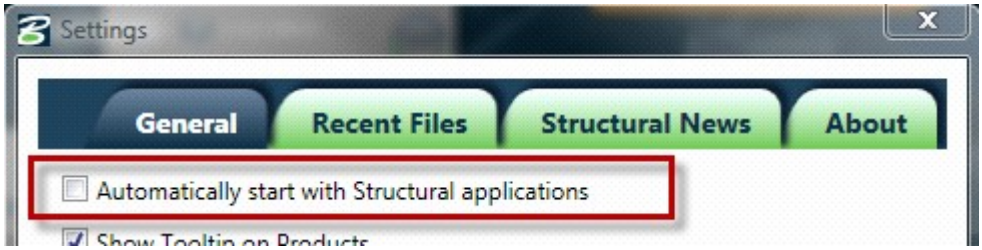
To download the Structural Dashboard, go to:
http://www.bentley.com/en-US/Promo/ISM/downloads.htm#dashboard

2.12.1 Automatic Launching of the Bentley Structural Dashboard

By default, the Structural Dashboard will automatically launch when RAM Manager starts if it is installed. The auto- launch feature can be set or disabled within the Structural Dashboard itself. To change the setting select the “Edit Settings” icon on the Structural Dashboard main screen.



On the “General” tab, check or uncheck the “Automatically start with Structural applications” box to enable or disable auto-launching of the Structural Dashboard program.



2.12.2 Manually Launching the Bentley Structural Dashboard

If the automatic launch feature has been disabled, the Structural Dashboard can be launch in one of the following ways:

- Select the **File – Structural Dashboard** command
- Click the Bentley logo icon on the left of the RAM Manager screen

2.13 RAM Manager Reports

The RAM Manager provides the engineer with a variety of Reports. They are a small subset of the output capabilities available in the various analysis and design modules. These Reports are found in the **Post-Processing** menu.

A Data Echo report is available to view the model data in a textual form. The Data Echo is an effective tool for finding modeling errors.

Output summary reports of Beam Loads, Column Loads, Frame Loads, Wall Loads and Foundation Loads can also be obtained. These reports facilitate the design of these members by means other than the Design Modules. Chapter 4, RAM Manager Output, provides complete information on these reports.

2.14 RAM Manager DXF Files

Three types of DXF files can be created from the RAM Manager for transferring the RAM Structural System model and design information to CAD: one containing the floor framing layouts with their associated concrete beam schedules, one containing column and footing layouts with their associated steel and concrete column schedules, and one containing the shear wall reinforcement. These options are provided in order to reduce duplication of effort between the designer and the drafter by allowing the creation of CAD drawings directly from the RAM Structural System database. A simple DXF file viewer is also included. Note that the RAM Frame program includes an option to produce another type of DXF file, a frame elevation. See the RAM Frame manual for more information.

2.14.1 Floor Framing DXF

To create a DXF File of the floor framing layouts for the current model:

- Select the **Post-Processing - DXF File – Floor Framing** command.

A dialog box will appear prompting for the desired floor type and the name of the DXF File to be created. In addition, numerous options exist for file creation such as scaling, grid bubble shape, layering, etc. Options for the creation of a concrete beam schedule are also given. Option defaults are set by the system administrator at the time of installation but can be modified on a project-by-project basis from this dialog box. Note that if a scale other than the default is selected, it may be necessary to modify some of the options in order to maintain the proper proportions.

2.14.2 Column and Footing Schedule DXF

To create a DXF File of the column and footing layouts for the current model:

- Select the **Post-Processing - DXF File – Column & Footing** command.

A dialog box will appear with numerous options for the exported file. Two styles of files can be created. **Schedule + Base Plan** creates a steel column schedule with sizes, a concrete column schedule with sizes and reinforcing, a spread footing schedule, a pile cap schedule and a continuous footing schedule, plus a base plan with column and footing marks. Identical columns are grouped and given the same column marks. Only steel gravity columns are included in the steel column schedule. Identical spread footings are also grouped and given the same footing mark. **Base Plan Only with Size** creates a base plan only, with the column sizes and footing shapes of the lowest level shown on the plan.

In the Base Plan tab the grid systems that will be included in the DXF drawing can be selected. The default selected grid systems are the grid systems that were selected for the bottom story, the last time the RAM Manager was opened.

The order by which the columns are included in the schedule can be specified to be by Coordinates, by Quantity, or by Weight. If **Coordinates** is selected for Mark Order, columns will be listed in order of ascending coordinates (e.g., columns in the lower left corner of the plan will be listed first, those in the upper right corner of the plan will be listed last). If **Quantity** is selected for Mark Order, columns will be listed in order of frequency of occurrence. That is, the column that is most common will be listed first. If **Weight** is selected for Mark Order, columns will be listed in order of total weight, with the heaviest column being listed first.

The columns can be further grouped by their base level by selecting **Group by Base Level**. This is only applicable if columns terminate at various levels throughout the structure. If selected, columns will be ordered based on the lowest level of the column (e.g., columns that terminate at the lowest level will be listed first). This establishes the primary order for which columns are listed. **Coordinates**, **Quantity**, or **Weight** then establish the order by which columns that terminate at the same level are listed. Note that if **Base Plan Only with Size** is selected, the Mark Order options are irrelevant.

The spread footing mark order method in the schedule is taken to be the same as the column order method. On the Footing Schedule tab there are options to give each footing a unique mark or to group similar footings. This option only works for spread footings and pile caps. All continuous footings will have unique marks. There are also options to include on the plan drawing the footing's Top Of Slab (TOS) and orientation with respect to the global coordinate system.

There are several options by which the DXF file can be customized. These include scale, pen colors, layer names, text size, etc. The defaults were established at the time the program was installed, but they may be modified for any particular DXF file.

2.14.3 Shear Wall DXF

To create a DXF File of the shear walls for the current model:

- Select the **Post-Processing - DXF File – Shear Wall** command.

A dialog box will appear with numerous options for file creation such as scale, grid bubbles, layering, etc. Options for the key plan are also given. Option defaults are set by the system administrator at the time of installation but can be modified on a project-by-project basis from this dialog box. Note that if a scale other than the default is selected, it may be necessary to modify some of the options in order to maintain the proper proportions.

2.14.4 DXF File Viewer

A simple DXF file viewer is included and can be used to view DXF files. In some cases the viewer does not show the information exactly as it will appear in the actual CAD program, but will generally show the information contained in the file. The drawing can be printed from the viewer.

To view a DXF file:

- Select the **Post-Processing – DXF File – View DXF** command.

2.15 RAM Manager CIS/2 Files

A CIS/2 data exchange file (CIMSteel Integration Standard v2.0) can be created for transferring the RAM Structural System model to other engineering software packages that offer a CIS/2 import translator. Currently, only geometry, end-fixities, and shapes are exported to the data exchange file.

Please note that the CIS/2 data exchange file uses the STEP file format. The generated file will have a STP extension.

To create a CIS/2 File for the current model:

- Select the **Post-Processing - CIMSteel File** command.

A dialog box will appear prompting for various items to include in the exported file. For more information about these options, click the **Help** button. To generate the file, select **Create File**. This will bring up the Save As dialog box prompting for the name of the STEP file to be created.

For more detailed information on CIMSteel files and the CIS/2 translator, see Appendix B.

2.16 License Management

RAM Structural System licenses can be manually checked in and checked out from the Select Server using the License Management tool. To access this tool, select the **Tools – Manage Licenses…** command.

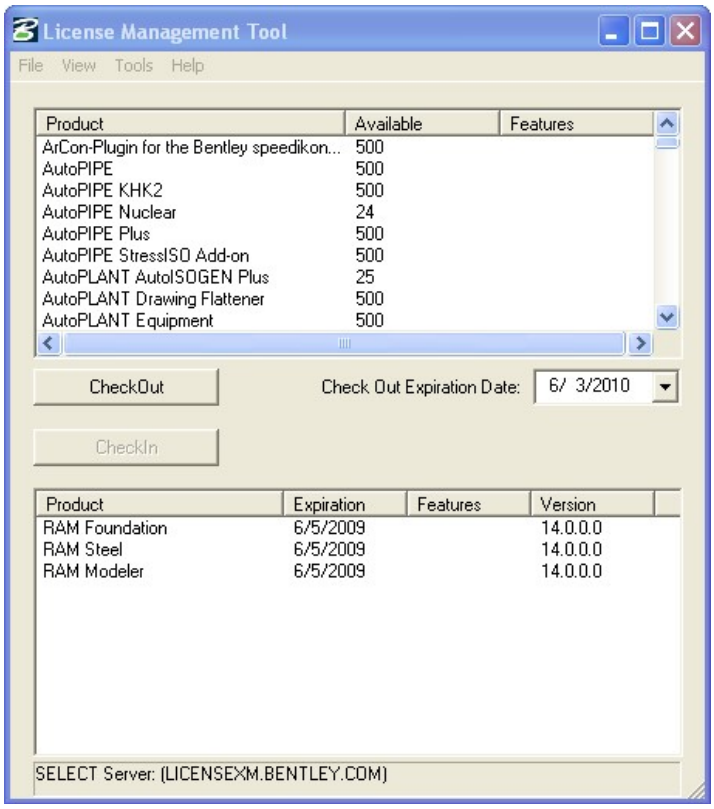


Figure 2-4

As the user enters and exits the various modules of the RAM Structural System, licenses are automatically checked out from the Select Server. These licenses remain checked out until the RAM Manager is closed.

To manually return a license to the Select Server, select the module and click the "Check In" button. Multiple modules can be selected for Check In by holding down the Ctrl key and clicking on the desired modules.

2.17 Report Preferences

The RAM Manager provides the user with the ability to change the report options for the current model. However, the default styles that are shipped with the RAM Structural System have been

selected for optimal viewing of the reports. Making the margins smaller or the font much bigger may cause the text in a table to wrap to the next line. If this happens, selecting a smaller font for the Body section of the reports and rerunning the reports should correct the problem.

To modify the report defaults for the current model:

- Select the **Tools – Report Styles** command.

The Report Styles dialog box will appear. The user can set the following report styles:

2.17.1 Margins

Set the top, bottom, left and right margins for the pages of the reports.

2.17.2 Paper Size

Select either a pre-defined paper size, or select Custom Size and specify the page width and height.

2.17.3 Text Styles

Each rich text format report contains four different styles of text: Title, Header, Section Headings, and Body. The user can specify font name, font size, color, bolding, underlining, etc., for each of the four different text styles.

2.17.4 Default Destination

There are four options for the default destination of the reports. The option selected here will be the selected option when one of the Analysis and Design modules is invoked. However, the output destination may be changed in each of those modules on a report-by-report basis. For the default destination, select one of the following:

Screen – reports are previewed on the screen in a non-editable viewer. The viewer has toolbar buttons that allow printing, zooming in and out, navigating through the pages of the report, and closing the preview.

Printer – reports are sent directly to the printer without previewing.
Text File – report information is sent to a file. The file is a comma-separated value file; that is, it contains all of the values of the original report, with commas separating the values, without formatting. The user can specify a file extension of either .csv or .txt, but the file is the same in either case. This is the only option that does not use the other user-selected report styles. One font is used for all sections of the report and all formatting is removed. However, commas are in place to separate table columns so that the file can be imported into a spreadsheet program.

Viewer File – reports are saved to the Virtual Print Engine file format (.vpe). These files can only be opened with the report viewer program, vpeview.exe, which is distributed with the RAM Structural System and installed in the same directory as the program executables. This format allows reports to be saved and later viewed on-screen outside the RAM Structural System.

2.17.5 Company Logo

A Company logo can be included in the upper left-hand corner of each page of the reports if this option is selected. The RAM International logo is distributed with the program, but this can be replaced with a logo of the user's company if desired. The logo to be shown must be contained in a .bmp or .jpg graphics file named logo.bmp or logo.jpg and placed in the same directory as the RAM Structural System program executables (the RAM Program subdirectory). The size of the bitmap is irrelevant, as it will be scaled appropriately.

2.18 Modifying Program Defaults

The program defaults can be changed using the RAM Defaults Utility. By changing the program defaults, all models created subsequent to the change will use the new program defaults. Note that this command should not be used to change defaults for the current model or any other existing models. Defaults for existing models may be changed in the various criteria commands.

To modify the program defaults:

- Select the **Tools – RAM Defaults Utility** command.

For more information about changing program defaults, see Section 6.1 of this manual and the Program Defaults section of the Installation manual.

2.19 Steel Table Editor

The RAM Manager is linked to an application called the RAM Table Editor. This table editor can be used to modify or create steel master, beam and column tables. To launch the Steel Table Editor with the tables of the currently selected model automatically loaded:

- Select the **Tools – RAM Table Editor** command.

Refer to the online help or the Steel Table Editor documentation on more information on how to use the Table Editor.

2.20 Getting the Current Version and Copyright Information

To get the current Version and Copyright information for the RAM Manager:

- Select the **Help - About RAM Manager** command.

The End User License Agreement (EULA) can also be accessed and viewed there.

2.21 Manual

The RAM Manager manual is available as an electronic file (.pdf) and can be accessed from the program. To view the manual:

- Select the **Help – Manual** command.

2.22 Tutorial Wiki

The RAM Structural System Tutorial is available on a Bentley Be Communities website wiki and can be accessed from the program. To view the wiki:

- Select the **Help – Tutorial Wiki** command.

2.23 Technical Support

For information on Technical Support go to the Bentley support website:

http://selectservices.bentley.com/

To create a new Service Ticket to request Support:

- Select the **Help – Contact Tech Support** command.

2.24 Program Updates

All program updates and patches must be obtained from the Bentley SELECT Server. For a shortcut to the server, select the **Help – Check for Program Update** command.

The Bentley SELECT Server can also be accessed through a web browser. Go to www.Bentley.com, and select the Support & Services menu.

It is a good idea to periodically check for updates to ensure that the latest version of the RAM Structural System is always used. Note that the RSS feed will provide timely notifications when a new version is available.

Before proceeding with any installation, exit completely from the RAM Structural System.

2.25 Online Resource

Links to several online resources are available directly from the **Help – Online Resources** menu.

To view the RAM Structural System product web page:

- Select the **Help – Online Resources – Product Information** command.

To go to Bentley's blogs, wikis and forums at Be Communities:

- Select the **Help – Online Resources – Community** command.

To see a list and access recent eSeminars:

- Select the **Help – Online Resources – OnDemand Seminars** command.

To see information on the latest Training resources offered by Bentley Institute:

- Select the **Help – Online Resources – Training** command.

To access the Structural Product TechNotes and FAQs page, provide by the Technical Support group:

- Select the **Help – Online Resources – Technotes and FAQs** command.

To view the SELECTservices web page, the portal to Technical Support and software downloads:

- Select the **Help – Online Resources – Select Services** command.

2.26 Exiting the RAM Manager

Exit the Modeler and all design modules before exiting the Manager.

Exit the RAM Manager by selecting the **File - Exit** command.

3 Integrated Structural Model (ISM) Link

The RAM Structural System can import from and export to Bentley Integrated Structural Model (ISM) repositories using its ISM link tools.

The ISM link supports import (from ISM repository) and export (to ISM repository) of the following model data:

- Stories
- Beams (including cantilevers)
- Columns
- Walls
- Horizontal and vertical braces
- Spread footings, pile caps
- Steel Joists – SJI parallel chord girder
- Radial and orthogonal grids
- Parametric sections
- Steel and concrete sections
- Steel joists
- Varying sections

The ISM link supports export only (to ISM repository) of the following model data:

- Parallel circular rebar in columns
- Parallel rectangular rebar in columns
- Perpendicular circular rebar in columns
- Perpendicular rectangular rebar in columns
- Perpendicular spiral rebar in columns
- Perpendicular straight rebar in columns
- Rebar in walls
- Shear studs (beam)
- Slab (geometry only)
- Composite decks

Note that ISM Link menu items will be grayed out if Bentley Structural Synchronizer is not installed on the same computer as the RAM Manager application. See the end of this section for more information on link limitations with this release.

The next sections contain a brief description of ISM and Structural Synchronizer. Please refer to the corresponding user's manual for more detailed information on their capabilities.

3.1 What is ISM?

Bentley's Integrated Structural Model (ISM) is a technology for sharing structural engineering project information among structural modeling, analysis, design, drafting and detailing applications. ISM is similar to Building Information Modeling (BIM), but focuses on the information that is important in the design, construction and modification of the load bearing components of buildings, bridges and other structures.

3.1.1 Purpose of ISM

There are two related purposes for ISM:

1. The transfer of structural information between applications.
2. The coordination of structural information between applications.

To provide for the first purpose (transferring information), ISM provides a means of defining, storing, reading and querying ISM models.

To provide for the second purpose (coordination of information), ISM additionally provides capabilities to detect differences between ISM models and to selectively (based on user selection) update either an ISM repository or an application's data to provide a user-controlled level of consistency between the two data sets.

3.1.2 ISM and Application Data

ISM is not intended to store all of the information that all of its client applications contain. Rather, it is intended to store and communicate a *consensus* view of data that is *common* to two or more of its client applications, such as RAM Structural System.

RAM Structural System continues to hold and maintain its own private copy of project data. Some of the application data will duplicate that of the associated ISM repository. The application data may even conflict with that in the ISM repository. RAM Structural System (or the user) may decide that a conflict gives the best data for RAM Structural System's and ISM's different uses.

3.2 ISM Sync Tools Overview

RAM Structural System can send structural data to and from an ISM repository through a set of ISM Syncing tools. These tools allow both creation and updating of RAM Structural System models as well as ISM repositories. Further, these flexible tools allow the user to begin models and move data as workflow dictates.

The ISM link tools are accessed through the '**ISM**' submenu of the main '**File**' menu. There are four menu options listed under the '**File – ISM**' menu as shown below:

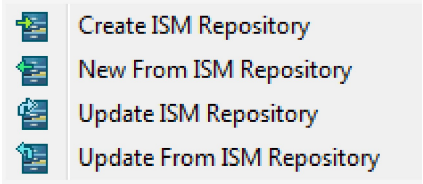


Figure 3-1

The following table explains which tool to use for a given task:

If the user needs to …	… use this tool	Description

If the user needs to …	… use this tool	Description
Create a new ISM repository from an existing RAM Structural System model		***Create ISM Repository*** transfers the current model opened in RAM Structural System and generates a new ISM repository. This is the most common way in which an ISM repository is initially created.
Create a new RAM Structural System model from an existing ISM repository		***New From ISM Repository*** creates a new RAM Structural System model from an existing ISM repository. This is used to transfer model data from other tools used in the workflow.
Update an existing repository to reflect changes made in a RAM Structural System model		***Update ISM Repository*** will coordinate changes made to the model in the RAM Structural System and coordinate some or all of those changes with an existing ISM repository.
Update an existing RAM Structural System model to reflect changes in an ISM repository		***Update From ISM Repository*** allows the user to update a RAM Structural System's model with some or all of the changes which have been made to the ISM repository.

Not all tools are available at all times as follows:

- If ISM is not installed these items will be grayed out and not available.
- If ISM is installed but no RAM Structural System model is loaded, only '**New From Repository**' will be available.

Structural Synchronizer

The program provided by ISM for accepting or rejecting model data changes is called "Structural Synchronizer". Care should be taken to make sure the latest Structural Synchronizer release is installed. This application provides the user with a powerful set of tools for moving data between the applications used in a daily workflow. Even relatively small structural models have enormous amounts of data and ISM allows this data to be re-used with ease. Care must be taken that only desired data is transferred between applications.

When accepting changes made by client applications to an ISM Repository, some attention must be paid to what changes are actually being made. A small change in a client application can have unintended repercussions, if accepted. A repository is intended to represent the data that is *common* to all the client applications. Some client application models will use only a subset of the repository data but changes made to them can affect the entire repository if these changes are accepted when a repository update action is performed.

This brief overview is not intended as a sufficient description on using the full capabilities of the Structural Synchronizer. Please refer to the Bentley Structural Synchronizer's user's manual for more detailed information.

3.2.1 Create ISM Repository

To Create an ISM repository from the currently loaded RSS model, select the **File – ISM – Create ISM Repository** command.

The "Set ISM Repository to Create" dialog will open. Specify the file name and destination directory for the new ISM Repository. If the destination directory has existing ISM models, and the **Preview** box is checked, selecting any existing repository will show its thumbnail view. Note that ISM repositories have a '**.ism.dgn**' file extension.

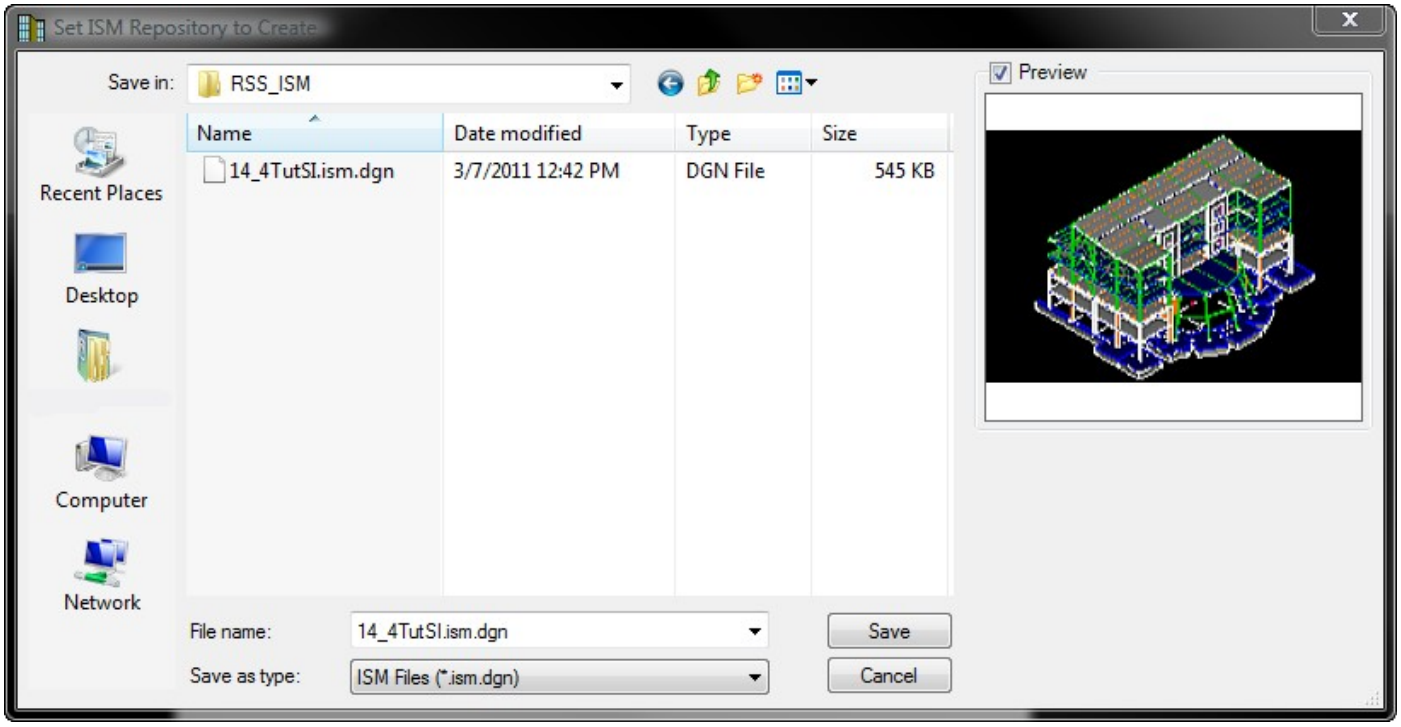


Figure 3-2

After specifying the file name of the destination repository, click the **[Save]** button to continue or **[Cancel]** to abort. If an existing ISM repository is selected, a dialog will be displayed asking the user if they want to replace it. Select **[Yes]** to continue or **[No]** to return to the previous dialog.

If model changes have not been saved, a dialog will be displayed allowing the user to Save **[Yes]**, Discard **[No]** or **[Cancel]** the operation. A reminder message also appears reminding the user to re-save after the repository is created.

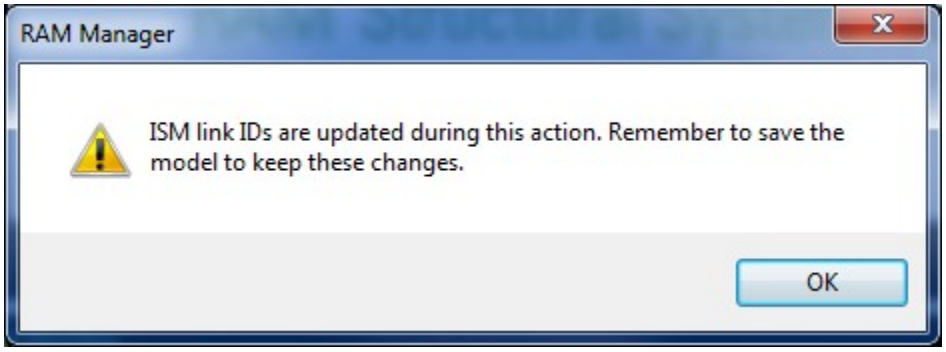


Figure 3-3

A common ISM link dialog will be displayed. This dialog, with title changes, is shown during any ISM link operation. Click the dialog's **[Show Details]**/**[Hide Details]** button for more or less information on the operation's progress. The **Log** tab displays expandable/collapsible tree lists of any warnings or errors that are encountered during the operation. Click the **[+]** or **[-]** symbols to expand or collapse the lists.

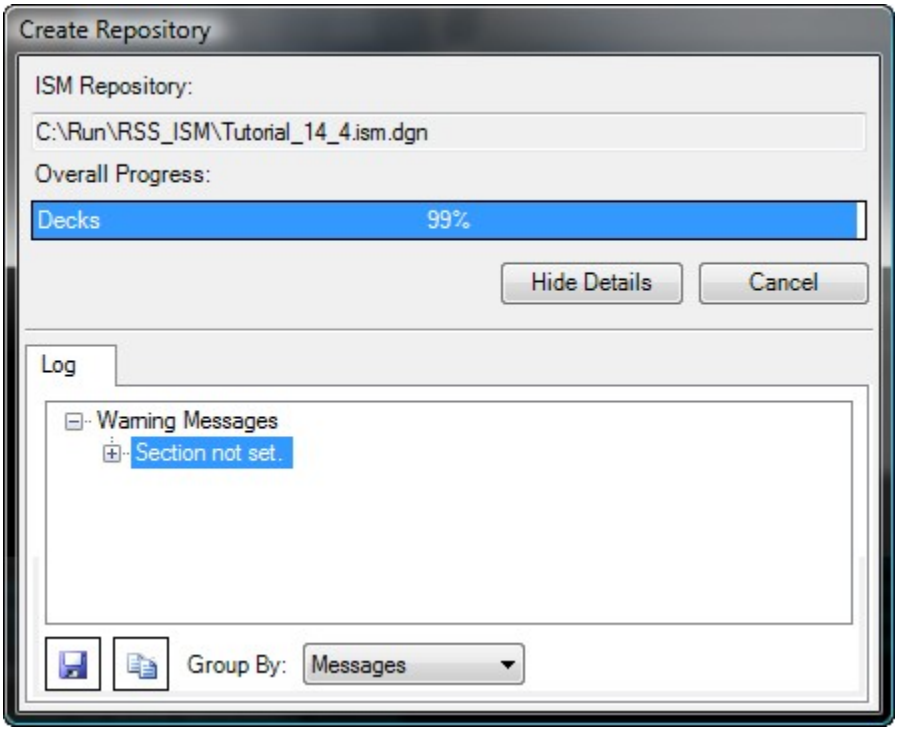


Figure 3-4

Note, if the model includes any embedded RAM Concept floor files, an option will be given whether to **Include RAM Concept Data** or not. When it is included, each floor file will be temporarily opened so that any modified geometry and user defined reinforcement can be included in the full ISM repository.

Slab/Deck Mappings

If the ISM link tool cannot find an appropriate deck mapping, the "Deck Mappings" dialog box will be displayed.

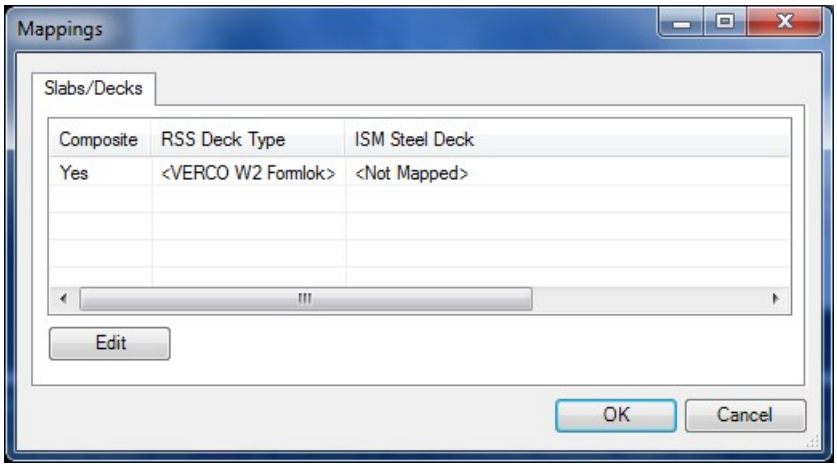


Figure 3-5

Click **[OK]** or **[Cancel]** to leave deck mappings unchanged. Click **[Edit]** or double-click on any entry line to bring up the "Composite Steel Deck Mapping" dialog.

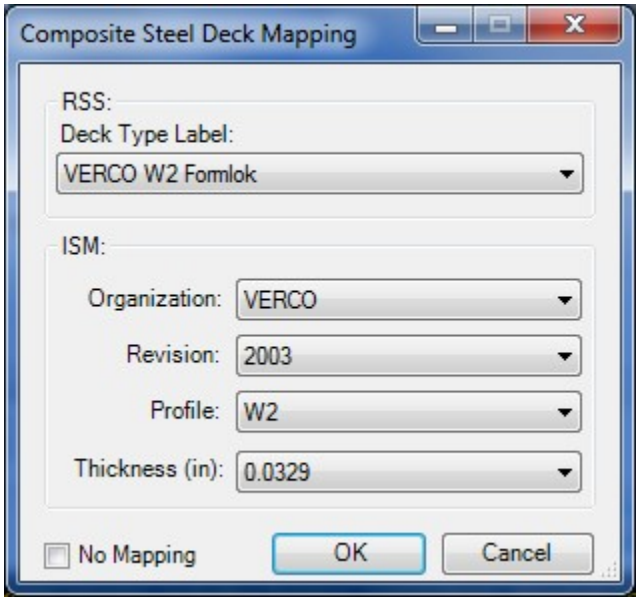


Figure 3-6

It is not required that an RSS deck be mapped to an ISM equivalent deck. To create a mapping uncheck the 'No Mapping' box and select the appropriate values. To remove a mapping, check the 'No Mapping' box. Click **[OK]** to save the changes or **[Cancel]** to return to the "Deck Mappings" dialog.

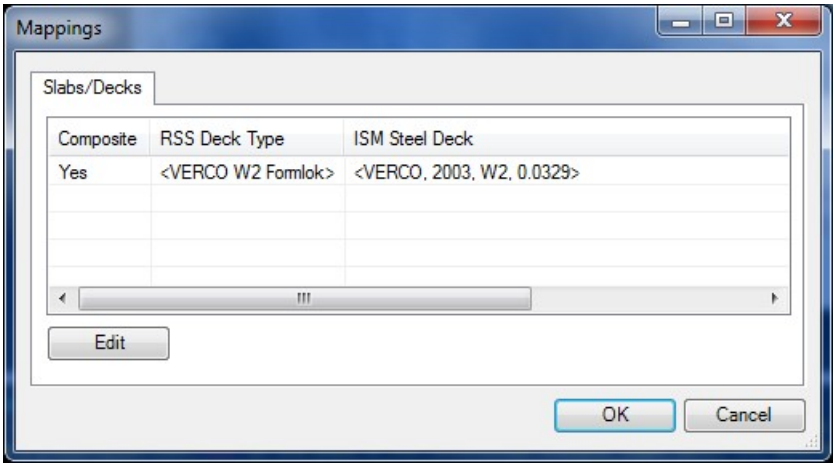


Figure 3-7

Click the **[OK]** button to save any deck mapping changes or **[Cancel]** to discard.

After initialization completes, the newly created ISM repository will be loaded and shown in the Bentley Structural Synchronizer application.

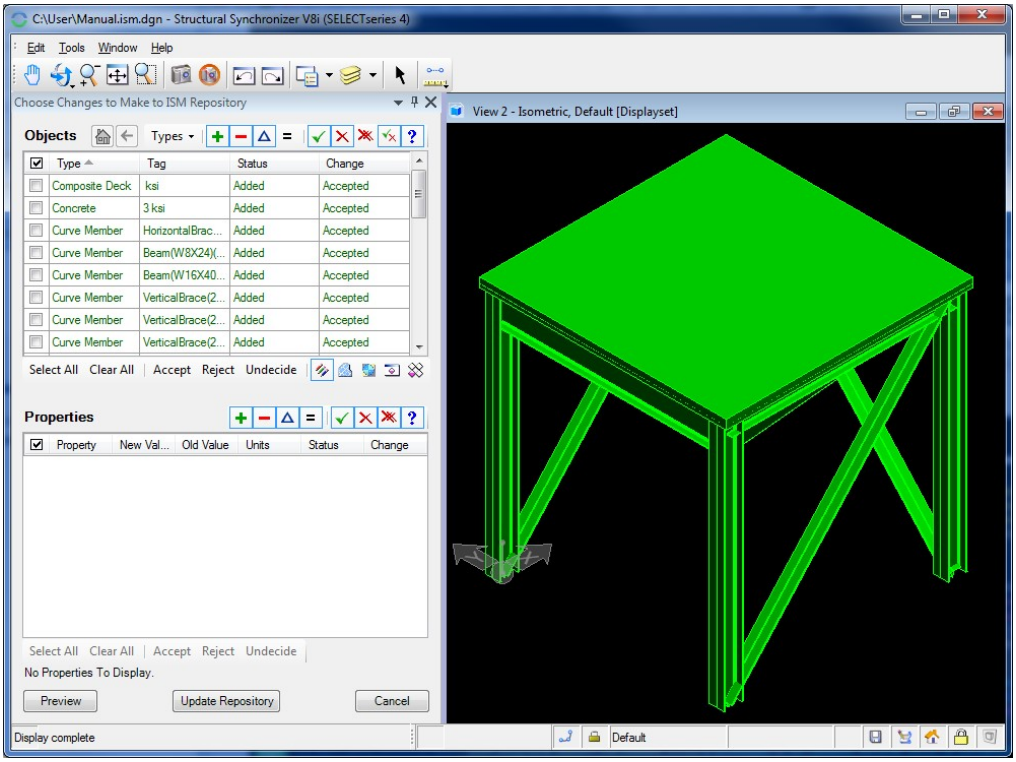


Figure 3-8

Note that this can take a minute or more depending on the size of the model and capabilities of the computer. The newly created candidate repository can now be worked with to select geometry and other data to include or exclude.

When done, click the **[Update Repository]** button to finalize creation and saving of the repository or **[Cancel]** to abort. A Design history window will open so that you can optionally enter a note about the last action, e.g. “New repository from RAM SS”.

After the Structural Synchronizer application closes, RAM Manager will reload the original model. The ISM link “Create Repository” dialog will remain open for final review of the log. Click the **[Close]** button to dismiss the dialog.

Please refer to the Structural Synchronizer’s user’s manual for details on its use.

3.2.2 New From ISM Repository

To Create a RAM Structural System model from an ISM repository select the **File – ISM – New From ISM Repository** command.

If model changes have not been saved, a dialog will be displayed allowing the user to **[Save]**, **[Discard]** or **[Cancel]** the operation. If either **[Save]** or **[Discard]** is selected, or the model was already in a saved state, the currently loaded RAM model will be unloaded and the “Select ISM Repository to Import From” dialog will open. Specify the ISM repository desired and click the **[Select]** button to continue or **[Cancel]** to abandon the import. Note that this is the same dialog as is displayed in the **Create ISM Repository** section.

After clicking **[Select]** the RAM Manager will display a message to ‘Please specify RSS model to create’. Press the **[OK]** button to close this dialog. RAM Manager’s standard “New” (model) dialog will now be displayed.

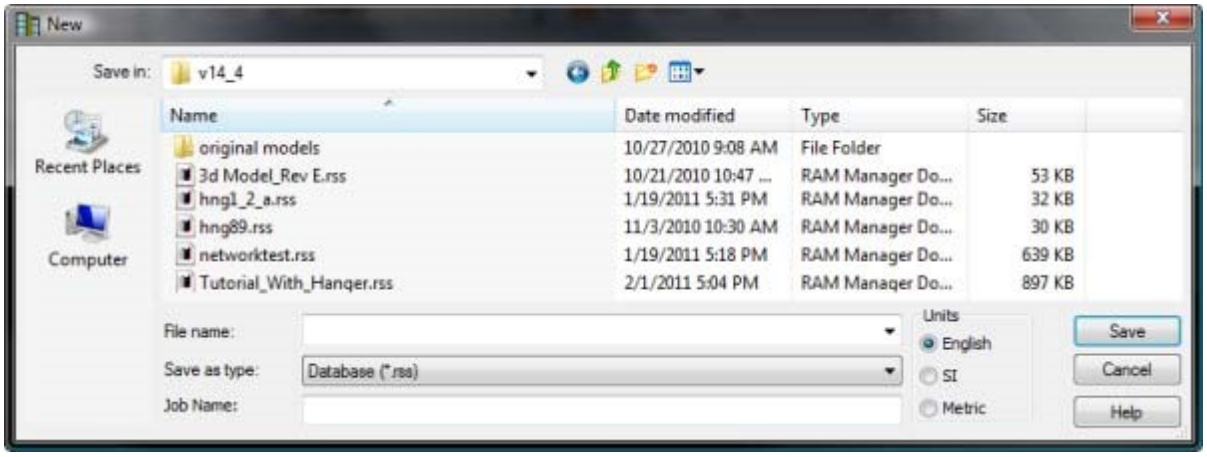


Figure 3-9

Specify the name and location for the new model just as when creating a new RAM model and click the **[Save]** button to continue the import from ISM or **[Cancel]** to abandon the operation.

A Tables Selection dialog will appear allowing the user to alter the default tables.

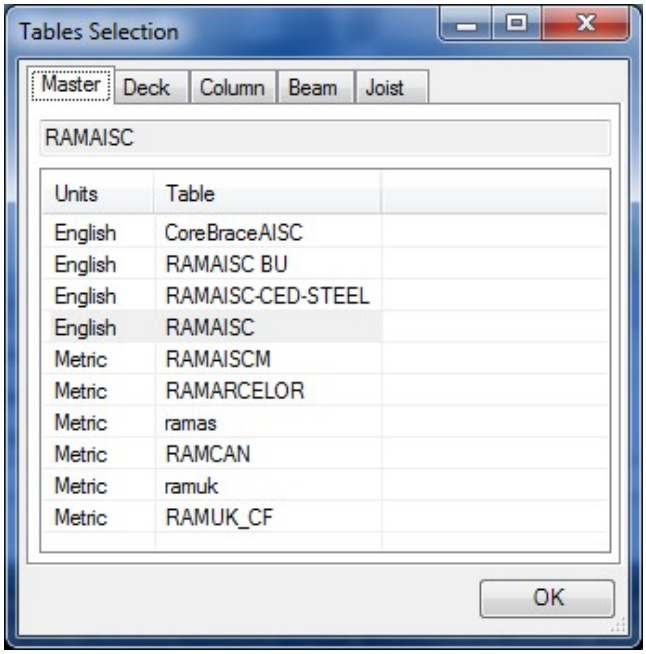


Figure 3-10

Select the appropriate tables and click **[OK]**. The tables can always be altered later on through the RAM Manager criteria menu options.

This is followed by the Mappings dialog box as described in section 3.2.1. Edit the mappings if necessary and click **[OK]**.

The common ISM RSS link dialog will now display with the title “Create From Repository”. Any warnings or errors encountered will appear in the **Log** tab.

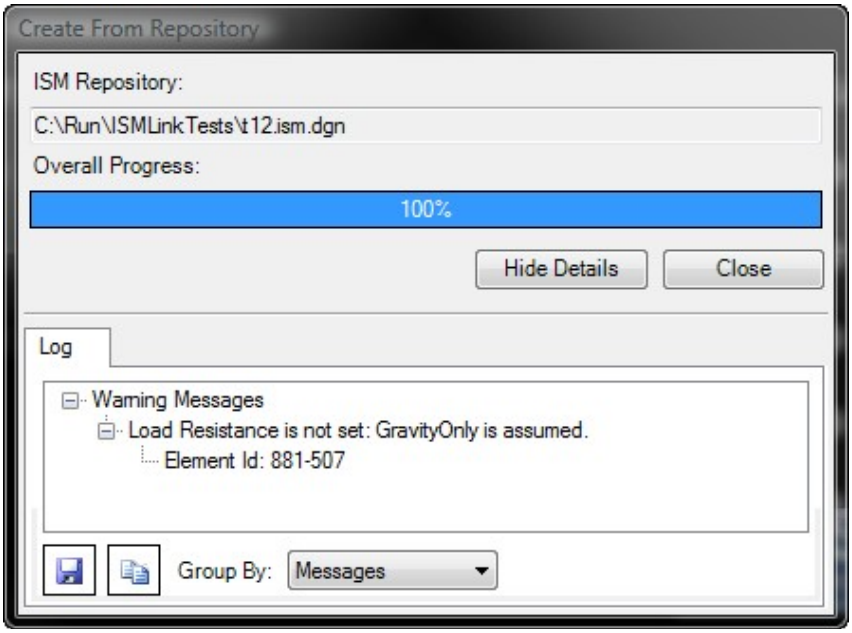


Figure 3-11

Click the **[Cancel]** button on the ISM link "Create From Repository" dialog at any time during the create to abort the process. The newly created RAM Structural System model will be automatically loaded by RAM Manager when 'create from repository' completes. The ISM link "Create From Repository" dialog will remain open for final review of the log. Click the **[Close]** button to dismiss the dialog.

If an error is encountered that prevents new the RAM model from being created, and another RAM model had been unloaded prior to the import, an error message will be displayed and then the previously unloaded model will be reloaded by the RAM Manager.

3.2.3 Update ISM Repository

To update an existing ISM repository using the currently loaded RAM Structural System model select the **File – ISM – Update ISM Repository** command.

If model changes have not been saved, a dialog will be displayed to **[Save]**, **[Discard]** or **[Cancel]** the operation. If either **[Save]** or **[Discard]** is selected, or the model was already in a saved state, the currently loaded RAM model will close. A dialog titled "Select ISM Repository to Update" will open. Specify the ISM repository desired and click the **[Select]** button to continue or **[Cancel]** to abandon the update. Note that this is the same dialog as is displayed in the **Create ISM Repository** section.

The common ISM link dialog will now display with the title "Update Repository". The RAM model is exported to a temporary ISM repository. This temporary repository is used by the Structural Synchronizer application to identify candidate model changes and is not seen by the user.

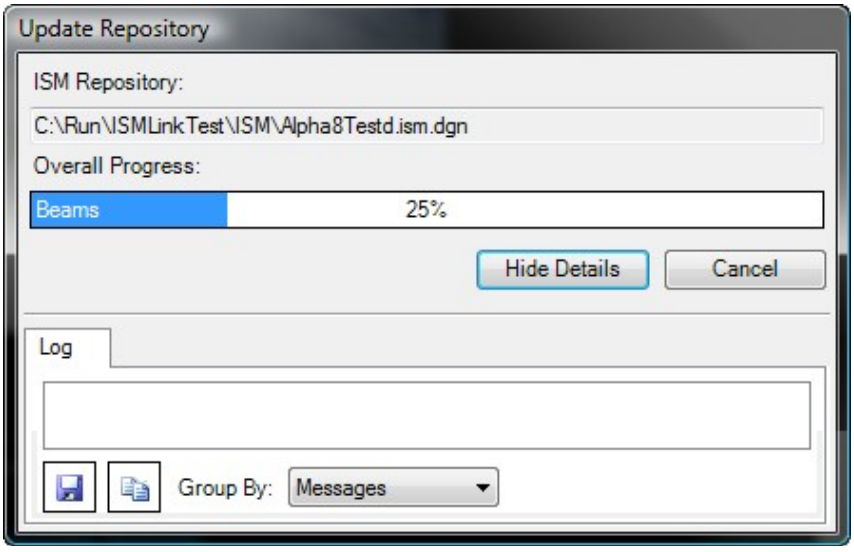


Figure 3-12

If deck mappings are needed and cannot be found, the same series of "Deck Mappings" dialogs as shown in the **Create ISM Repository** section will display. Click the **[Cancel]** button on the **Update Repository** dialog at any time during the export to abort the update.

The Bentley Structural Synchronizer application will launch after the ISM link finishes exporting of the RAM model. In the Structural Synchronizer, accept or reject changes as desired and click **[Update Repository]** to finish the update or **[Cancel]** to abort.

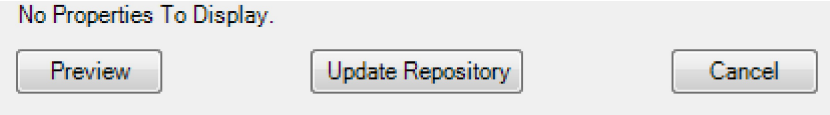


Figure 3-13

The Design History window will appear for you to add notes.

RAM Manager will automatically reload the previously closed RAM model after Structural Synchronizer closes. Note that this may take some time as Structural Synchronizer finalizes repository creation. After reviewing the **Log** tab if desired, click the **[Close]** button to dismiss the "Update Repository" dialog.

3.2.4 Update From ISM Repository

To update the currently loaded RAM Structural System model from an existing ISM repository select the **File – ISM – Update From ISM Repository** command.

If model changes have not been saved, a dialog will be displayed to **[Save]**, **[Discard]** or **[Cancel]** the operation. If either **[Save]** or **[Discard]** is selected, or the model was already in a saved state, the currently loaded RAM model will close. A dialog titled "Select ISM Repository to Update From" will open. Specify the ISM repository desired and click the **[Select]** button to continue or **[Cancel]** to abandon the update. Note that this is the same dialog as is displayed in the **Create ISM Repository** section.

The common ISM link dialog will now display with the title "Update From Repository". The RAM model is then exported to an unseen temporary ISM repository. If deck mappings are needed and cannot be found, the same series of dialogs as shown in **Create ISM Repository** will be displayed during creation of the temporary repository.

The selected 'From' repository will then be loaded and displayed in the Bentley Structural Synchronizer application. The Structural Synchronizer display will indicate identified differences found. Use the Structural Synchronizer interface to select geometry to import or exclude. After desired geometry and data changes have been selected, click the [**Update Application]** button to begin update of the RAM Structural System model or [**Cancel]** to abort. The Structural Synchronizer application will close.

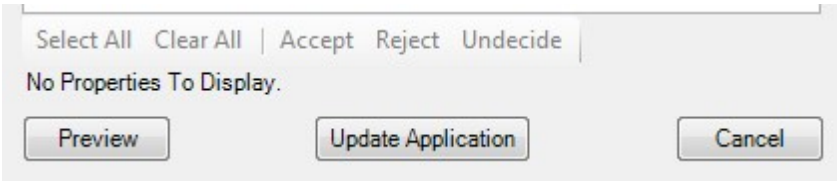


Figure 3-14

If **[Update Application]** is selected, the ISM link will update the RAM Structural System model with the user accepted changes. The "Update From Repository" dialog will show progress and log messages during the update process.

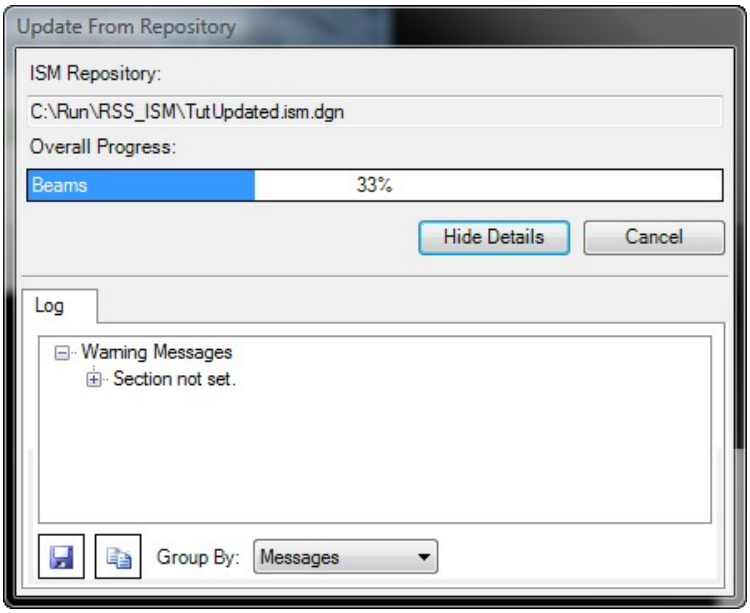


Figure 3-15

Click the **[Cancel]** button on the "Update From Repository" dialog at any time to abort the update. When the update completes, the model will be reloaded by the RAM Manager and set to an "unsaved" state. Click the **[Close]** button to dismiss the "Update From Repository" dialog.

It is recommended that changes be reviewed before saving the model. The RAM model will be returned to its last saved (pre-update) state if changes are not saved before the model is closed.

3.2.5 Errors during ISM Link Operations

The RAM Manager creates backup files of all RAM models before link operations begin. This backup is redundant since the original model is closed before the ISM link tool is activated and will still exist in its last saved state. If a serious error does occur during link execution, the next time that RAM model is loaded and an ISM link menu item is selected, a dialog will be displayed with showing the full path to the created backup file. The name of the file also gives the date and time the error occurred. The following shows an example of this dialog:

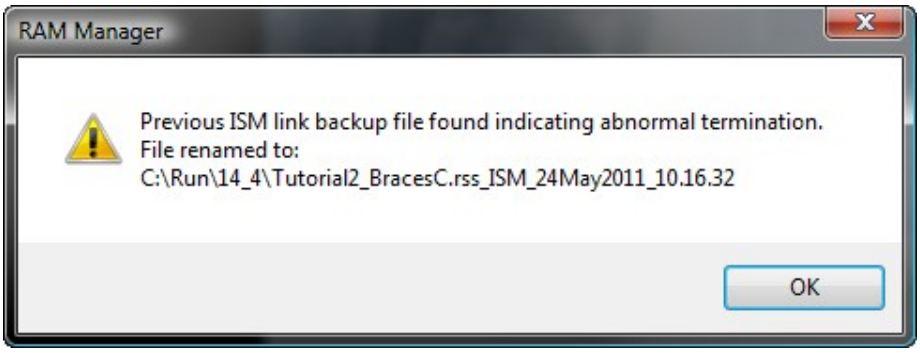


Figure 3-16

3.2.6 Current Limitations and Considerations

This release of the ISM link has known limitations. Some of these stem from the differences between analytical and physical models. Others are due to the differences in capabilities of RAM Structural System, ISM, and other programs. The following highlights these:

- Parametric and table sections are supported for steel elements. Not all built-ups are interchangeable.
- Only rectangular and T sections are supported for concrete beams.
- Only rectangular and circular sections are supported for concrete columns.
- Kinked or curved members are imported from their end points as a linear member.
- Members have fixed placement points. Centroid-centroid for columns. Middle top for beams.
- Some configurations of cantilevers may be unresolveable and will require modification in Modeler.
- Composite beams stud values are exported only.
- Camber values are exported only.
- Only rectangular wall openings are imported. If an opening is not rectangular a bounding box is imported.
- Only rectangular or circular beam web openings are imported.
- Only rectangular and horizontal isolated foundations are imported.

4 Model Status

Because it is a fully integrated system, the RAM Structural System modules are dependent upon each other's data and results. For instance when self-weight is being considered in a design, changing a member size in one module can invalidate the results of one or more of the other modules.

The Model Status feature that has been implemented in the RAM Structural System has three notable attributes: the model status indicator lights in RAM Manager, the **File – Model Status** command in each module and status bar indicator lights in each module.

4.1 Model Status Indicator in RAM Manager

When a model is opened in the RAM Manager, the logo screen will be replaced by a list of modules with status indicator lights. These lights provide an overview to the status of the model.

4.1.1 An Overview of the Lights

A gray light indicates that module is not relevant for the open model. For example, the concrete modules will have gray lights if a model consists entirely of steel members.

A light blue light indicates that additional data is needed before a design can be performed. For example, load combinations need to be generated or bar patterns need to be assigned before concrete columns can be designed.

A red light indicates either that an analysis or design has never been run or that an action occurred to invalidate an existing analysis or design. Note that the red light has a white tick mark in it to distinguish it from the other lights for those who may have difficulty distinguishing between colors.

A yellow light indicates that an action occurred that impacts the results. The results should now be considered approximate and used with caution.

A green light indicates that the analysis or design is current and valid for all members.

4.1.2 Model Status Information for Specific Modules

To get additional information for each module, do one of the following:

- Click on the "click here" message on the screen (see Figure 4-1)
- Click the model status toolbar icon (see Figure 4-2)
- Issue the **File – Model Status** command.

Figure 4-1

Figure 4-2

This will cause the Model Status dialog to be displayed (see Figure 4-3).

At the top of the dialog is a light legend to explain the meaning of each light color.

The second section of the dialog lists each module and gives more information about its status. If additional data is needed for the model to be complete for the given module, the needed data is listed here.

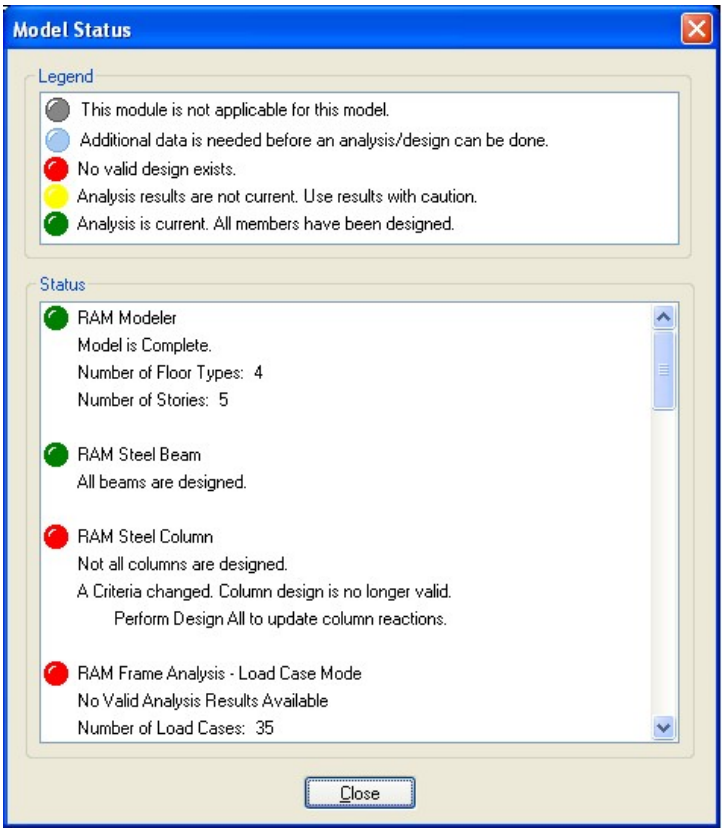


Figure 4-3

4.2 Model Status within each Module

Each of the RAM Structural System modules has an Indicator Light in lower right hand corner of the status bar. As changes occur in the module, the status light is a visual indicator of state of the model. The status bar indicator lights are synchronized with the lights on the RAM Manager screen.

Additionally, each module has a **File – Model Status** command that launches a dialog that provides more detailed information on the status of the model.

5 RAM Manager Output

The RAM Manager provides several Post-Processing output reports to aid in the design of a structure. The model data can be output in textual form to aid in debugging. Regardless of the design modules licensed, reports summarizing the gravity loads on beams, columns, walls, and foundations are available to facilitate the design of these members. A complete Foundation Load report, including lateral column loads, may be obtained for users with the RAM Frame module. A report of gravity loads on all lateral frame members is also available. Refer to Section 2.16.4, Default Destination for a description of the various ways the output can be shown or saved.

In addition to the load reports, the RAM Manager Post Processor provides means for converting the structural model and designs from the RAM Structural System into DXF files that can be read by major CAD Systems.

5.1 General Criteria

The General Criteria report lists the criteria settings selected in the **Criteria – Member Loads** and **Criteria – Self-Weight** commands. It includes listings of criteria that impacts the gravity loads used in each design module. Information on Live Load reduction, the treatment of roof and snow loads, and self-weight settings is displayed.

To print the General Criteria report:

- Select the **Post-Processing – General Criteria** command.

5.2 Echo of Input Data

The Echo of Input Data report provides a textual description of the model data. It includes listings of story data and deck and loading properties. Information on each member, such as coordinates, shape and orientation for columns and user selected beam depth restrictions and user-assigned sizes is displayed in an easy to read tabular form. This provides a written record of the model and provides a means of verifying its accuracy.

To print the data echo report:

- Select the **Post-Processing - Data Echo** command.

5.3 Gravity Beam Loads

The RAM Structural System calculates Gravity loads on beams when the “Framing Tables” are “built”. A Loads on Gravity Beams Report is available to assist in the Gravity Beam Design for the structure. This report is useful to those who will be designing gravity beams by means other than the RAM Steel Beam Design module and is not normally needed by those who will be using RAM Steel Beam Design

The data in the Loads on Gravity Beams Report is organized by story level. For each beam in a given story, span coordinates, beam length, and loading information is provided.

To obtain a printout of the Loads on Gravity Beams Report it is necessary to first invoke any one of the design modules. Once the "Framing Tables" have been "built":

- Select the **Post-Processing – Gravity Beam Loads** command.

5.3.1 Loading Information - Point Loads

Dist:
This is the location of the point load represented as a distance measured from the left end of the beam.

DL:
This is the magnitude of the dead load.

CDL:
This is the magnitude of the construction (precomposite) dead load.

RedLL:
This is the magnitude of the unreduced reducible live load.

NonRLL:
This is the magnitude of the non-reducible live load.

StorLL:
This is the magnitude of the storage live load.

RoofLL:
This is the magnitude of the unreduced roof live load or snow load.

Red%:
This is the amount of live load reduction allowed for the corresponding Live Load Type, based on the currently selected building code. For Roof Live Loads, if snow load has been specified, "Snow" will appear in lieu of a reduction percent.

CLL:
This is the magnitude of the construction live load.

5.3.2 Loading Information - Line Loads

Dist:
This is the distance, measured from the left end of the beam, to the beginning or end of a line load, for the give load segment.

DL:
This is the magnitude of the dead load at Dist.

CDL:
This is the magnitude of the construction (precomposite) dead load at Dist.

LL:
This is the magnitude of the unreduced live load at Dist.

Red%:
This is the percent of live load reduction allowed for the type of load indicated, based on the currently selected building code.

Type:
This is the type of live load reduction used based on the currently selected building code. “Red” indicates that the live load is Reducible, and can be reduced by the percent shown. “Roof” indicates that the live load is a Roof load. “Snow” indicates that the roof live load is a non-reducible snow load. “NonR” indicates that the live load is non-reducible. “Stor” indicates that the live load is a storage live load.

CLL:
This is the magnitude of the construction live load at Dist.

5.4 Gravity Column Loads

The RAM Structural System calculates Gravity loads on columns when the “Framing Tables” are “built”. A Loads on Gravity Columns Report is available to assist in the Gravity column and foundation design for the model. This report is useful to those who will be designing gravity columns by means other than the RAM Steel Column Design module and is not normally needed by those who will be using RAM Steel Column Design

The data in the Loads on Gravity Columns Report is organized by column line. A summary of the gravity loads at each story level within a column line is provided.

To obtain a printout of the Loads on Gravity Columns Report it is necessary to first invoke any one of the design modules. Once the “Framing Tables” have been “built”:

- Select the **Post-Processing - Column Loads** command.

This output provides a listing of the loads on every gravity column in the structure. The output includes:

- The column line.
- The dead load on the column. This includes the self-weight if the column has been designed in the RAM Steel Column Design module and the option to include column self-weight has been selected.
- The unreduced positive and negative live loads on the column. The reduction type for the live load and the allowable reduction percent based on the currently selected building code is also provided.
- The total dead plus reduced positive live load and total dead plus negative live load.

All column loads listed in this report are unfactored loads.

5.5 Gravity Wall Loads

The RAM Structural System calculates gravity loads on walls when the “Framing Tables” are “built”. A Loads on Gravity Walls Report is available to assist in the Wall Design for the Model.

The Loads on Gravity Walls Report is organized by story level. For each wall on a given story, the coordinates, length, thickness, tributary area, and loading information are provided.

To obtain a printout of the Loads on Gravity Walls Report it is necessary to first invoke any one of the design modules. Once the “Framing Tables” have been “built”:

- Select the **Post-Processing - Wall Loads** command.

See Sections 5.3.1 and 5.3.2 for a description of the variables listed on the output.

5.6 Frame Loads

The RAM Structural System calculates gravity loads on frame members when the “Framing Tables” are “built”. A Gravity Loads on Lateral Members Report is available to assist in the lateral frame design for the Model. Users who do not have the RAM Frame program can enter data from this report manually into another finite element analysis package for Frame analysis and design.

The loads listed are not an accumulation of the loads. Rather, they are the loads applied to that member at that level from the floor and from gravity members. It is a listing of the gravity loads that need to be applied to the frame members for their analysis.

Note: In this report these loads are based on an assumption of simple span framing, not the results of a frame analysis considering member fixity and continuity. That is, it is based on simple tributary loading.

The data in the Gravity Loads on Lateral Members Report is organized by story level. For each lateral beam on a story level, the span information and loading specifications is provided. The coordinates and loading information are tabulated for lateral columns. The coordinates, length, thickness, tributary area, and loading information is tabulated for walls.

To obtain a printout of the Gravity Loads on Lateral Members Report it is necessary to first invoke any one of the design modules. Once the “Framing Tables” have been “built”:

- Select the **Post-Processing - Frame Loads** command.

See Sections 5.3.1 and Section 5.3.2 for a description of the variables listed on the beam load output section of the Frame Loads output.

5.7 Foundation Loads

The Foundation Loads report gives the forces at the base of the structure. The report includes forces on both gravity and lateral members. If the lateral analysis was not performed when the report was generated the lateral member foundation loads will not be reported.

The foundation forces can be taken from either the RAM Concrete Analysis or RAM Steel Analysis. To obtain forces from RAM Concrete, an analysis must be successfully completed in the RAM Concrete Analysis module. To obtain forces from RAM Steel any of the design modules must have been invoked (Framing must have succeeded). If the Foundation loads on the lateral members are also desired lateral analysis must be performed in RAM Frame. Once the "Framing Tables" have been "built" and/or RAM Concrete Analysis completed:

- Select the **Post-Processing - Foundation Loads - Gravity Forces From RAM Steel** OR
- Select the **Post-Processing - Foundation Loads - Gravity Forces From RAM Concrete** command.

Note that irrespective of the selected option, gravity forces are not carried down brace members (which are lateral members in RAM Structural System). Also the choice of source of forces is with respect to the gravity members. All gravity (and lateral) forces on lateral members (walls, braces and columns) are taken from the RAM Frame Analysis results.

Refer to the RAM Foundation Manual for more specifics about the difference between RAM Steel and RAM Concrete Analysis forces.

The Foundation Loads Report is organized into two parts. The first part reports the loads at the base of each member for all gravity columns and walls. For gravity members the data in the report is organized by column or wall number. The report includes the lowest Story level columns with loads on the member for Dead, Self weight, Positive Live, Negative Live, Positive Roof Live, Negative Roof Live (Positive and Negative loads only apply to loads originating from RAM Steel) as well as the Minimum and Maximum total unfactored loads.

The second part describes all the load cases used in the lateral (RAM Frame) model followed by Frame Column and Brace loads and finally by the Frame Wall loads. For gravity members the data in the report is organized by column number and includes the lowest Story level. For each load case used in the Frame analysis the Major and Minor Shear and Moment are reported as well as the axial and torsional forces. Where braces frame into the base of a column, the column and brace forces are integrated and reported as a single load at the column line. For Frame walls the report is organized by Wall number and includes the lowest Story level, the Axial and Major Shear forces and the Major Moment.

6 Customizing the RAM Structural System

There are specifications, such as the Building Code used for calculating Live Load Reduction, and values, such as the Yield Strength of beams, that are common to the majority of the structures modeled by one office. Rather than having to specify these options or input these values for every model, many of the common values are saved as program defaults. At the time a model is created, the defaults are assigned to that model.

The RAM Structural System uses numerous program defaults for modeling and designing structures. These defaults are set at the time the program is installed but can be modified at a later time.

The composite deck table, master steel tables, and the design steel tables supplied with the RAM Structural System can be modified and additional tables can be created. They can be customized to better conform to the design philosophy of the design office. The manipulation of tables for customizing the member design is discussed in the next chapter.

6.1 Modifying the Program Defaults for All Models

Defaults can be modified by running the RAM Defaults Utility Program, which can be invoked from the **Tools** menu. When defaults are modified in this way, they overwrite the original defaults set at installation. All ***subsequent*** models will have the new set of defaults assigned to it. This change does ***not*** affect any ***previously*** created databases, including the currently selected database. The exception to this is the DXF defaults, which will automatically be updated to use the new values for all databases, both new and existing.

See Appendix A of the RAM Manager Manual for more information on using the RAM Defaults Utility Program.

6.2 Modifying Defaults on a Project-by-Project Basis

Although the Defaults are assigned to each database when it is created, they can be modified on a Project-by-Project basis to meet the needs of each particular project. Commands such as **Criteria** in the various modules, the **Post-Processing - DXF File** command, or the **Tools** – **Report Styles** command are examples of where the default values can be modified and over-ridden with a different selection. Defaults modified in this way affect ***only*** the current database; it does not affect existing or future databases.

Generally any changes made to the various Criteria and settings are saved as part of the database. Thus if a change is made for a particular analysis, it will retain that value for subsequent analyses of that database. The exception to this is the DXF file settings. Changes made to the DXF file settings only affect the currently created DXF files; it does not permanently change the default DXF settings for this database.

7 RAM Structural System Tables

Carefully choosing program defaults is one way to customize the RAM Structural System working environment. Modifying the system tables or creating new system tables is another way to impose control over the RAM Structural System Software.

This chapter addresses the format and use of all tables associated with the RAM Structural System modules. Not all sections will be relevant to all users.

Note that the maximum length of any individual line is 178 characters.

A table editor is available for editing the Master Steel and Design Steel tables. The Table Editor is invoked using the **Tools – RAM Table Editor** in the RAM Manager. See Appendix C for more information on the Table Editor.

If a Master Steel or Design Steel table used by a database is modified, either by using the Table Editor or some other editing program, the changes will take effect in that database the next time that model is opened.

7.1 Metal Deck Tables

In order to accurately calculate the properties of the composite section in the RAM Steel Beam Design module it is necessary to know certain parameters of the metal deck being used. Rather than requiring that these parameters be calculated and input each time the program is run, the RAM Structural System utilizes a database of decks containing the required information. The user merely has to select the deck type, by name, and the program automatically uses the proper values.

The information for these decks is only used in the design of composite beams. Only decks that are used in composite design should be added to the deck tables. Only decks that have configurations that conform to the steel design specification that is going to be selected should be included. For example, some specifications limit the rib depth; in that case those deep decks should not be included in the deck tables since they would not produce valid designs. The program may not warn of unacceptable deck profiles in all cases. Note that Roof decks that are not used for composite design should not be included in the deck tables.

The metal deck information is used when defining the composite deck and slab information with the **PropTable - Decking** command in the Modeler. The assignment of the deck file to be assigned to the database is made or changed using the **Criteria - Design Steel Tables** command in the RAM Manager. The assignment may be changed at any time; however, changing it after the model has been created will not have any effect on the model since the values in the original table will have already been assigned to the model at the time the **PropTable - Decking** command was performed.

Provided with the RAM Structural System are several deck files. Each file contains the deck properties required by the program.:

- RAMDECKS.DCK contains decks commonly used in the United States.
- RAMCAN.DCK contains decks commonly used in Canada.

- RAMUK.DCK contains decks commonly used in the United Kingdom.
- RAMAS.DCK contains decks commonly used in Australia.

The files are in text format that can be edited, thus allowing the engineer to customize the table to meet current needs. Decks not listed in the original file can be added while decks not used can be deleted. If desired, “generic” decks can be created.

A backup copy of the .DCK file should be made before editing so if errors are made, the original file can be recovered.

Rather than modifying the original deck type table, the engineer might prefer to create a new table consisting of the most commonly used decks. A separate file of decks, perhaps for those commonly used in other countries, might be created. There is no limit to the number of deck tables that the RAM Structural System will support. The only criterion that must be met is that the new files be in the prescribed format.

Deck tables may be defined in either English units or SI units. It is not necessary for the units of the deck table to correspond to the model units; the program will make the appropriate units conversion if necessary. For example, a deck table whose units are English could be assigned to a model whose units are SI, and vice versa.

Helpful Hint: The metal deck information is used when defining the composite deck with the **PropTable - Decking** command in the Modeler. Since deck types in the decks list box are listed in the same order as they appear in the .DCK file, moving the most commonly specified deck type to the top of the file will save time modeling since it will always appear first in the list box.

The deck tables must be located in the Tables directory.

7.2 Metal Deck Table File Format

Files of user defined metal decks may be created using a simple text editor. Do NOT use a word processor as it might embed control characters into the file. Use spaces, not tabs, between data items. The file name must have the extension “.DCK”.

The file has the following format:

The character in the first line indicates the system of units used in the table: an E indicates English units and an S indicates SI units.

The fields on each subsequent line of the file describe the Deck Name and its associated properties. Each field is separated by **one** comma and one or more blanks, in the following format:

Deck Name, Hr, RibSpacing, Wr, AcRib, Ybar, ProfileType

where:

Deck Name is the name of the deck type (19 alphanumeric character or less).
Hr is the nominal rib height (in. or mm).

RibSpacing is the rib spacing (in. or mm).
Wr is the average width of concrete rib (in. or mm).
AcRib is the area of concrete in the rib per foot or per meter of deck width (in.^{2}/ft. or cm^2/m). This value is used for the calculation of the self-weight of the concrete in the deck ribs, for AISC ASD composite section properties and for BS 5950 transverse reinforcement calculation.
Ybar is the distance from the bottom of the deck to the centroid of the concrete area AcRib (in. or mm). This value is used only for AISC ASD, and may be left blank otherwise (except for Australia decks, which should use a value of 0.0).
ProfileType indicates the profile type. For Australia AS 2327.1 a value of 1 indicates an Open-rib Profile; a value of 2 indicates a Closed-rib Profile. For BS 5950-3.1:1990 a value of 1 indicates an Open Trough Profile; a value of 2 indicates a Re-entrant Trough Profile. This value is used only for BS 5950-3.1:1990 Amendment 1:2010 and Australia AS 2327.1, and may be left blank otherwise.

Hr and Wr are used to calculate the allowable shear stud values per Chapter I of AISC Manual of Steel Construction (ASD and LRFD), Clause 17.7 of CAN/CSA-S16-01, Clause 5.4.7 of BS 5950: Part 3: Section 3.1: 1990 or Clause 6.6 of Eurocode EN 1994-1-1. Rib Spacing is used to determine the practical spacing and number of studs along a beam. Hr, AcRib, and Ybar are used to calculate the transformed section moduli (see Figure 7-1). Note that even though the figure seems to indicate that AcRib is the area of concrete in one rib, AcRib is actually the area of concrete in the rib per one foot of deck width if in English units, or per one meter of deck width if in SI units. For example, if the table is in English units and the ribs are spaced 12" on center, AcRib is equal to the area of concrete in one rib; if the ribs are spaced 9" on center, AcRib is equal to the concrete in 1.333 ribs (1.333 = 12.0 / 9.0).

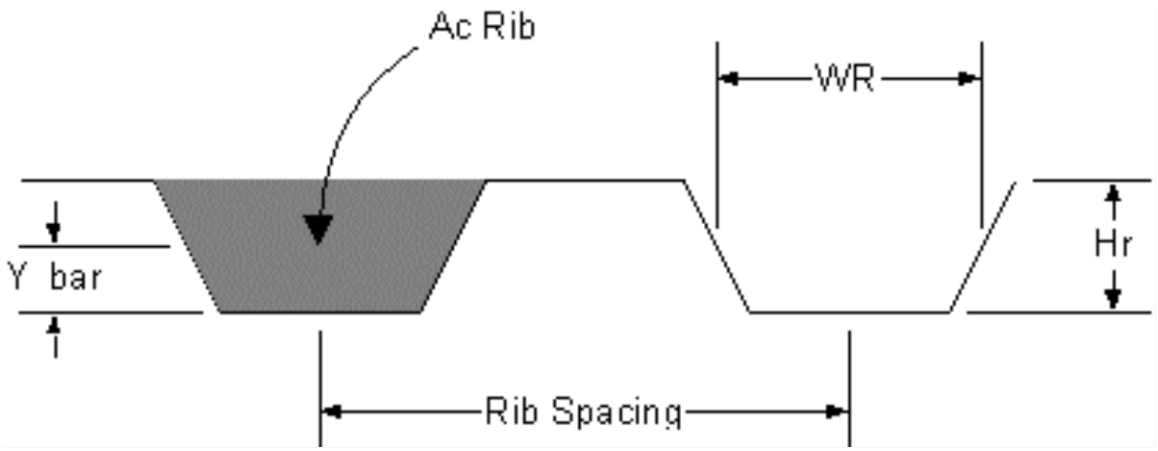


Figure 7-1

Figure 7-2 is an example listing of a RAMDECKS.DCK file:

```
E
ASC 3W, 3.0, 12.0, 6.0, 18.0, 1.604
ASC 2W, 2.0, 12.0, 6.0, 12.0, 1.056
ASC N-24, 3.0, 8.0, 2.312, 10.406, 1.568
ASC B-36, 1.5, 6.0, 2.062, 6.188, 0.788
CMC-USD 3.0LokFloor, 3.0, 12.0, 6.0, 18.0, 1.583
CMC-USD 2.0LokFloor, 2.0, 12.0, 6.0, 12.0, 1.056
CMC-USD 1.5LokFloor, 1.5, 12.0, 6.0, 9.0, 0.792
```

```
CMC-USD 1.5B-Lok, 1.5, 6.0, 2.25, 6.375, 0.794
VERCO W3 Formlok, 3.0, 12.0, 6.0, 18.0, 1.625
VERCO W2 Formlok, 2.0, 12.0, 6.0, 12.0, 1.056
VERCO B Formlok, 1.5, 6.0, 2.125, 6.375, 0.794
VULCRAFT 3.0VL, 3.0, 12.0, 6.0, 18.0, 1.604
VULCRAFT 2.0VL, 2.0, 12.0, 6.0, 12.0, 1.056
VULCRAFT 1.5VL, 1.5, 6.0, 2.125, 6.375, 0.794
WHEELING 3.0SB, 3.0, 12.0, 6.0, 18.0, 1.583
WHEELING 2.0SB, 2.0, 12.0, 6.0, 12.0, 1.056
WHEELING 1.5SB, 1.5, 6.0, 2.032, 6.095, 0.808
WHEELING 1.5SBR, 1.5, 6.0, 3.968, 11.90, 1.104
Flat Slab, 0.0001, 0.0001, 1.0, 0.0, 0.0
```

Figure 7-2

7.3 Master Steel Tables

The Master Steel Tables contain all of the section properties for each available section. The Analysis and Design modules access these tables to gather information for calculations.

Several Master Steel Tables are provided with the RAM Structural System (see Table 7-1 for more details).

- RAMAISC.TAB contains the AISC shapes, including the shapes with the HSS designations.
- RAMAISCM.TAB is identical to RAMAISC.TAB except that it contains the metric equivalent of the AISC shapes.
- RAMCAN.TAB contains the Canadian shapes.
- RAMUK.TAB contains the British shapes.
- RAMUK_CF.TAB contains the standard UB and UC shapes, like the RAMUK.TAB, but it contains the cold-formed hollow structural sections rather than the hot finished hollow sections.
- RAMARCELOR.TAB contains the ArcelorMittal shapes.
- RAMAS.TAB contains the Australian shapes.

The Master Steel Tables are in text file format and can be edited, allowing the engineer to customize the tables. Steel sections are listed in groups according to their shape. Sections not included in the original files can be added and those that are not used can be removed. When removing sections, keep in mind that any section that appears in a Design Steel Table must also appear in the corresponding Master Steel Table. The converse of this is not true. The Master Table can contain sections that do not appear in other tables. Besides adding and deleting sections, groups and sections within groups can be re-ordered in any way the engineer finds convenient.

It is recommended that edited files be given different names than the original files to distinguish them from the originals. Otherwise, a backup copy of the file should be made before editing so the original file can be recovered if desired.

Additional Master Steel Tables can be created to contain specially designed built-up sections or other sections not included in one of the Master Tables. There is no limit to the number of tables supported by the

RAM Structural System. If new Master Tables are created, corresponding Design Tables for the Beam Design module and the Column Design module must also be created.

7.4 Master Steel Table File Format

New Master Steel Tables can be created using a simple text editor. Do NOT use a word processor as it might embed control characters into the file. Use spaces, not tabs, between data items. The file name must have the extension ".TAB".

The file has the following format. The data for a given section must be placed on a single line. Starting a line with a semicolon (;) makes it a comment only line which is ignored by the program.

The first line of the file contains an E or S, indicating English or SI units.

The steel sections are then listed, grouped by shape. Each group begins with a header (I, Channel, L, TS, Pipe, Tee, Roundbar, or Flatbar). W, M, and S shapes are included in the "I" group. Single angle properties are used for single angles as well as to calculate the double angle properties. Square and Rectangular Hollow Sections are included in the "TS" group. Round Hollow Sections are included in the "Pipe" group. Single angle, Tee, Round bars and Flat bar shapes are applicable only to braces.

For each group a section is created. Each section begins with a heading and is followed by lines containing the section properties of members of that group, one line per member. Each group has the format seen in Figure 7-3, where:

Desig is the shape designation (e.g. W18X35), 15 characters maximum.
RollFlg is a flag, either an R or a B, indicating Rolled or Built-up shape. By placing this flag with each section rather than at the beginning of the table, the user can mix Rolled and Built-up shapes in the same Master Table.
Depth is the total depth of the section (in. or mm).
Tw is the web thickness (in. or mm).
Bf_{top} is the top flange width (in. or mm).
Tf_{top} is the top flange thickness (in. or mm).
Bf_{bot} is the bottom flange width (in. or mm).
Tf_{bot} is the bottom flange thickness (in. or mm).
K_{top} is the distance from the outer face of the top flange to the web toe of the fillet (in. or mm).
K_{bot} is the distance from the outer face of the bottom flange to the web toe of the fillet (in. or mm).
Area is the total area (in.^{2} or cm^2).
Ix is the moment of inertia about the X-X axis. (in.^{4} or cm^4).
Sx_{top} is the elastic section modulus with respect to the top flange of the member (in.^{3} or cm^3). (Note that this is Zx by European terminology.)
Sx_{bot} is the elastic section modulus with respect to the bottom flange of the member (in.^{3} or cm^3). (Note that this is Zx by European terminology.)
Zx is the plastic section modulus of the member (in.^{3} or cm^3). (Note that this is Sx by European terminology.)
Iy is the moment of inertia about the Y-Y axis (in.^{4} or cm^4).
Sy is the elastic section modulus of the member with respect to the Y-Y axis (in.^{3} or cm^3).
Zy is the plastic section of the member with respect to the Y-Y axis (in.^{3} or cm^3).
J is the Torsional Constant (in.^{4} or cm^4),

Cw is the Warping Constant (in.^{6} or cm^6).
RT_{top} is the radius of gyration (see AISC Manual) with respect to the top flange of the member (in. or mm). This is used only for AISC ASD and may otherwise be left blank.
RT_{bot} is the radius of gyration (see AISC Manual) with respect to the bottom flange of the member (in. or mm). This is used only for AISC ASD and may otherwise be left blank.

For Channels:
xbar is the distance from the web face to the centroid with respect to the Y-Y axis (in. or mm).
e0 is the distance from the web face to the shear center (in. or mm).
For L's:
Vleg is the vertical leg width (in. or mm).
Hleg is the horizontal leg width (in. or mm).
tLeg is the leg thickness (in. or mm).

For Pipes (Round Hollow Sections):
OutDia is the outer diameter of the member (in. or mm).
Tw is the wall thickness.
I is the moment of inertia of the member (in.^{4} or cm^4).
S is the section modulus of the member (in.^{3} or cm^3).
Z is the plastic section of the member (in.^{3} or cm^3).

```
I
Desig RollFlg Depth Tw Bftop Tftop Bfbot Tfbot ktop kbot Area Ix Sxtop Sxbot
Zx Iy Sy Zy J Cw RTtop RTbot

Channel
Desig Rollflg Depth Tw Bftop Tftop Bfbot Tfbot ktop kbot Area Ix Sxtop Sxbot
Zx Iy Sy Zy J Cw xbar e0

L
Desig RollFlg Vleg Hleg tleg Area Ix Sx Zx Iy Sy Zy J Cw

TS
Desig RollFlg Depth Tw Bf Tf Area Ix Sx Zx Iy Sy Zy J

Pipe
Desig RollFlg OutDia Tw Area I S Z

Tee
Desig RollFlg Depth Tw Bftop Tftop ktop Area Ix Sxtop Sxbot Zx Iy Sy Zy J Cw

RoundBar
Design Diameter Area

FlatBar
Desig RollFlg Bf Tf Area
```

Figure 7-3

NOTE: The section properties of any given member must appear on a single line in the Master Steel Table File. Some shapes are shown above in multiple lines due to space limitations.

NOTE: For HSS, if Bf isn't set, it is set equal to Depth (it is assumed to be Square Hollow Section). If Tf isn't set, it is set to Tw.

It is not necessary to specify values for the italicized variables in the format list. If any of the italicized values in the format list are set equal to zero or are left blank, those values will be calculated based on the dimensions given for the web and flanges except as explained below.

If Bf_{bot} is set equal to zero in the table, Bf_{bot} will be set equal to Bf_{top}.

If Tf_{bot} is set equal to zero in the table, Tf_{bot} will be set equal to Tf_{top}.

If Sx_{bot} is set equal to zero but Sx_{top} is not equal to zero, Sx_{bot} will be set equal to Sx_{top}. If Sx_{top} and Sx_{bot} are both set equal to zero, Sx_{bot} will be calculated based on the dimensions given for the web and flanges.

For angles, if Hleg is set equal to zero, Hleg will be set equal to Vleg.

When zeros occur between non-zero values on a data line, the zeros must appear on the data line. If a zero occurs and ALL fields to the right also contain zeros, those zeros may be removed and the remainder of the line left blank.

Since the actual section properties for rolled shapes are larger, due to the contribution of the fillets, than would be calculated based strictly on the web and flange dimension, some increase in accuracy can be obtained in the design by specifying all of the section properties in the tables.

Figure 7-4 is a sample listing from the RAMAISC.TAB file.

```
E
I
W44X335 R 44.0 1.03 15.9 1.77 0.0 0.0 2.56 0.0 98.5 31100 1410 0.0 1620 1200
150 236 74.7 535000
W44X290 R 43.6 0.865 15.8 1.58 0.0 0.0 2.36 0.0 85.4 27000 1240 0.0 1410 1040
132 205 50.9 461000
.
.
.

Channel
C15X50   R 15.0 .716 3.716 .650 0. 0. 1.4375 0. 14.70 404.00 53.80 68.20
11.000 3.780 8.170 2.67  492.00  .798  .583
C15X40   R 15.0 .520 3.520 .650 0. 0. 1.4375 0. 11.80 349.00 46.50 57.20
9.230 3.370 6.870 1.46  411.00  .777  .767
.
.
.

L
```

```
L8X8X1-1/8    R 8.0 8.0 1.1250 16.700 98.000 17.500 31.600 98.000 17.500
31.600 7.13000 32.50000
L8X8X1        R 8.0 8.0 1.0000 15.000 89.000 15.800 28.500 89.000 15.800
28.500 5.08000 23.40000
.
.
.



TS
HSS16X16X5/8    R 16.0 0.581 16.0 0.581 35.0 1370.  171.  200.  1370.  171.
200.   2170.
HSS16X16X1/2    R 16.0 0.465 16.0 0.465 28.3 1130.  141.  164.  1130.  141.
164.   1770.
.
.
.

PIPE
HSS20.000X0.500 R 20.000  0.465  28.5  1360   136    177
HSS20.000X0.375 R 20.000  0.349  21.5  1040   104    135
HSS18.000X0.500 R 18.000  0.465  25.6   985   109    143
.
.
.
TEE
WT22X167.5 R 22.0 1.02 16.0 1.77 2.56 49.1 2160 392.0 131 233 600 75.3 118
15400 117.0
WT22X145   R 21.8 0.87 15.8 1.58 2.37 42.9 1840 349.1 111 197 523 66.1 103
10700  74.8
.
.
.


ROUNDBAR
RB1/4   0.2500
RB5/16  0.3125
RB3/8   0.3750
.
.
.

FLATBAR
FB3/16X1/2  R  0.50  0.1875
FB3/16X3/4  R  0.75  0.1875
FB3/16X1    R  1.00  0.1875
```

Figure 7-4

NOTE: The section properties of any given member must appear on a single line in the Master Steel Table File. Several members are shown above in two or three lines due to space limitations.

In addition to the standard rolled steel shapes, tables of built-up shapes can be created. Figure 7-5 is a portion of a file of built-up shapes. Notice that in this example, only the web and flange dimensions are included. If other section properties are not included in the file the program will calculate the section properties from the given data. For built-up shapes it is recommended that only the section dimensions (not the section properties) be listed in the table, as shown here.

```
E
I
.
.
.
B14X19.2  B  13.75  0.188  4.25  0.375
B17X19.3  B  16.63  0.188  4.25  0.250  4.25  0.375
B12X19.3  B  11.63  0.188  4.25  0.250  6.75  0.375
B15X19.6  B  14.63  0.188  4.25  0.250  5.25  0.375
B10X19.8  B   9.75  0.188  5.50  0.375
B16X20.2  B  15.63  0.188  4.25  0.250  5.50  0.375
B15X20.2  B  14.63  0.188  4.25  0.250  4.25  0.375
B14X20.4  B  13.50  0.250  5.50  0.250
B18X21.7  B  17.50  0.250  4.25  0.250
.
.
.
```

Figure 7-5

7.5 Design Steel Tables

The Design Steel Tables contain listings of shapes without section properties. These tables are used to indicate which sizes are to be considered in design. While the Beam Design module and the Column Design module share the same Master Steel Tables, they have separate Design Steel Tables.

The Design Steel Tables can be modified by adding, deleting or re-ordering members. When adding new members, it is important to keep in mind that all members in the Design Steel Tables must also be listed in the Master Steel Tables. If members are deleted from Design Steel Tables it is NOT necessary to delete them from the Master Steel Tables.

Table 7-1 lists the Master Steel Tables provided with the RAM Structural System, the shapes they contain and the units in which the section properties are listed. Table 7-2 and Table 7-3 list the Design Steel Tables provided with the RAM Structural Systems, the shapes they contain and their corresponding Master Steel Table.

Several Column Design Steel Tables are included with the RAM Steel Column Design Program (See Table 7-3**):**

- RAMAISC.COL contains the American shapes and includes only those shapes that are considered column shapes.

- RAMAISC2.COL is similar to RAMAISC.COL except it also includes shapes that would normally be considered beam shapes.
- RAMAISCM.COL is the metric equivalent of RAMAISC.COL.
- RAMA2M.COL is the metric equivalent of RAMAISC2.COL.
- RAMCAN.COL contains the Canadian shapes.
- RAMARCELOR.COL contains the ArcelorMittal shapes.
- RAMUK.COL contains the British shapes.
- RAMUK_CF.COL contains the British shapes like RAMUK.COL, except it contains the cold-formed hollow structural sections rather than the hot finished hollow sections.
- RAMAS.COL contains the Australian shapes.

Sections are grouped by member type (e.g. W14's) and these groups are referred to as "Trial Groups". Within the groups, sections are sorted by weight in ascending order. Using Trial Groups, the program can optimize up to three unique designs in one run. The first three groups that appear in the Design Steel Table are taken as the default Trial Groups. It is recommended that the three Trial Groups used most often by the Column Design Module be moved to the top of the Column Design Steel Table so that they will be the default Trial Groups. Sections within Trial Groups may also be re-ordered to meet the needs of the engineer.

Several Beam Design Steel Tables are included with the RAM Steel Beam Design Program. See Table 7-2. In these tables, sections are grouped into I-shaped members, Square and Rectangular Hollow Sections and Channels. Within the groups, sections are sorted by weight in ascending order. When the Beam Design module selects the optimal beam, it starts at the top of the file and works it way down until it finds the first beam that satisfies all criteria. While the order of the groups and the order of sections within groups can be re-ordered, the Beam Design's selection scheme should be kept in mind when doing so.

It is recommended that edited files be given different names than the original files to distinguish them from the originals. Otherwise, a backup copy of the file should be made before editing so the original file can be recovered if desired.

RAM Structural System Steel Tables

Table 7-1 Master Tables

Table Name	Shapes Contained	Units
RAMAISC.TAB	AISC shapes: W, M, S, C, MC, L, HSS, WT, MT, ST, Flat bar, Round bar	English
RAMAISCM.TAB	AISC shapes, metric equivalent of RAMAISC.TAB	English
RAMCAN.TAB	Canadian shapes	SI
RAMUK.TAB	British shapes	SI
RAMUK_CF.TAB	British shapes, identical to RAMUK.TAB except with cold-formed hollow structural sections	SI
RAMARCELOR.TAB	ArcelorMittal European shapes: IPE, IPN, HE, HL, HD, HP, W, UAP, UPE, UPN, U, UE, L, Flat bar, Round bar	SI
RAMAS.TAB	Australian shapes	SI

Table 7-2 Beam Design Tables

Table Name	Shapes Contained	Master Table
RAMAISC.BMS	AISC shapes: W, M, C, MC, HSS	RAMAISC.TAB
RAMS.BMS	AISC S-shapes	RAMAISC.TAB
RAMAISCM.BMS	AISC shapes, metric equivalent to RAMAISC.BMS	RAMAISCM.TAB
RAMSM.BMS	AISC S-shapes, metric equivalent to RAMS.BMS	RAMAISCM.TAB
RAMCAN.BMS	Canadian shapes: W, C, HS	RAMCAN.TAB
RAMCANWF.BMS	Canadian WWF shapes	RAMCAN.TAB
RAMUK.BMS	British shapes: UB, C, SHS, RHS	RAMUK.TAB
RAMUK_CF.BMS	British shapes: UB, C, cold-formed SHS, cold-formed RHS	RAMUK_CF.TAB
RAMARCELOR.BMS	ArcelorMittal European shapes: IPE, IPN, HE, HL, HD, HP, W, UAP, UPE, UPN, U, UE	RAMARCELOR.TAB
CORUSADV.BMS	Corus Advance shapes: UKB, UKC, UKPFC	RAMUK.TAB
RAMAS.BMS	Australian Shapes: UB, UC, WB, WC, PFC	RAMAS.TAB

Table 7-3 Column Design Tables

Table Name	Sections Contained	Master Table
RAMAISC.COL	AISC shapes, column sections only: W, HSS, Pipe	RAMAISC.TAB
RAMAISC2.COL	AISC shapes, all sections: W, HP, HSS, Pipe	RAMAISC.TAB
RAMAISCM.COL	AISC shapes, metric equivalent to RAMAISC.COL	RAMAISCM.TAB

RAMA2M.COL	AISC shapes, metric equivalent to RAMAISC2.COL	RAMAISCM.TAB
RAMCAN.COL	Canadian shapes: W, WWF, HP, HS	RAMCAN.TAB
RAMUK.COL	British shapes: UC, SHS, RHS, CHS, Corus Advance shapes	RAMUK.TAB
RAMUK_CF.COL	British shapes: UC, cold-formed SHS, cold-formed RHS, cold-formed CHS, Corus Advance shapes	RAMUK_CF.TAB
RAMARCELOR.COL	ArcelorMittal European shapes: HD, HE, HL, HP, IPE, IPN, W	RAMARCELOR.TAB
RAMAS.COL	Australian Shapes: UC, WC	RAMAS.TAB

In addition to modifying the existing tables, new Design Steel Tables can be created. When creating new tables, it is imperative that they be created in the format described in the following sections. There is no limit to the number of tables that the RAM Structural System can support.

The tables may be in either English or SI units.

The tables must be located in the Tables directory.

7.6 Column Design Steel Table Format

Files of column shapes to be used by the RAM Steel Column Design module may be created and modified by the user. The file must be edited using a simple text editor, not a word processor as it might embed control characters into the file. Use spaces, not tabs, between data items. The file name must have the extension ".COL".

The file has the following format:

The first line is an optional comment line indicating which Master Table is to be used in conjunction with the Design Table. Comment lines must start with a semicolon (;).

The Trial Group sections follow. Each section begins with a heading with the following format:

```
Group Shape
```

where:

Group is the Group Label (e.g., W14),
Shape is the Group Shape.
I indicates wide flange and other I shapes ('W' used in previous versions is also acceptable),
T indicates TS and Square and Rectangular Hollow Sections,

P indicates pipe and Round Hollow Sections,
C indicates channel,
A indicates single angle,
E indicates tee,
R indicates round bar,
B indicates flat bar or plate, and
L indicates double angles.

Some of these shapes are only used for hanging columns, not standard columns.

After the heading, the members within that Trial Group are list in ascending order by weight.

Groups are separated by a blank line.

Figure 7-6 is a sample listing from the RAMAISC.COL file.

```
;MasterTable = RAMAISC.TAB
W14  I
W14X43
W14X48
W14X53
.
.
.
W12  I
W12X40
W12X45
W12X50
.
.
.
HSS16X16  T
HSS16X16X1/2
HSS16X16X5/8
.
.
.
HSS20  P
HSS20.000X0.375
HSS20.000X0.500
.
.
```

Figure 7-6

7.7 Beam Design Steel Table Format

Files of beam shapes to be used by the RAM Steel Beam Design module may be created and modified by the user. The file must be edited using a simple text editor, not a word processor as it might embed control characters into the file. Use spaces, not tabs, between data items. The file name must have the extension ".BMS".

The file has the following format:

The first line is an optional comment line indicating which Master Table is to be used in conjunction with the Design Table.

The section groups follow. Groups are separated by a blank line. Each section begins with a heading with the following format:

```
Group
```

where:

Group is the Group Shape
I indicates I-beams,
TS indicates TS and Square and Rectangular Hollow sections, and
Channel indicates Channels).

Each subsequent line has the following format:

```
Label Flag
```

where:

Label is the member size (e.g. W8x10).
Flag is a single character that indicates how the program will use the member.
" / " indicates that the size is to be ignored. It has the same effect as deleting the member from the table.
" * " indicates that the size is considered a "common" size. When selecting the optimum beam size the program investigates only the asterisked sizes when designing simple composite beams, or when designing any beam when the criteria "Check Unbraced Length" is not selected, unless there is a beam depth restriction. This speeds up the design of beams significantly because it allows the program to ignore unusual sizes when designing for simple conditions. Thus, when the table is to be used to design such members, at least some of the sections must have an asterisk. In the RAMAISC.BMS table supplied with the program, the asterisked sizes are those shown in bold in the "W Shapes – Selection by Zx" tables.
" + " indicates that the size is to be considered in optimization except as explained for "*". When there is a depth restriction or an unbraced length consideration the program investigates sizes flagged by either a plus sign or an asterisk until an acceptable size is found. The plus symbol should be used for any sizes (e.g., small sizes) that the user does not want designed as composite beams.

The order in which the members are listed is critical to the proper performance of the program. The members must be listed in the optimum order (least expensive, generally least weight), beginning with the smallest member and ending with the largest member. RAM Steel Beam Design selects the first member it encounters that satisfies all design requirements and constraints.

If there is no size in the table which satisfies all of the design criteria and requirements for a given member, the program will indicate **No Design** on the output, View/Update, and on-screen. The user must then take whatever action is appropriate.

The file may contain 2000 member sizes.

Figure 7-7 is a sample listing from the RAMAISC.BMS file:

```
;MasterTable=RAMAISC.TAB
I
.
.
.
W6X9   /
M12X10 /
W8X10  *
M12X10.8 /
M12X11.8 /
W10X12 *
W6X12  /
W8X13  +
.
.
.

Channel
C3X4.1    /
C3X5      /
C4X5.4    +
C3X6      /
MC10X6.5  +
C5X6.7    +
.
.
.

TS
HSS2X1X1/8      /
HSS3X1X1/8      /
HSS2.5X1.5X1/8  /
HSS2X1X3/16     /
.
.
.
```

Figure 7-7

7.8 Steel Joist Tables - Standard Joists

For Standard joists, those loaded with a single uniform load, RAM Steel Beam Design is capable of selecting steel joists from joist load tables. Four joist tables are supplied with the Beam Design module. RAMSJIK.JST, RAMSJILH.JST, and RAMSJIDH.JST contain the Steel Joist Institute K-series, LH-series and DLH-series respectively. The RAMSJI.JST file is a combination of the three previously mentioned files.

The tables are based on the load tables published by S.J.I. The members in these tables are ordered most economically by cost. Since this varies from manufacturer to manufacturer the user should review the joist table files and make adjustments as appropriate.

The tables may be in either English or SI units.

The tables must be in the Tables directory.

Steel joist tables can be created or modified by the user. The file must be edited using a simple text editor, not a word processor as it might embed control characters into the file. Use spaces, not tabs, between items. The file must have the extension ".JST".

The file has the following format:

The character in the first line indicates the system of units used in the table: an E indicates English units and an S indicates SI units.
For each joist a section is created with the following format:

```
Label Depth W
Length1 TL1 LL1
Length2 TL2 LL2
Length3 TL3 LL3
   .
   .
   .
Lengthn TLn LLn
```

where:

Label is the name of the joist (13 characters or less),
Depth is the depth (in. or mm),
W is the weight per unit length (lb/ft or kg/m),
Length*i* is the length for which the uniform load capacity is TL*i* and LL*i* (ft or m),
TL*i* is the total uniform load capacity (lb/ft or kN/m),
LL*i* is the uniform live load capacity which causes a deflection of L/360, not to exceed TL*i* (lb/ft or kN/m).

Joists are listed in order of ascending cost. For each joist, each line of Length, TL, and LL is listed in order of ascending length.

Each joist section is separated by a blank line.
Figure 7-8 is an example listing of a portion of a joist file.

```
E
10K1 10 5.0
10 550 550
11 550 542
   .
   .
   .
20 199 97

12K1 12 5.0
12 550 550
13 550 510
   .
```

```
   .
   .
24 166 81
```

Figure 7-8

If no member in the table is adequate for a given member, the last joist in the table will be selected whether it is adequate or not. It is necessary, therefore, that a "dummy" joist be included at the end of the table. The "dummy" joist label will serve as a flag that no sizes were found adequate for the conditions. The weight of the dummy joist should be 0.0 so that it is not included in the takeoff, and the depth of the member should be 0.0 so that it is not affected by depth restrictions. An example of such an entry is shown below:

```
xxSJxx 0 0.0
9999 0 0
```

The file may contain 2000 different joist sizes and a maximum of 8100 joist entries.

7.9 Steel Joist Tables - Constant Shear Joists

For joists with variable loading, RAM Steel Beam Design is capable of selecting steel joists from constant shear joist load tables. The table RAMKCS.CSJ is supplied with the Beam Design module. It contains the Steel Joist Institute KCS-series joists.

The table is based on the load tables published by S.J.I. The members in this table are ordered by economy, in this case by least weight. Since this may vary from manufacturer to manufacturer the user should review the joist table file and make adjustments as appropriate.

The tables may be in either English or SI units.

The tables must be in the Tables directory.

Constant Shear Joist tables can be created or modified by the user. The file must be edited using a simple text editor, not a word processor as it might embed control characters into the file. Use spaces, not tabs, between items. The file must have the extension ".CSJ".

The Constant Shear Joist table has the following format:

The character in the first line indicates the system of units used in the table: an E indicates English units and an S indicates SI units.

Each subsequent line lists the joist information:

```
Label  Depth  Wt  I  M  V  w
```

where:

Label is the name of the joist (13 characters or less),
Depth is the depth (in. or mm),

Wt is the weight per unit length (lb/ft or kg/m),
I is the moment of inertia (in**4 or cm**4),
M is the Moment capacity (k-ft or kNm),
V is the Shear capacity (kip or kN),
w is the maximum total uniform load (lb/ft or N/m).

A portion of the KCS table, RAMKCS.CSJ, is shown in Figure 7-9:

```
E
12KCS1  12   6.0   43  17.417  2.4  550
10KCS1  10   6.0   29  14.333  2.0  550
14KCS1  14   6.5   59  20.583  2.9  550
10KCS2  10   7.5   37  18.750  2.5  550
14KCS2  14   8.0   77  27.000  3.4  550
12KCS2  12   8.0   55  22.833  3.0  550
16KCS2  16   8.5   99  29.083  4.0  550
.
.
.
30KCS5  30  21.0  934 152.750  9.2  550
xxSJxx   0   0.0 99999 99999 99999 99999
```

Figure 7-9

If no member in the table is adequate for a given member, the last joist in the table will be selected whether it is adequate or not. It is necessary, therefore, that a "dummy" joist be included at the end of the table. The "dummy" joist label will serve as a flag that no sizes were found adequate for the conditions. The weight of the dummy joist should be 0.0 so that it is not included in the takeoff, and the depth of the member should be 0.0 so that it is not affected by depth restrictions. Large values should be specified for the capacities. In the above example, a joist with a label of xxSJxx has been included.

7.10 Smartbeam Tables – Castellated

A table of CMC (formerly SMI) castellated Smartbeams, called RAMSMI.CAS, is provided with the program. Although tables can be created and edited by the user, it is not recommended that the user modify the RAMSMI.CAS table unless directed by the manufacturer, CMC. If the file is edited, it must be done so using a simple text editor, not a word processor which may embed format and control characters into the file. Use spaces, not tabs, between data items. The file name must have the extension ".CAS". Note that the format for the castellated Smartbeams is different than that of the cellular Smartbeams (see Section 7.11).

The file has the following format:

The first line contains an E or S, indicating English or SI units.

The Smartbeam sections are then listed in the following format:

```
Label TopSize BotSize emin emax dtmin dtmax dtstandard flag
```

where:

Label is the designation for the castellated Smartbeam (e.g., CB18x14).
TopSize is the label of the wide-flange section used for the top (e.g.,W12 X14).
BotSize is the label of the wide-flange section used for the bottom (e.g., W12X16).
emin is the minimum e to be considered.
emax is the maximum e to be considered.
dtmin is the minimum dt to be considered.
dtmax is the maximum dt to be considered.
dtstandard is the standard (ideal) dt to be considered.
flag is a single character used to indicate whether the member is to be considered for composite or noncomposite design:
"-" means consider for noncomposite only,
"*" means consider for composite only,
"+" means consider for composite or noncomposite, and
"/" means do not use (this is a way for the user to remove a size from consideration without actually deleting it from the table).

Each label must be unique (the same label can't appear twice in the table) and it is limited to a maximum of 15 characters.

The sections must be listed in order of cost, least expensive first (generally the lightest). RAM Steel Beam Design selects the first member in the table that satisfies all design requirements and constraints.

The values of emin, emax, dtmin, and dtmax are used by the program to limit the number of hole configurations that are considered.

Figure 7-10 is a sample listing from the RAMSMI.CAS file:

```
E
CB12x10     W8x10     W8x10    3.0  5.000  1.750  2.875  2.25      +
CB15x12     W10x12    W10x12   3.0  5.625  1.875  3.125  2.50      +
CB12x13     W8x13     W8x13    3.0  6.875  1.750  2.875  2.25      +
CB15x12/15  W10x12    W10x15   3.0  5.625  1.875  3.125  2.50      *
CB18x14     W12x14    W12x14   3.0  6.000  2.250  3.750  3.00      +
CB15x12/17  W10x12    W10x17   3.0  5.625  1.875  3.125  2.50      *
CB15x15     W10x15    W10x15   3.0  6.875  1.875  3.125  2.50      +
CB12x15     W8x15     W8x15    3.0  7.250  1.750  2.875  2.25      +
CB18x14/16  W12x14    W12x16   3.0  6.000  2.250  3.750  3.00      *
.
.
.
```

Figure 7-10

7.11 Smartbeam Tables – Cellular

A table of CMC (formerly SMI) cellular Smartbeams, called RAMSMI.CEL, is provided with the program. Although tables can be created and edited by the user, it is not recommended that the user modify the RAMSMI.CEL table unless directed by the manufacturer, CMC. If the file is edited, it must be done so

using a simple text editor, not a word processor which may embed format and control characters into the file. Use spaces, not tabs, between data items. The file name must have the extension ".CEL". Note that the format for the cellular Smartbeams is different from that of the castellated Smartbeams.

The file has the following format:

The first line contains an E or S, indicating English or SI units.

The Smartbeam sections are then listed in the following format:

```
Label TopSize BotSize Dostandard flag
```

where:

Label is the designation for the cellular Smartbeam (e.g., LB18x14).
TopSize is the label of the wide-flange section used for the top (e.g., W12X14).
BotSize is the label of the wide-flange section used for the bottom (e.g., W12X16).
Dostandard is the standard (ideal) hole diameter Do to be considered.
flag is a single character used to indicate whether the member is to be considered for composite or noncomposite design:
"-" means consider for noncomposite only,
"*" means consider for composite only,
"+" means consider for composite or noncomposite, and
"/" means do not use (this is a way for the user to remove a size from consideration without actually deleting it from the table).

Each label must be unique (the same label can't appear twice in the table) and it is limited to a maximum of 15 characters.

The sections must be listed in order of cost, least expensive first (generally the lightest). RAM Steel Beam Design selects the first member in the table that satisfies all design requirements and constraints.

Figure 7-11 is a sample listing from the RAMSMI.CEL file:

```
E
LB12x10      W8x10     W8x10      7.500    +
LB15x12      W10x12    W10x12    10.000    +
LB12x13      W8x13     W8x13      7.500    +
LB15x12/15   W10x12    W10x15    10.000    *
LB18x14      W12x14    W12x14    12.000    +
LB15x12/17   W10x12    W10x17    10.000    *
LB15x15      W10x15    W10x15    10.000    +
LB12x15      W8x15     W8x15      7.500    +
LB18x14/16   W12x14    W12x16    12.000    *
LB15x12/19   W10x12    W10x19    10.000    *
.
.
.
```

Figure 7-11

7.12 Westok Cellular Beam Tables

Two tables for design of Westok Cellular Beams, called RAMUK.WCB (containing the standard British UB and UC shape) and RAMARCELOR.WCB (containing the Arcelor I-shapes), are provided with the program. Although additional tables can be created and edited by the user, it is not recommended that the user modify these .WCB tables unless directed by the manufacturer, Westok. If the file is edited, it must be done so using a simple text editor, not a word processor which may embed format and control characters into the file. Use spaces, not tabs, between data items. The file name must have the extension ".WCB".

The file has the following format:

The first line contains an E or S, indicating English or SI units. This is not currently used but is specified here to allow future possible expansion of the tables.

The beam sections that compose the Westok Cellular Beams are then listed in the following format:

```
Label flag
```

where:

Label is the designation for the I-shaped beam (e.g., UB127x76x13).
flag is a single character. This is not currently used but is specified here to allow future possible expansion of the tables.

Each label must be unique (the same label can't appear twice in the table) and it is limited to a maximum of 15 characters.

The sections should be listed in order of cost, least expensive first (generally the lightest). RAM Steel Beam Design selects the first member in the table that satisfies all design requirements and constraints.

Figure 7-12 is a sample listing from the RAMSMI.CEL file:

```
S
UB127x76x13   +
UB152x89x16   +
UB178x102x19  +
UB254x102x22  +
UB203x102x23  +
UC152x152x23  +
UB305x102x25  +
UB254x102x25  +
UB203x133x25  +
UB305x102x28  +
UB254x102x28  +
UB203x133x30  +
.
.
.
```

Figure 7-12

7.13 Connection Check Tables

When the **Print - Connection Check** command in the RAM Steel Beam Design module is invoked, the maximum reaction at each end of each ***gravity*** Steel beam (except those with cantilevers) is compared with the capacity for that beam size of the typical connection as defined by the user in a Connection Check Table. If the reaction exceeds the allowable reaction listed in the Connection Check Table, the beam, its reaction and its connection capacity is listed on the report.

Since there is no "standard" typical connection detail, no tables are supplied with the program. If the **Reports - Connection Check** command is to be used, Connection Check Tables must be created by the user, based on the capacity of the connections used on the project. Any number of tables may be created corresponding to different types of connections. When the command is invoked the user may then select the connection table appropriate for the current model. In order to better remember the connection associated with a given tables, it is recommended that the tables be given names which identify the connection. For example, SPA325N.CON would indicate a shear plate connection using A325N bolts, EPA307.CON would indicate an end plate connection using A307 bolts, etc. The tables must have the extension ".CON".

Connection Check tables may be available from third-party vendors of proprietary connections. These files will have the extension ".COX". They are encrypted files and cannot be edited by the user, but they can be selected for use in the Connection Check feature.

The Connection Check can use either factored or unfactored reactions, as specified within the table itself. The values in the table should list the most conservative value of the connection capacity for any condition to be encountered in the model. The program does not take into account the possible differences in the connection capacity, for example, of a beam with a yield strength Fy of 36 ksi versus one with a yield strength of 50 ksi. It merely compares the actual beam reaction with the value listed in the Connection Check table. If a model contains beams with a mix of yield strengths, the user can conservatively use the table based on lower strength, or invoke the **Print - Connection Check** command twice, using different connection tables. The command can be invoked any number of times, each time using a different connection table. The listing of beams on the output that do not apply to a particular connection type can then be ignored.

The connection tables have the following format:
An optional Description section is at the beginning of the file. It begins with a line containing the text [Start] and ends with a line containing the text [End]. Since these comments are optional, if the first line is not [Start], the program will assume that there is no Description Text. The Description text can include a description of the connections upon which the table was based. The Description text will be printed at the beginning of the report. A virtually unlimited number of lines of text is permitted.

The first line of the file (or the first line after the Description section) contains an E or S, indicating English (US Imperial) or SI units.

The next line of the file contains a U or F, indicating Unfactored or Factored capacity values.

Each subsequent line has the following format:

```
Label Capacity Comment
```

where:

Label is a beam label or group label, and
Capacity is the associated connection capacity. The capacities should be in kips for English units and kN for SI units.
Comment is an optional text string that can be used to describe the particular connection. It may contain up to 50 characters, but should generally be brief. It will be included on the report.

The entries should be separated by spaces, not by tabs.

Shown below is an example of part of a table:

```
[Start]
Shear tab connection capacities using 3/4" A325 bolts.
Shear tabs: A36
Beams: A992 Fy=50
[End]
E
U
W8      9.6   2 bolts
W10     9.6   2 bolts
W12    18.5   3 bolts
W14    18.5   3 bolts
W16    28.8   4 bolts
```

In this case, for example, all W16 beams would be compared against an unfactored reaction of 28.8 kips. Those beams whose reaction exceeded this value would be listed on the output.

Individual beam sizes may also be included in the table:

```
[Start]
Shear tab connection capacities using 3/4" A325 bolts.
Beams: A992 Fy=50
Shear tabs: A36
[End]
E
U
W8       9.6   2 bolts
W10      9.6   2 bolts
W12X14  16.3   3 bolts
W12X16  16.8   3 bolts
W12     18.5   3 bolts
W14     18.5   3 bolts
```

In this case, for a W12x14 the capacity is 16.3 kips, for a W12x16 it is 16.8 kips, and for all other W12's it is 18.5 kips. Note that the order is important. For example, if “W12” was listed *before* “W12X14”, all W12’s, even the W12X14’s would use the value listed for “W12”; the value subsequently listed for “W12X14” would be ignored. Also be aware that if an entry is made for “W4” and then subsequently an entry is made for “W40” and/or “W44”, all W40’s and W44’s will be checked against the value listed for “W4” since that is the first entry that matched a portion of the member label. Instead of listing “W4”, list “W4X”.

The designations in the connection table must match exactly the designations in the beam design table. For example, if the sizes in the beam design table use upper case letters, the connection table must also use upper case letters. The beam design tables supplied by RAM International use upper case letters.

If there is no corresponding capacity in the Connection Check table for a particular beam size, that size will be listed at the end of the report with a note indicating that there was no capacity given.

The units in the table do not need to match the units in the model. The program will automatically make the appropriate conversions if necessary.

The capacities may be listed as either factored or unfactored capacities. If the table indicates factored capacities, the program will compare those values against the factored reactions, where the factors used are those associated with the current Beam Design Code selection (as specified in **Criteria – Steel Design Codes** in the RAM Steel Beam Design module). Note: if the current Beam Design Code is AISC ASD, a table with factored capacities should not be selected, since there would be a conflict between using factored capacities and the unfactored ASD reactions, and would be unconservative.

Files must be in the Tables subdirectory.

7.14 Concrete Reinforcement Tables

The Concrete Reinforcement Tables contain the reinforcement geometry information used for concrete design in RAM Concrete and RAM Foundation. The Concrete Reinforcement Table is assigned in RAM Manager.

Two Concrete Reinforcement Tables are provided with the RAM Structural System. RAMASTM.REN contains the ASTM reinforcement bar definitions for English units, RAMASTMM.REN contains the ASTM bars for SI or metric units.

The Concrete Reinforcement Tables are in text file format and can be edited, allowing the engineer to customize the tables. Bar sizes are listed in order of increasing size. Sizes not included in the original files can be added and those that are not used can be removed. In the RAM Concrete and RAM Foundation modules there is also an option to select which bars are used for flexure and which are used for shear reinforcement.

It is recommended that edited files be given different names than the original files to distinguish them from the originals. Otherwise, a backup copy of the file should be made before editing so the original file can be recovered if desired.

7.15 Concrete Reinforcement Table Format

Concrete reinforcement bar size files to be used by RAM Concrete and RAM Foundation may be created and modified by the user. The file must be edited using a simple text editor, not a word processor as it might embed control characters into the file. Use spaces, not tabs, between data items. The file name must have the extension ".REN".

The file has the following format:

The first line contains an E or S, indicating English or SI units.

Each subsequent line lists each reinforcement bar size with the following format:

```
Name  Area  Diam
```

where:

Name is the name of the bar size (e.g., #7).
Area is the bar cross-sectional area (sq in. or sq mm.).
Diam is the bar Diameter (in. or mm.).

Any information after a double slash "//" will be considered a comment and ignored by the program.

Below is a copy of RAMASTM.REN file:

```
E
//US - English unit bars
//Name Area Diam
 #3 0.11 0.375
 #4 0.20 0.500
 #5 0.31 0.625
 #6 0.44 0.750
 #7 0.60 0.875
 #8 0.79 1.000
 #9 1.00 1.128
#10 1.27 1.270
#11 1.56 1.410
#14 2.25 1.693
#18 4.00 2.257
```

7.16 Pan Form Tables

The Pan Form Tables contain the Pan Form geometry information used to generate concrete pan joist and beam systems in the Modeler for design in RAM Concrete. The Pan Form Table is assigned in RAM Manager.

One Pan Form Table is provided with the RAM Structural System. RAMCECO.PNJ contains the pan form dimensions for pans produced by CECO Concrete, LLC. Tables for Pan forms produced by other suppliers can easily be created.

The Pan Form Tables are in text file format and can be edited, allowing the engineer to customize the tables. Pans may be listed in any convenient order. Sizes not included in the original files can be added and those that are not used can be removed.

It is recommended that edited files be given different names than the original files to distinguish them from the originals. Otherwise, a backup copy of the file should be made before editing so the original file can be recovered if desired.

7.17 Pan Form Table Format

Pan Form Table files to be used by RAM Modeler to create pan joist and beam systems to be designed in RAM Concrete may be created and modified by the user. The file must be edited using a simple text editor, not a word processor as it might embed control characters into the file. Use spaces, not tabs, between data items. The file name must have the extension ".PNJ".

The file has the following format:

The first line contains an E or S, indicating English or SI units.

Each subsequent line lists a Pan Form label and size with the following format:

```
Name  Width  Depth
```

where:

Name is the name of the Pan.
Width is the width of the Pan (in. or mm.).
Depth is the depth of the Pan (in. or mm.).

Any information after a double slash "//" will be considered a comment and ignored by the program.

Below is a copy of RAMCECO.PNJ file:

```
E
//English unit CECO Pans
//Label Width Depth
14x66 66.0 14.0
16x66 66.0 16.0
20x66 66.0 20.0
24x66 66.0 24.0
16x53 53.0 16.0
20x53 53.0 20.0
24x53 53.0 24.0
14x30 30.0 14.0
16x30 30.0 16.0
20x30 30.0 20.0
24x30 30.0 24.0
14x20 20.0 14.0
16x20 20.0 16.0
20x20 20.0 20.0
```

7.18 RAM Frame Response Spectra Tables

RAM Frame is capable of performing a dynamic response spectrum analysis, utilizing tables of ground motion response spectra curves. The tables supplied with the program contain the values from the NEHRP and UBC Normalized Response Spectra curves and the El Centro Earthquake Response Spectrum curve. For NEHRP and UBC, one table is provided for each soil type (S1, S2, and S3).

Response Spectra table selection is made when the data for the Dynamic Response Spectra load case is defined in RAM Frame.

Files of Response Spectra curves to be used by RAM Frame may be created and modified by the user. The file must be edited using a simple text editor, not a word processor that embeds control characters into the file. Use spaces, not tabs, between data items. The file name must have the extension ".RSP".

The file has the following format:

Any number of lines at the beginning may be used as comment lines. These are indicated by double backslash (\\) at the beginning of each comment line.

Each subsequent line has the following format:

```
Period  Accel
```

where:

Period is the period, in seconds.
Accel is the Spectral Acceleration corresponding to the period. It may be the normalized value.

The values should be in order of increasing period. There should be a pair of values sufficiently large for whatever magnitude of structural periods may be encountered.

Shown below is a portion of the NEHRP_S1.RSP file:

```
\\NEHRP Normalized Response Spectra
\\Damping = 0.05
\\ Soil Profile Type S1
0.00    1.00
0.02    1.00
0.11    2.50
0.40    2.50
0.50    2.00
.
.
.
2.63    0.27
2.75    0.24
2.88    0.22
  99    0.22
```

7.19 Fabricators and Suppliers

This section contains a list of organizations, fabricators and suppliers whose products are included in tables supplied with the RAM Structural System, and is provided as a convenience. For more information on their products, please consult them directly.

Steel Shapes

American Institute of Steel Construction
1 East Wacker Drive, Suite 3100
Chicago, IL 60601
(312) 670-2400

ArcelorMittal
www.arcelormittal.com

CMC Steel Products
2305 Ridge Road
Suite 202
Rockwall, TX 75087
(800) 308-9925

Steel Joists

Steel Joist Institute
3127 10th Avenue North Ext.
Myrtle Beach, SC 29577
(803) 626-1995

Steel Deck

ASC Steel Building Products
2110 Enterprise Boulevard
West Sacramento, CA 95691
(800) 726-2727

Verco Manufacturing Co.
P.O. Box 508
Benicia, CA 94510
(707) 745-9658

Vulcraft
Alabama - (205) 845-2460
Indiana - (219) 337-5411
Nebraska - (402) 644-8500
South Carolina - (803) 662-0381
Utah - (801) 734-9433
Texas - (409) 687-4665

Appendix A: The RAM Defaults Utility

The RAM Defaults Utility can be used to change the program defaults that are saved in the RAMIS.INI file. These defaults are used when creating new models with the RAM Structural System. Note that changing the defaults with this utility will not affect any existing models.

The RAM Defaults Utility provides a list of all of the Defaults dialog boxes sorted by program. Each item is preceded by a colored checkbox. A blue checkbox indicates that the defaults in the corresponding dialog box are new for the current version and should be set if this is the first time the utility has been run. A gray checkbox indicates that the defaults in the corresponding dialog box existed in previous versions of the RAM Structural System.

Double-clicking on any item in the list invokes the corresponding dialog box, allowing the current program defaults to be inspected and modified. To modify the current defaults, make the desired changes and click the **OK** button. To retain the initial settings, click the **Cancel** button. **Back** brings up the previous Defaults dialog box without saving the settings in the current dialog box. **Next** accepts the changes to the current dialog box and brings up the next Defaults dialog box. After a dialog box has been visited, a green checkmark will appear in the checkbox next to the name of that dialog box in the list.

Appendix B: CIS/2 and the CIS Translator

The RAM Structural System offers the ability to export a RAM Structural model to a neutral data exchange file. This new data exchange file is based on the CIMsteel Integration Standards Release 2 (CIS/2).

The CIMsteel Integration Standards

The CIS/2 is a set of formal specifications developed to help reduce duplication of effort in the steel design and construction industry. It defines a standard neutral data exchange file format for the sharing of engineering and construction information. The data is stored in neutral data structures in a standardized file format that is independent of the particular application that created the data. This allows for data interchange between any two software applications that have CIS/2 translators. For example, a model can be designed and analyzed in one software application, exported to a CIS/2 file, then imported into a detailing package, eliminating the need to recreate the model in the second application.

The CIS/2 is based upon a Logical Product Model (LPM/5) that defines the neutral data structures of the CIS/2 file. This LPM determines the scope of the engineering information that can be shared among CIS/2 compatible applications. The LPM encompasses more engineering data than does any one application, from the planning stages of the structure, all the way through construction. Thus, each software package will only import and export a subset of the entire LPM/5.

Valid subsets of the LPM/5 that can be exchanged between applications are known as Conformance Classes (CCs). Two CIS/2 compliant software packages are able to interchange data when they support overlapping Conformance Classes. A list of the CCs that an application supports should be available with the application's documentation.

Visit the AISC website for more information on the CIMsteel Integration Standards (www.aisc.org).

RAM CIS/2 Export Translator

The RAM Structural System CIS Export Translator supports the second release of the CIMsteel Integration Standards (CIS/2).

The translator currently supports the export of:

- beams, columns, braces and joists
- member shape
- material properties
- end conditions
- end reactions

The translator currently does not export the following information:

- Smartbeams
- walls
- loads
- design criteria
- analysis results

By default, all beams are exported so that the column joint is at the top of steel of beams. The RAM Structural System analyzes the structure with the assumption that the column joint is at the centerline of beams. The column joint option can be changed under Advanced Options during export.

End reactions are exported for gravity beams. Additionally, (under Advanced Options), the engineer may specify a minimum reaction for export as well as a scale factor by which all exported reactions will be multiplied.

Joist girders are exported at the same elevation as beams, while standard joists are exported so that the joist seat rests on top of the supporting girders.

The RAM CIS/2 Translator supports the following CIS/2 Conformance Classes:
CC003, CC005, CC014, CC019, CC024, CC026, CC029, CC030, CC031, CC032, CC034, CC035, CC110, (CC166, +CC167), CC170, CC305, CC306, (CC177, +CC307), CC310, CC325, CC327, CC331

Appendix C: The RAM Steel Table Editor

The RAM Structural System utilizes a series of steel tables to determine appropriate steel section properties and to facilitate optimization of beams and columns. These tables can be customized to impose some control over the optimized design that the program performs. The three tables utilized by the RAM Structural System in its steel design are the Master steel table, the Beam steel table and the Column steel table as described in Chapter 6, RAM Structural System Tables.

RAM Structural System Steel Tables

Master Steel Table

The Master Steel Tables contain all of the section properties for each available section. The Analysis and Design modules access these tables to gather information for calculations.

Beam Steel Table

Files of beam shapes to be used by the RAM Steel Beam Design module may be created and modified by the user. This table controls what sections the Beam Design considers when optimizing a member. The order in which the beams are listed is critical to the design optimization results. The Beam table includes a flag with each section to determine how the section is considered during optimization. The flags are used as follows:

Common (* in table): Indicates that the size is considered a "common" size. When selecting the optimum beam size the program investigates only the common sizes when designing simple composite beams, or when designing any beam when the criteria "Check Unbraced Length" is not selected, unless there is a beam depth restriction. This speeds up the design of beams significantly because it allows the program to ignore unusual sizes when designing for simple conditions. Thus, when the table is to be used to design such members, at least some of the sections must have an asterisk (be considered common). In the RAMAISC.BMS table supplied with the program, the asterisked sizes are those shown in bold in the "W Shapes – Selection by Zx" tables in the AISC steel manuals.

Optimization (+ in table): Indicates that the size is to be considered in optimization except as explained for Common sections. When there is a depth restriction or an unbraced length consideration the program investigates sizes flagged by either a plus sign (Optimization) or an asterisk (Common) until an acceptable size is found. The optimization flag should be used for any sizes (e.g., small sizes) that the user does not want designed as composite beams.

Ignore (/ in table): Indicates the size is to be ignored. It has the same affect as deleting the member from the table.

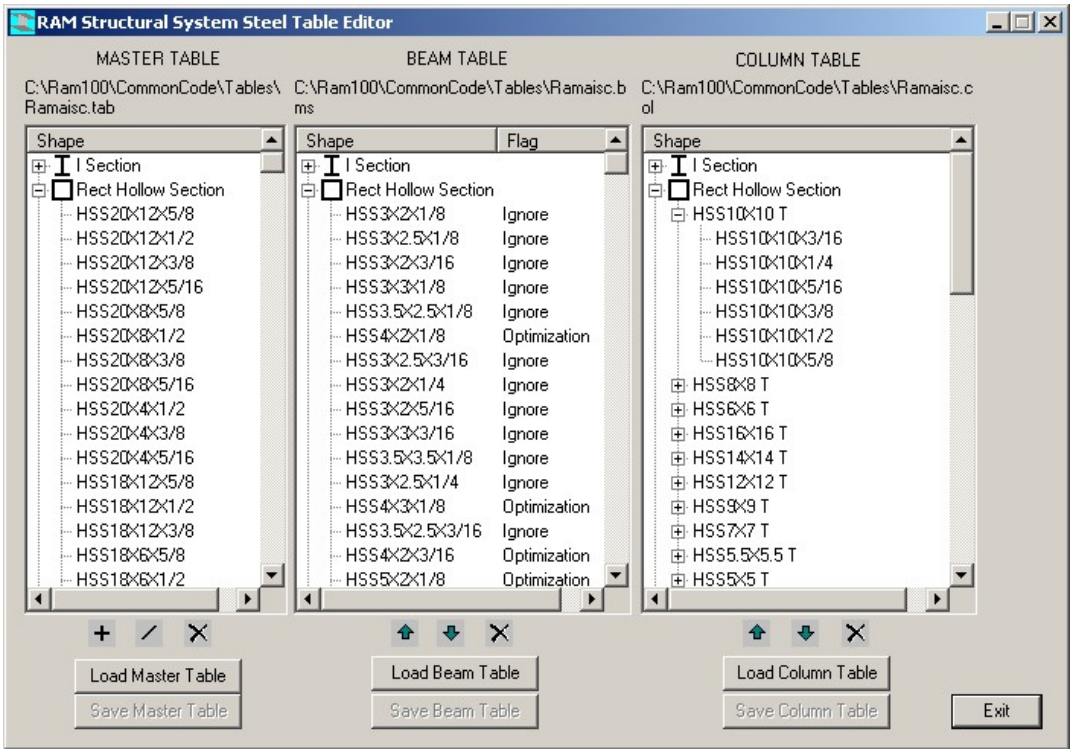
Column Steel Table

Files of column shapes to be used by the RAM Steel Column Design module to optimize columns from may be created and modified by the user. The order in which the beams are listed is critical to the design optimization results.

The Steel Table Editor is a utility provided for editing these three tables: The Master, Beam, and Column steel tables. The application works as both a standalone application and can also be launched from RAM Manager. When launched from the Manager the tables selected for the current active model will be automatically loaded in Table Editor. If the application is run as a standalone, the user must load the tables to be edited.

Using Steel Table Editor

The interface to the RAM Steel Table Editor is illustrated below. This illustration shows the table with three tables already loaded. The names of the table files in use are displayed above the lists. Each list contains a tree view which when expanded lists the sections, the order of the sections and any flags associated with the section. The following is a description of how to add to, delete from and otherwise edit the three separate tables.



Open a Table:

Master Table	Select **Load Master Table.** Choose a file with extension .tab from the appropriate location on the hard drive. If successful the table will be loaded with the section groups (I Section, Rect Hollow Section etc) listed.
Beam Table	Select **Load Beam Table.**

	Choose a file with extension .bms from the appropriate location on the hard drive. If successful the table will be loaded with the section groups (I Section, Rect Hollow Section etc) listed.
Column Table	Select **Load Column Table**. Choose a file with extension .col from the appropriate location on the hard drive. If successful the table will be loaded with the section groups (I Section, Rect Hollow Section etc) listed.

Save a Table:

Master Table	Select the **Save Master Table** command. From the Save dialog choose to save over an existing table or to a new table. ***NOTE:*** A change to an existing table will affect the analysis and design of any model that uses this table and the section/s changed. Be sure to re-analyze and design any model that uses the changed table.
Beam Table	Select the **Save Beam Table** command. From the Save dialog choose to save over an existing table or to a new table. ***NOTE:*** A change to an existing table will affect the optimization and design of any model that uses this table and the section/s changed. Be sure to re-analyze and design any model that uses the changed table.
Column Table	Select the **Save Column Table** command. From the Save dialog choose to save over an existing table or to a new table. ***NOTE:*** A change to an existing table will affect the column design of any model that uses this table and the section/s changed. Be sure to re-analyze and design any model that uses the changed table.

Add a Section:

Master Table	 To add a section to the Master table right click the mouse in the list of sizes and select Add from the pop-up menu, or click the **plus** button below the list of sections. In the dialog that is displayed (see The Master Steel Table section below) enter the new section properties. Required fields have a white background, optional/calculated values have a light grey background. Any optional value left blank will be calculated when a new section is added. ***NOTE:*** The new section is added to the end of the list for the currently selected shape. This will be modified in the future.
Beam Table	To Add a section to the Beam table expand the appropriate list of sections in the tree. Drag the desired section from the master table to the beam design table. Place the cursor on the section below the position the new section should be located.
Column	To Add a section to the Column table expand the appropriate list of sections and group in

Table	the tree. Drag the desired section from the master table to the column design table. Place the cursor on the section below the position the new section should be located.

Edit a Section:

Master Table	 To edit a section in the Master table right click the mouse in the list of sizes and select Edit from the pop-up menu, or click the **slash** button below the list of sections. In the dialog that is displayed (see The Master Steel Table section below) enter the new section properties. Required fields have a white background, optional/calculated values have a light grey background. Any optional value left blank will be calculated when a new section is added. ***NOTE:*** Section properties are not recalculated if left blank on an edit command. The user must provide each value when editing a section. This will be changed in the future.
Beam Table	Beam sections cannot be edited.
Column Table	Column sections cannot be edited.

Re-order Sections within a Table:

Master Table	Master Table cannot be reordered.
Beam Table	The order of the sections is critical to the design optimization.  To re-arrange the order of the s either drag and drop the beam sections in the list or use the Up and Down arrows to move the currently selected section up and down in the table.
Column Table	The order of the sections is critical to the design optimization.  To re-arrange the order of the sections either drag and drop the column sections in the list or use the Up and Down arrows to move the currently selected section up and down in the table.

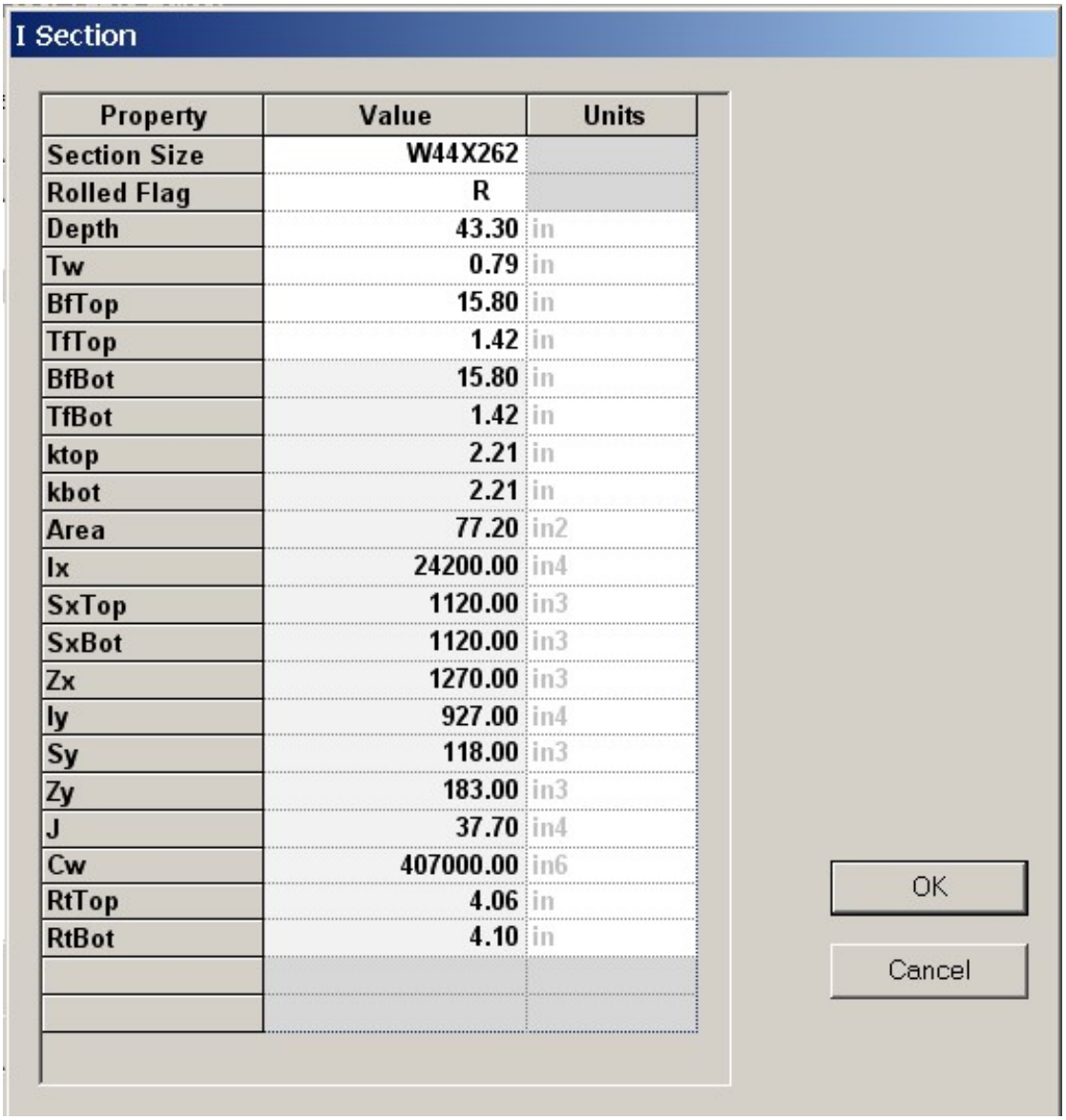
Delete a Section:

Master Table	 To remove a section from the master table list select the section and either select the X button below the list or right click and select delete from the pop-up menu. ***NOTE:*** Any section included in the Beam or Column table should be deleted as well as from the Master table to avoid an error during design in the RAM Structural System.
Beam Table	 To remove a section from the beam table select the section and either select the X button below the list or right click and select delete from the pop-up menu.
Column Table	 To remove a section from the column table list select the section and either select the X button below the list or right click and select delete from the pop-up menu.

The Master Steel Table

Sections in the master table can be added, edited or deleted by right clicking the mouse, in the list of sizes, to bring up the popup menu, or by using the buttons below the list. Selecting a size to be edited or added, a dialog appears displaying the properties for that shape. Required fields have a white background, optional/calculated values have a light grey background. Any optional value left blank will be calculated when a new section is added. Currently the user must provide each value when editing a section.

Section Add/Edit dialog



Index

I

L

M

O

P

R

S

T

U

V

Z